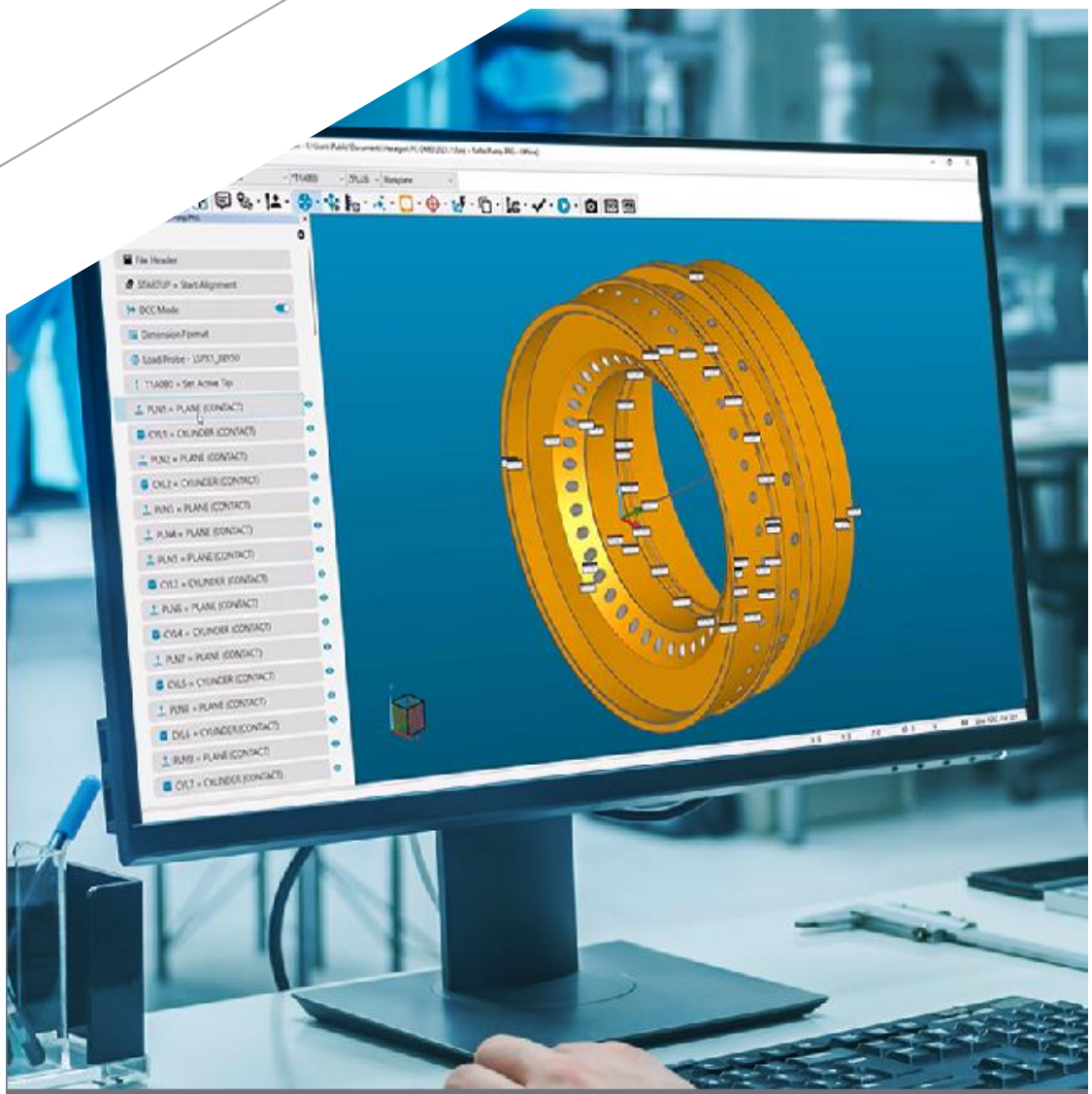
HEXAGON

PC-DMIS Core Manual

2026.2



Copyright and Licensing

This documentation is copyrighted. For more information, see the "Copyrights, Trademarks, and Legal Information.pdf" in the same folder as this documentation.

Table of Contents

PC-DMIS Overview 17
PC-DMIS Overview: Introduction 17
Organization of the PC-DMIS Core Documentation 18
Automating PC-DMIS 23
Software Configurations 23
Navigating the User Interface 37
Navigating the User Interface: Introduction 37
The User Interface 37
The Home Page 39
The Menu Bar 53
The Toolbar Area 57
The Graphic Display Window 57
The Status Bar 57
Dialog Box Description 58
Customizing the User Interface 67
Docking and Undocking User Interface Elements 78
Using Basic File Options 79
Using Basic File Options: Introduction 79
Creating New Measurement Routines 79
Creating a Package 81
Opening Existing Measurement Routines 81
Saving Measurement Routines 84
Working with Measurement Routine Archives 88
Performing File Operations 89
Changing the Language 117
Connecting to Teamcenter 117
Setting Printer Options 118
Printing the Graphic Display Window 119
Printing the Edit Window 120
Printing the Report Window 122
Closing or Quitting a Measurement Routine 140
Exiting the Software 141
Using Advanced File Options 143
Using Advanced File Options: Introduction 143
Importing CAD Data or Feature Data 143
Exporting CAD Data or Feature Data 196
Using Direct CAD Interfaces 216
Setting Import Options 216
Executing Measurement Routines 218
Setting Your Preferences 233

Setting Your Preferences: Introduction ... 233
Selecting Setup Options ... 235
Modifying Report and Motion Parameters ... 307
Setting Up the Edit Window ... 353
Setting Up the Probe Readouts Window ... 359
Setting Up Multiple Arms ... 379
Defining the Rotary Table ... 379
Setting Up Probe Changer Options ... 389
Managing Multiple Probe Changers ... 389
Loading the Active Probe ... 398
Examples of Working with Single and Multiple Probe Changers ... 398
Setting Up the Machine Interface ... 406
Generating a Debug File ... 407
Compensating for Temperature (Legacy) ... 409
Using Simplified Temperature Compensation ... 417
Temperature Gage ... 429
Editing Part Materials and Coefficients ... 434
Specifying Search Paths ... 437
Changing OpenGL Options ... 439
Using the Measurement Strategy Editor (MSE) ... 444
Setting Up Sensor Mapping ... 460
Defining Colors on Imported CAD Entities ... 460
Understanding File Locations ... 460
Understanding Data Files ... 464
Defining Default Environment ... 472
PC-DMIS FUSION Settings ... 473

Editing the CAD Display ... 477
Editing the CAD Display: Introduction ... 477
Setting Up the Screen View ... 479
Setting Up CAD and Graphics ... 483
Working with CAD Levels ... 509
Making CAD 3D by Using the Create Levels Window ... 514
Working with CAD Groups ... 514
Working with CAD Model Views ... 520
Managing and Positioning On-Screen Elements ... 523
Switching Between Curve and Surface Modes ... 524
Drawing Surfaces ... 527
Highlighting CAD Elements ... 527
Displaying Mouse Coordinates in Screen Counter Text ... 527
Repainting the Screen ... 528
Changing Screen Modes ... 528
Changing Screen Colors ... 535
Changing Trihedron Settings ... 540
Scaling the Drawing ... 543
Rotating the Drawing ... 544
Using the Rotation Widget ... 547

Changing Rotation and Other Motion Options 548
Showing and Hiding Graphics 554
Working with CAD GD&T Callouts 555
Working with Assemblies of Parts 573
Creating a CAD Section Cut 581
Working with a Machine Assembly 584
Viewing CAD Information 588
Verifying CAD Information 589
Using Screen Captures of the Graphic Display Window 593
Editing CAD Elements 594
Editing CAD Vectors 597
Selecting Features Using the Graphic Display Window 599
Identifying Features by Using Tooltips 605
Automatically Positioning Feature ID Labels 606
Editing Dimension Colors 607
Editing Feature Appearance 612
Showing, Animating, and Moving Path Lines 616
Graphical Hit Selection Method 626
Deleting CAD Elements 634
Deleting Features 635
Deleting Dimensions 636
Transforming a CAD Model 637
Working with CAD Coordinate Systems 642
Checking and Fixing Point Nominal Deviation 644
Modifying Display Symbols 647
Changing CAD Tolerances 651
Centering the Probe on the Screen 653
Viewing Clearance Planes 653

Editing a Measurement Routine 657

Editing a Measurement Routine: Introduction 657
Using Standard Edit Commands 658
Defining and Pasting Patterns of Features 660
Finding and Replacing Text 669
Editing Data Fields 677
Jumping to Specified Line Numbers 678
Jumping To and From Referenced Commands 679
Editing Nominal and Target Values 679
Overriding Guessed Features 680
Overriding Found Nominals 681
Marking Commands for Execution 687
Creating and Executing Marked Sets 694
Using Breakpoints 698
Setting Start Points 700
Using Bookmarks 701
Changing Fonts and Colors 702
Editing External Objects 702

Using Protected Mode to Protect Measurement Routines from Edits 702
Updating Nominals from CAD 704
Resetting Measured Values to Nominals 706
Estimate Execution Time 706
About Estimate Execution Time 706
Workflow 707
Calibrating Execution Time Parameters 707
Calibrating Execution Time Parameters on Your Machine 708
Estimating Execution Time 710
Using the Edit Window 713
Using the Edit Window: Introduction 713
Understanding Core Concepts 714
Working in Summary Mode 727
Working in Command Mode 748
Working in DMIS Mode 774
Copying and Pasting Parameters 775
Working with User-Defined Groups 779
Mini Routines 785
Using Other Windows, Editors, and Tools 805
Using Other Windows, Editors, and Tools: Overview 805
Using the Report Window 806
Using {FusionProductName} 806
Using the Basic Script Editor 807
Using the Form Editor 813
Viewing an Inspection Report 813
Using the Quick Start Interface 815
Using the Settings Window 850
Using the Preview Window 856
Using the Marked Sets Window 859
Using the Probe Readouts Window 859
Using the Status Window 862
Using the Dimension Colors Window (Dimension Colors Bar) 864
Using the Q-DAS Editor Window 866
ClearanceCube Settings 869
Using the Probe Toolbox 871
Selecting a Tip 873
Viewing Path Lines 873
Viewing CAD Import History 874
Using Toolbars 875
Using Toolbars: Introduction 875
File Operations Toolbar 876
Graphic Modes Toolbar 879
Graphic View Toolbar 881
Graphic Items Toolbar 883
Edit Window Toolbar 885

Window Layouts Toolbar 888
CAD Setup Toolbar 890
Measurement Strategy Toolbar 892
Auto Features Toolbar 892
Measured Features Toolbar 894
Constructed Features Toolbar 895
Pointcloud Toolbar 897
Mesh Toolbar 897
Dimension Toolbar 898
Alignment Toolbar 900
Settings Toolbar 901
Probe Mode Toolbar 904
Select Tip Toolbar 905
Active Arms Toolbar 906
Active Rotary Table Toolbar 907
ClearanceCube Toolbar 908
Wizards Toolbar 909
Build and Inspect Toolbar 914
QuickMeasure Toolbar 914
QuickCloud Toolbar 915
Quick Fixture Toolbar 915

Defining Hardware 917
Defining Hardware: Introduction 917
Defining Probes 917
Defining Machines 998
Inserting Quick Fixtures 1015
Defining Probe Changers 1025
Showing an Animated Probe Changer 1247
Protecting Probes from Probe Changer Collisions 1252
Working with Rotary Tables 1253
Configuring Advanced Mouse Devices 1256
Working with Environment Builder 1262

Creating Auto Features 1329
Creating Auto Features: Introduction 1329
Graphical Representation of Auto and QuickFeature Vectors 1330
Using the ArcReductionAngle Setting 1333
Quick Ways to Create Auto Features 1335
Creating QuickFeatures 1351
Using the Measurement Strategy Widget 1366
The Auto Feature Dialog Box 1369
Inserting Auto Features 1431
Auto Feature Field Definitions 1433
Setting Up Relative Measure (RMEAS) 1462

Creating Measured Features 1467
Creating Measured Features: Introduction 1467
Understanding the Command Format 1468

Inserting a Measured Feature ... 1476
Editing a Measured Feature ... 1480
Overriding a Guessed Measured Feature ... 1490
Creating Measured Feature Sets ... 1490
Constructing New Features from Existing Features ... 1493
Constructing New Features from Existing Features: Introduction ... 1493
Understanding the General Command Format ... 1496
Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions ... 1498
Specifying Feature Theoreticals ... 1503
Constructing a Point Feature ... 1505
Constructing a Line Feature ... 1539
Constructing a Plane Feature ... 1573
Constructing a Circle Feature ... 1610
Constructing an Ellipse Feature ... 1650
Constructing a Round Slot Feature ... 1662
Constructing a Square Slot Feature ... 1677
Constructing a Curve ... 1695
Constructing a Cylinder Feature ... 1700
Constructing a Cone Feature ... 1718
Constructing a Sphere Feature ... 1734
Constructing a Surface ... 1750
Constructing a Set of Features ... 1761
Constructing a Filter Set ... 1766
Constructing an Adjusted Filter ... 1781
Constructing a Width Feature ... 1784
Creating Generic Features ... 1789
Creating Generic Features: Introduction ... 1789
Generic Feature Command Format ... 1789
Creating a Generic Feature ... 1790
Creating a ReadPoint from the Probe's Position ... 1795
Creating and Using Alignments ... 1797
Creating and Using Alignments: Introduction ... 1797
Alignment Overview ... 1798
Alignment Command Format ... 1800
Description of the Alignment Utilities Dialog Box ... 1803
Creating a 3-2-1 Alignment ... 1815
About QuickAlign ... 1817
Creating an Iterative Alignment ... 1827
Creating a Best Fit Alignment ... 1838
Saving an Alignment ... 1858
Recalling an Existing Alignment ... 1860
Using an Alignment Inside Loops or Branches ... 1863
Equating an Alignment ... 1865
Equating CAD to Measured Part Data ... 1870
Performing a Leapfrog Operation ... 1870
Changing Alignment Nominal Values ... 1875

Updating Dependent Commands When the Alignment Changes 1876
Using Gages 1881
2D Radius Gage 1881
Caliper Gage 1881
Temperature Gage 1881
Thickness Gage 1881
Using Legacy Dimensions 1889
Using Legacy Dimensions: Introduction 1889
ISO Dimension Tolerances 1890
Reporting Dimensions for Measured Points 1890
Dimension Command Format 1890
Printing Dimensions to the Inspection Report 1893
Common Dimension Dialog Box Options 1894
Dimensioning Angle 1903
Dimensioning Angularity 1912
Dimensioning Circularity 1915
Dimensioning Coaxiality 1917
Dimensioning Concentricity 1919
Dimensioning Cylindricity 1920
Dimensioning Distance 1922
Dimensioning Flatness 1930
Dimensioning Location 1932
Dimensioning Parallelism 1944
Dimensioning Perpendicularity 1948
Dimensioning Position 1950
Dimensioning Position In-Between 1966
Dimensioning Profile - Line or Surface 1967
Dimensioning Runout - Circular or Total 1977
Dimensioning Straightness 1981
Dimensioning Symmetry 1982
Dimensioning through Keyboard Input 1985
Dimensioning Variables 1987
Using Geometric Tolerances 1989
Introduction to Geometric Tolerances and Feature Control Frames 1989
Structuring your Measurement Routine for Geometric Tolerances 1996
Defining and Using Datums 2002
Defining Geometric Tolerances and Controlling Reporting 2018
How PC-DMIS Solves and Uses Datums 2156
Feature Types With and Without Surface Data 2213
Evaluating Size with the Geometric Tolerance Command 2220
Deriving the Toleranced Feature 2227
Simultaneous Tolerances 2242
Outputting Results from Geometric Tolerances 2246
Migration from Prior Versions of PC-DMIS 2250
Dimensioning Size 2255
Using GD&T Selection Modes to Create FCFs 2266

Troubleshooting Error Messages and Warnings 2279
Scanning Your Part 2307
Scanning Your Part: Introduction 2307
Components of a Scan 2311
Working with CAD Surfaces 2311
Common Functions of the Scan Dialog Box 2312
Common Functions of the BASICSCAN Dialog Box 2362
Additional Manual Scan Options 2374
Inserting Move Commands 2375
Inserting Move Commands: Introduction 2375
Inserting a Move Point Command 2376
Using a Moveset Command 2382
Inserting a Move Increment Command 2386
Inserting a Move Circular Command 2387
Inserting Clearance Moves with Collision Detection 2388
Inserting a Move Sync Command 2388
Inserting a Move Sweep Command 2389
Inserting a Move All Command 2390
Inserting a Move Exclusion Zone Command 2392
Inserting a Move Rotary Table Command 2395
Inserting a Clear Point Move 2396
Inserting a Clearance Plane 2396
Inserting a Move Clear Plane Command 2396
Using the ClearanceCube 2397
Inserting Clearance Moves Automatically 2412
Branching by Using Flow Control 2421
Branching by Using Flow Control: Introduction 2421
Using Control Pairs 2422
Using Generic Loops 2433
Using Labels 2440
Reset Loop 2443
Branching on an Error 2443
Branching with Subroutines 2447
Ending a Measurement Routine 2465
Tracking Statistical Data 2467
Tracking Statistical Data: Introduction 2467
Using Tracefields 2467
Sending Statistics to a Q-DAS Database 2478
Sending Current Stats to a File 2480
Using the Statistics Options Dialog Box 2481
Sending Data to a Local DataPage+ Database 2494
Sending Statistics to an XML file using DataPageStats.exe 2496
Sending Statistics to an XML File using XMLSTATS/ON 2496
Table of Supported Methods 2499
Reporting Measurement Results 2501

Reporting Measurement Results: Overview 2501
Reporting Tutorials 2502
Working with Legacy (HyperView) Reports 2503
Report Generation Sequence 2504
About the Report Window 2506
Using Standard Reports 2527
Viewing a Legacy Text Report 2531
Reported Tolerance Zones for Form Dimensions 2532
Rearranging Report Pages 2533
Modifying Standard Templates 2534
Using .DAT Files in Generated Reports 2535
Modifying the Report's Header 2535
Changing the Report's Text Colors 2540
Adding Notes to a Report 2540
Adding Watermarks to Certified Measurement Routine Reports 116
About the Reporting and Form Editors 2544
Creating Templates 2687
Creating Forms 2732
Creating Custom Reports 2746
About Report Expressions 2768
Using PC-DMIS ActiveX Controls 2853
Tutorial - Using Scripting to Enhance your Report 2859
Working with PDF 3D Files 2869
QIF Report 2871

Inserting Report Commands 2881
Inserting Report Commands: Introduction 2881
Inserting Commands Related to the Analysis Window 2882
Inserting a 2D Plot Command 2895
Inserting Dimension Info Boxes 2904
Inserting Point Info Boxes 2915
Inserting Programmer Comments 2923
Embedding Reports or Report Templates in a Measurement Routine 2940
Inserting External Objects 2945
Inserting a Print Command 2945
Inserting a Form Feed Command 2953
Working with Viewsets 2953
Inserting a FORM Command 2957
Inserting Screen Captures 2960
Inserting Snapshots 2961

Using File Input / Output 2965
Using File Input / Output: Introduction 2965
Understanding Basic File I/O Concepts 2966
Using the File I/O Dialog Box 2967
Opening a File for Reading or Writing 2968
Closing an Opened File after Reading or Writing 2970
Reading a Character from a File 2971

Reading a Line from a File 2974
Reading a Block of Text from a File 2980
Reading Text up to a Delimiter 2983
Writing a Character to a File 2986
Writing a Line to a File 2989
Writing a Block of Text to a File 2991
Positioning a File Pointer at the Beginning of a File 2993
Saving a File Pointer's Current Position 2995
Recalling a Saved File Pointer's Position 2996
Copying a File 2998
Moving a File 3001
Deleting a File 3002
Checking for a File's Existence 3004
Displaying a File Dialog Box 3006
Checking for the End of a File or the End of a Line 3007

Using Expressions and Variables 3009
Using Expressions and Variables: Introduction 3009
Using Expressions in a Measurement Routine 3009
Creating Expressions with the Expression Builder 3011
Using Variables with Expressions 3017
Understanding Expression Components 3018
Accessing a Report's Object Properties 3098
Accessing Information from a Constructed Scan Minimum Circle 3101

Adding External Elements 3103
Adding External Elements: Introduction 3103
Inserting an External Command 3103
Inserting BASIC Scripts 3107
Attaching an External Measurement Routine 3111
Inserting External Objects 3113

Using Multiple Arm Mode 3121
Using Multiple Arm Mode: Introduction 3121
Setting Up a Multiple Arm CMM 3122
Creating a Measurement Routine Using Multiple Arm Mode 3139
Running the Arm1 Measurement Routine on Arm2 3145
Dialog and Message Boxes in Multiple Arm Mode 3146
Dual Arms with Wrists Calibration Example 3147

Navigating and Displaying Multiple Windows 3179
Navigating and Displaying Multiple Windows: Introduction 3179
Switching Between Windows of Open Measurement Routines 3179
Arranging Open Windows 3180
Giving Open Windows the Routine Focus 3180

Working in Offline Mode 3183
Working in Offline Mode: Introduction 3183
PC-DMIS Offline Prerequisites 3184
Offline Probes 3189

Setting Probe Depth 3190
Measuring Features Offline 3193
Executing and Debugging Measurement Routines Offline 3196
Using Inspection Plans in PC-DMIS 3197
Using Inspection Plans in PC-DMIS: Introduction 3197
Importing an Inspection Plan from Planner 187
Mapping Features to Sensors 3198
Importing an Inspection Plan from Legacy PC-DMIS Planner 188
Updating Inspection Plans with the Change Manager 3208
Optimizing the Path 3215
Void Detection 3225
Inserting Clearance Moves Automatically 3227
Using a Wrist Device 3229
Using a Wrist Device: Introduction 3229
Wrist Installation 3229
Tip Calibration 3230
Calibrate the Unit for Infinite Wrist Devices 3234
Qualification Check 3237
Home the Unit 3238
Using the Wrist in a Measurement Routine 3239
Calibrating and Using a CJoint Device 3241
Working in Operator Mode 3245
Working in Operator Mode: Introduction 3245
File Open Options 3245
Operator Mode Menu Options 3246
Using the Marked Sets Window in Operator Mode 3248
Using Shortcut Keys and Shortcut Menus 3251
Using Shortcut Keys and Shortcut Menus: Introduction 3251
Shortcut Keys Reference 3251
Shortcut Menus Reference 3259
Modifying Setting Entries 3293
Glossary 3295
Index 3321

PC-DMIS Overview

PC-DMIS Overview: Introduction

PC-DMIS is a full-featured, geometric measurement package. It translates the high-level commands required to measure parts into the detailed steps necessary to drive a Coordinate Measuring Machine (CMM). PC-DMIS incorporates the Microsoft Windows interface to create, as well as execute, measurement routines. You can begin the measurement process by taking advantage of the drop-down menus, dialog boxes, and icons. The versatility of PC-DMIS's user interface also provides a way to customize the software to meet individual specifications.

The primary goals driving the design of PC-DMIS for Windows are flexibility and user friendliness. To meet these goals, PC-DMIS provides a flexible environment allowing for real-time error correction as opposed to a long debugging process, characteristic of some other geometric software packages. It is dimension-oriented to reduce the need to analyze and interpret the results of CMM measurements. The techniques used for measurement routine programming on a CMM are straightforward and, with practice, systematic. This documentation does not attempt to completely describe the dimensional inspection process. If additional information is needed on the fundamentals of dimensional inspections, please contact your software representative.

To see what's new in this version, from the PC-DMIS Home page, choose **What's New**.

The main topics in this chapter include:

- Organization of Manual
- Automating PC-DMIS
- Software Configurations

This documentation contains the core documentation needed to operate PC-DMIS. However, due to PC-DMIS's modular nature, additional documentation may be installed on your computer system. Click the **Contents** tab on your help file to view the other files installed.

Organization of the PC-DMIS Core Documentation

This documentation is divided into numerous main chapters and some supplemental appendices. To give you an idea of the documentation layout, the following list provides a brief description of each chapter.

To get help with how to search the Help documentation, see the "Using the Offline Help" topic in the PC-DMIS Help Center.

- "PC-DMIS: An Overview" - This is the current chapter you are reading. It gives a brief overview of PC-DMIS and the contents of the PC-DMIS documentation.
- "Navigating the User Interface" - This chapter describes the PC-DMIS user interface and how to customize it to suit your needs.
- "Using Basic File Options" - This chapter discusses the basic file operations that you can use to start a new measurement routine, save it, and exit PC-DMIS, and so on.
- "Using Advanced File Options" - This chapter discusses the more advanced file operations such as how to import and export CAD files and execute your measurement routine.
- "Setting Your Preferences" - This chapter describes the setup options and measurement routine parameters.
- "Editing the CAD Display" - This chapter describes how to use the Graphic Display window to edit the display of your CAD file.
- "Editing a Measurement Routine" - This chapter describes the Edit window and how to edit your measurement routine.
- "Using the Edit Window" - This chapter describes how to use the PC-DMIS built-in editor to create, debug, edit, and execute your measurement routine.
- "Using Other Windows, Editors, and Tools" - This chapter discusses additional windows, editors, and tools.
- "Using Toolbars" - This chapter discusses the main toolbars and toolbar icons.
- "Defining Hardware" - This chapter describes how to define probes, machines, quick fixtures, and tip calibration.
- "Creating Auto Features" - This chapter describes how to create single-click automatic features.
- "Creating Measured Features" - This chapter describes how to create measured features.

- "Constructing New Features from Existing Features" - This chapter describes how to construct features from other features already in your measurement routine.
- "Creating Generic Features" - This chapter describes how to create generic features and the ReadPoint command.
- "Creating and Using Alignments" - This chapter describes how to create, save, and recall alignments.
- "Using Gages" - This chapter describes how to use quick-check tools to measure various system and feature characteristics.
- "Using Legacy Dimensions" - This chapter describes the legacy dimensions.
- "Using Geometric Tolerances" - This chapter describes how to dimension your features with geometric tolerances using Feature Control Frames and symbols from the GD&T standard.
- "Scanning Your Part" - This chapter describes basic and advanced scan options.
- "Inserting Move Commands" - This chapter discusses various commands that can control machine or probe motion.
- "Branching by Using Flow Control" - This chapter covers commands that control the flow of your measurement routine.
- "Tracking Statistical Data" - This chapter explains how to track and use statistics in your measurement routine.
- "Reporting Measurement Results" - This chapter walks you through reports, the Report window, report and label templates, custom reports, and forms.
- "Inserting Report Commands" - This chapter discusses the report commands that you can insert into your measurement routine.
- "Using File Input / Output" - This chapter explains how to work with file input and output in your measurement routines to open files to read from them, write to them, and other file operations.
- "Using Expressions and Variables" - This chapter describes how to create expressions and how to assign expression results to variables.
- "Adding External Elements" - This chapter covers how to use external applications, scripts, measurement routines, and other objects in your measurement routine to further enhance its capability.
- "Using Multiple Arm Mode" - This chapter describes how to use PC-DMIS with dual-arm CMMs.
- "Navigating and Displaying Multiple Windows" - This chapter describes how to navigate between and view open windows.

In addition to the above sections, the PC-DMIS documentation provides appendices that document the following:

- "Working in Offline Mode"
- "Using Inspection Plans in PC-DMIS"
- "Using a Wrist Device"
- "Working in Operator Mode"
- "Using Shortcut Keys and Shortcut Menus"
- "Modifying Setting Entries"

Documentation Conventions

The following conventions are used throughout the documentation:

Boldface Text

The boldface font is used when referring to:

- Dialog box elements
- Dialog box titles
- Command buttons
- Menus and menu items
- Toolbars
- Toolbar icons
- Text that you must type

Some examples:

- Select the **File | New** menu item.
- Access the **Open** dialog box.
- Click the **Save** button.

Boldface is also used on **Notes**, **Warnings**, **Hints**, and **Examples**.

When referring to menus and menu items inside procedures and other topics, the documentation directs you to a menu in this fashion:

Menu | Submenu or **Menu Item | Menu Item**

However, since you can customize your menu locations and names, this documentation only displays the default locations for menu items.

Italicized Text

Italicized text is primarily used for *emphasis*. For example, "If…the probe hits an obstruction, it will *not* automatically stop".

Italicized text may also be used for book or manual titles and arguments in programming.

Uppercase Text

Uppercase text is generally used when referring to:

- Acronyms (such as VDAFS, DMIS)
- Time (such as 9 A.M. and 4 P.M.)
- Device names (such as LPT1, or COMM PORT 2)

In those instances where elements of computer programming are displayed, the following items also use uppercase:

- Control classes
- Data formats
- Environment variables
- Handles
- Hooks
- Indexes
- Macros
- Statements
- Structures
- System Commands
- Values

Underlined Text

Underline is used for emphasis when appropriate, but usually emphasis uses *italicized text*.

Hyperlinks are also underlined.

Numbered Lists

Ordered lists show a series of steps in instructions and procedures and other sequential lists. They are also used to identify elements in diagrams and images.

For example,

1. Choose **File**.
2. Select **New**.
3. Fill in the **New Measurement Routine** dialog box.

Note Boxes

Throughout the documentation, you'll find special boxes that offset the documentation. Here is what they mean:

This indicates a note.

This indicates an important note.

This indicates a hint.

This indicates an example.

This indicates a caution. Failure to follow a caution may result in data loss or hardware damage.

This indicates a warning. Failure to follow a warning may result in bodily harm.

Automating PC-DMIS

PC-DMIS supports full automation from any third-party software. For example, you can create your own customized application and use automation commands to launch and use PC-DMIS via that application.

For complete information on the automation commands that control PC-DMIS, and on supported BASIC language commands, see your "PC-DMIS Basic Language" documentation and "Automation Objects" documentation for additional information.

Software Configurations

If you have purchased the necessary modules, and your license is properly configured, you can launch PC-DMIS in different configurations. To do this, you can add these switches to the shortcut command line, batch file, or command prompt window that launches the application.

Available Software Configuration Switches

/f - You can use this switch for Offline mode.

/u - You can use this switch for user privilege level when you log onto the system as an administrator.

/o - You can use this switch for Operator mode.

/r - You can use this switch for Reverse Axes mode on Arm2 of a multiple arm system.

/p - You can use this switch for Pro mode. You cannot use this switch with the **/c** switch.

/c - You can use this switch for CAD mode. You cannot use this switch with the **/p** switch.

/d - You can use this switch for Debug mode for the current session.

/envFile - You can use this switch to run PC-DMIS with the selected environment or with the default environment.

/envFile: <name of the environment> - You can use this switch to run PC-DMIS with the specified environment.

/nc0 - You can use this switch to run PC-DMIS without the CNC functionality.

/5unique - You can use this switch to run PC-DMIS without this functionality.

/cmt - You can use this switch to run PC-DMIS without this functionality.

/dccscanning - You can use this switch to run PC-DMIS without this functionality.

/displaycad - You can use this switch to run PC-DMIS without this functionality.

/f - You can use this switch to run PC-DMIS in Offline mode.

/fullcrashdump - You can use this switch to have PC-DMIS generate more detailed crash report files than normal. These more-detailed crash report files can help Hexagon programmers track down what caused the product to hang or stop responding. A full crash dump also includes screen captures. If you don't use this switch, the software uses the standard crash reporting. Standard crash reporting does not include screen captures.

/laserinterface - This switch allows you to choose the laser stripe controller and its associated functionality. You can add the specific controller after the colon:

- /laserinterface:cms
- /laserinterface:mercury
- /laserinterface:romerintegrated

/laserprobe - You can use this switch to run PC-DMIS without this functionality.

/masterslave - You can use this switch to run PC-DMIS without this functionality.

/nc0 - You can use this switch to run PC-DMIS without the CNC functionality.

/nocontactprobe - You can use this switch to run PC-DMIS without this functionality.

/nocrashdump - You can use this switch to disable crash report files in PC-DMIS.

/nomigrate - You can use this switch to prevent the migration of entries and data files from a previous version. This switch can provide you with a clean "out of the box" experience. Only use this switch if you are sure you don't want to transfer key settings and calibration data from a previous version. To use this switch, add it to whatever shortcut or command you use that runs PC-DMIS for the first time as an administrator. When you do this, PC-DMIS does not copy your settings or data files from the most recent version.

/o - You can use this switch to run PC-DMIS in Operator mode.

/p - You can use this switch to run PC-DMIS in Pro mode. You cannot use this switch with the **/c** switch.

/r - You can use this switch to enable Reverse Axes mode on Arm2 of a multiple arm system.

/remotepanel - You can use this switch to run PC-DMIS without this functionality.

/rotarytable - You can use this switch to run PC-DMIS without this functionality.

/sheetmetal - You can use this switch to run PC-DMIS without this functionality.

/statsoutput - You can use this switch to run PC-DMIS without this functionality.

/systemv - You can use this switch to run PC-DMIS without this functionality.

/toolchanger - You can use this switch to run PC-DMIS without this functionality.

/u - You can use this switch to force PC-DMIS to run with normal user privileges even if you log onto the computer as the administrator.

/vision - You can use this switch to run PC-DMIS without this functionality.

/wrist - You can use this switch to run PC-DMIS without this functionality.

/DCILOAD:<file> - You can use this switch to load a file using Direct CAD Interfaces (DCI) where <file> indicates the complete path and file name of the model.

/DCTLOAD:<file> - You can use this switch to load a supplied part model using the direct CAD translators where <file> indicates the complete path and file name of the model.

/ISPROE - You can use this switch to specify that the software uses the .prt file in Creo (both NX and Creo use the same .prt file name extension). This follows the **/DCILOAD** switch.

/NC0 - You can use this switch to run PC-DMIS in normal Offline mode when you run PC-DMIS NC. The CNC licenses option is defined as greater than 0 in your portlock.

DCI/DCT Examples

- This command line launches PC-DMIS and loads an .igs CAD model using PC-DMIS Direct CAD Translators (DCT):

  C:\Program Files\Hexagon\<version>\Pcdlrn.exe New.prg /DCILOAD:d:\partmodels\hexblock_wireframe_surface.igs

- This command line launches PC-DMIS and loads a .prt CAD model from the NX Direct CAD Interface (DCI):

  C:\Program Files\Hexagon\<version>\Pcdlrn.exe New.prg /DCILOAD:d:\partmodels\hexblock.prt

- This command line launches PC-DMIS and loads a .prt CAD model as a Creo DCI:

  C:\Program Files\Hexagon\<version>\Pcdlrn.exe New.prg /DCILOAD:d:\partmodels\proe.prt /ISPROE

Routine of Shortcut Modification with Switches

1. Access the directory where the PC-DMIS executable shortcut is located. This directory is used:

   C:\ProgramData\Microsoft\Windows\Start Menu\Programs\2026.2\

2. Right-click in the directory and select **New | Shortcut**.

3. Follow the on-screen instructions to have the shortcut point to where the PC-DMIS executable (pcdlrn.exe) is located (wherever you installed it).

4. Add any of the command line switches (or arguments), following the shortcut's full path to the executable:

    So, if you want to run PC-DMIS *pro* in *Offline* mode, the target location would look something like this:

    "*C:\Program Files\Hexagon\<version>\PCDLRN.exe" /p /f*

    Where <version> is your version of PC-DMIS.

You can also tell PC-DMIS to automatically open a specific measurement routine (or measurement routines) by appending a pathway string pointing to a measurement routine as a command line argument.

- When you specify a measurement routine to load, however, you do not need the ***/*** character required for the other arguments discussed above.
- To open more than one measurement routine, type a space between each additional path.
- If your file name contains spaces, put the path within quotation marks.

    For example, if you wanted to launch two measurement routines named *test.prg* and *test2.prg* in Offline mode from a directory called "My Measurement Routines", the target locations might look something like this:

    "*C:\Program Files\Hexagon\<version>\PCDLRN.exe" /f "d:\my measurement routines\test.prg" "d:\my measurement routines\test2.prg*"

The most common modes are Offline and Online. These two distinct products are designed to meet particular needs. They can be used together to create a complete measurement routine, measurement analysis, and reverse engineering system.

These configurations and other configurations are explained below in greater detail.

Comparison of Offline and Online Modes

Since both the Online and Offline modes of PC-DMIS share the same features and functions, this documentation describes the use of both products. However, in some instances, a particular feature of the software in Online mode does not apply to the software in Offline mode. These cases have been noted where appropriate.

Offline Measurement Routine Programming

You can use PC-DMIS in Offline mode to develop and debug measurement routines away from the CMM by editing a part created in Online mode, by importing a CAD input file, or through a DMIS measurement routine. You can then directly execute the measurement routine in Online mode, or export (post processed) into DMIS or one of several vendor-specific formats. You cannot directly drive a CMM in Offline mode.

To start PC-DMIS in Offline mode, do the following:

1. Turn on your computer and all its peripherals, and log on to your computer.
2. Double-click with the left mouse button on the **Offline** icon in the PC-DMIS Program Group.

3. Use PC-DMIS to program your part's measurement routines. If you are new to PC-DMIS, see "A Simple Tutorial" in the PC-DMIS CMM documentation for help.

The "Working in Offline Mode" appendix describes the particulars of working with PC-DMIS in Offline mode.

Online Measurement Routine Programming

You can use PC-DMIS in Online mode to execute measurement routines, quickly inspect parts (or sections of parts), and develop measurement routines directly on the CMM. PC-DMIS in Online mode does not function unless it is connected to a CMM or other measuring device. Offline programming techniques work in Online mode.

To start PC-DMIS in Online mode, see the "Getting Started" chapter in the PC-DMIS CMM documentation. It covers the CMM startup and homing processes.

PC-DMIS CMM

PC-DMIS CMM works with your Coordinate Measuring Machine (CMM) to inspect your parts. PC-DMIS was the first CMM software to:

- Use CAD models in the inspection process.
- Directly link CAD systems and measurement software through its Direct CAD interface (DCI) technology.
- Implement a full set of sheet metal measurement routines tailored for the automotive industry.
- Digitally simulate measurements in an offline virtual CMM environment.
- Easily align complex, contoured parts using our breakthrough iterative alignment technology.
- And many, many more.

PC-DMIS CMM comes in various standard configurations. Each is carefully tailored to meet the needs of a specific group of customers. Also, PC-DMIS offers a wide range of optional modules for doing specific tasks. This means that users can fine-tune the software to meet specific needs.

PC-DMIS Pro – The baseline PC-DMIS CMM package offers customers who don’t need to integrate CAD into their inspection process a powerful, easy-to-use metrology software package. PC-DMIS Pro, with its built-in **Quick Start** routines, lets users get up and running on their CMMs with a minimum of fuss.

PC-DMIS CAD – Brings CAD to the inspection process. PC-DMIS CAD lets customers create inspection programs and evaluate measurement results taking full advantage of their CAD models. The software provides the tools you need to work with CAD files ranging from simple 2D drawings through complex 3D solid models.

PC-DMIS CAD++ – Enhances the capabilities of PC-DMIS CAD with sophisticated tools for high speed scanning, sheet metal measurement, part alignment and the like. PC-DMIS CAD++ makes measuring complex shapes easy.

Items that are specific to contact probes and CMMs are discussed in the PC-DMIS CMM documentation. Other information can be found in this PC-DMIS Core documentation.

PC-DMIS Vision

This special version of PC-DMIS allows you to use certain optical probing devices to measure features. If you've purchased this module, see your "PC-DMIS Vision" documentation for additional information.

PC-DMIS NC

This special version of PC-DMIS allows you to perform part measurements using Numeric Control (NC) machine tools. These machine tools are also referred to as CNC (Computer Numerical Control) machines. If you've purchased this module, see your "PC-DMIS NC" documentation for additional information.

PC-DMIS Laser

This module of PC-DMIS lets you use a laser probe to easily measure your parts by passing a laser stripe over the part to collect large amounts of points which are then interpreted and defined into features. If you've purchased this module, see your "PC-DMIS Laser" documentation for additional information.

PC-DMIS Portable

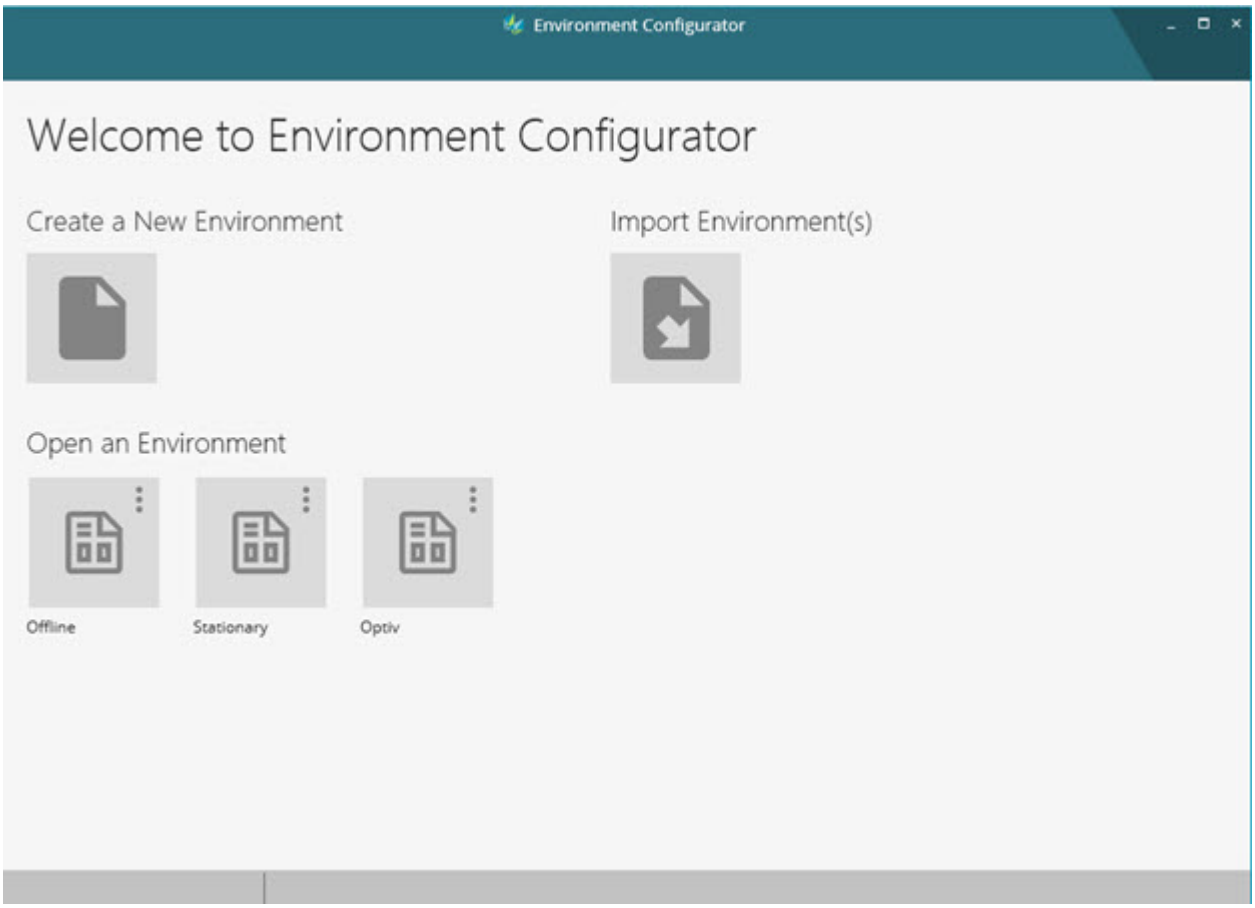
The PC-DMIS Portable configuration works with portable measuring devices to inspect your parts. Portable devices are manually operated measuring machines that are relatively easy to move to new locations due to their size and design. See the "PC-DMIS Portable" documentation for more information.

PC-DMIS Gear

PC-DMIS Gear is a software product that works in conjunction with PC-DMIS to allow you to quickly and easily measure any gear parts you have. If you've purchased this module, see the "PC-DMIS Gear" help file that gets installed with the PC-DMIS Gear application for more information.

We hope you will enjoy using PC-DMIS and find it a useful and powerful tool. If you have any suggestions or comments, please let us know. We are always looking for ways to improve our product.

Environment Configurator



You can use the Environment Configurator utility to define, open, and import PC-DMIS environments.

You can define an environment and select that environment before you open any measurement routine.

- To define an environment, select **Edit | Create/Edit Environment** from the Home page.
- To select an environment, select **Edit | Select Environment File** from the Home page.

About the Environment Configurator: Introduction

PC-DMIS generates an environment file automatically based on your current license configuration. The file contains this information:

- Interface type
- Non-Contact Sensor type

When you launch PC-DMIS, PC-DMIS uses this information to connect to the online device.

Hexagon now ships with offline license to support these interfaces:

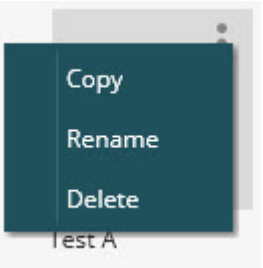
- All machine interface
- All Laser interface
- All Vision interface
- All Portable interface

The environment file allows you to configure offline PC-DMIS with a UI that closely matches your online environment.

The Menu

In the **Open an Environment** area, Environment tiles have their own menu that is accessible from the **More Actions** button located in the upper-right of each tile.

To show the menu, click the **More Actions** button. The menu shows a list of actions that you can do with the environment:



Copy - This menu item opens the **Copy Environment** dialog box to provide a name for the copied settings.

Rename - This menu item opens the **Rename Environment** dialog box to provide a new name.

Delete - This menu item deletes the environment from the view.

Opening a Measurement Routine in a Selected Environment

To open a measurement routine in a selected environment, follow these steps:

1. From the Home page, select **Edit | Select Environment File**. This menu shows all the items that you created on your computer.
2. Select the environment.
3. PC-DMIS shows a message to inform you to re-start the application to use the new environment. Click **OK** to re-start PC-DMIS.

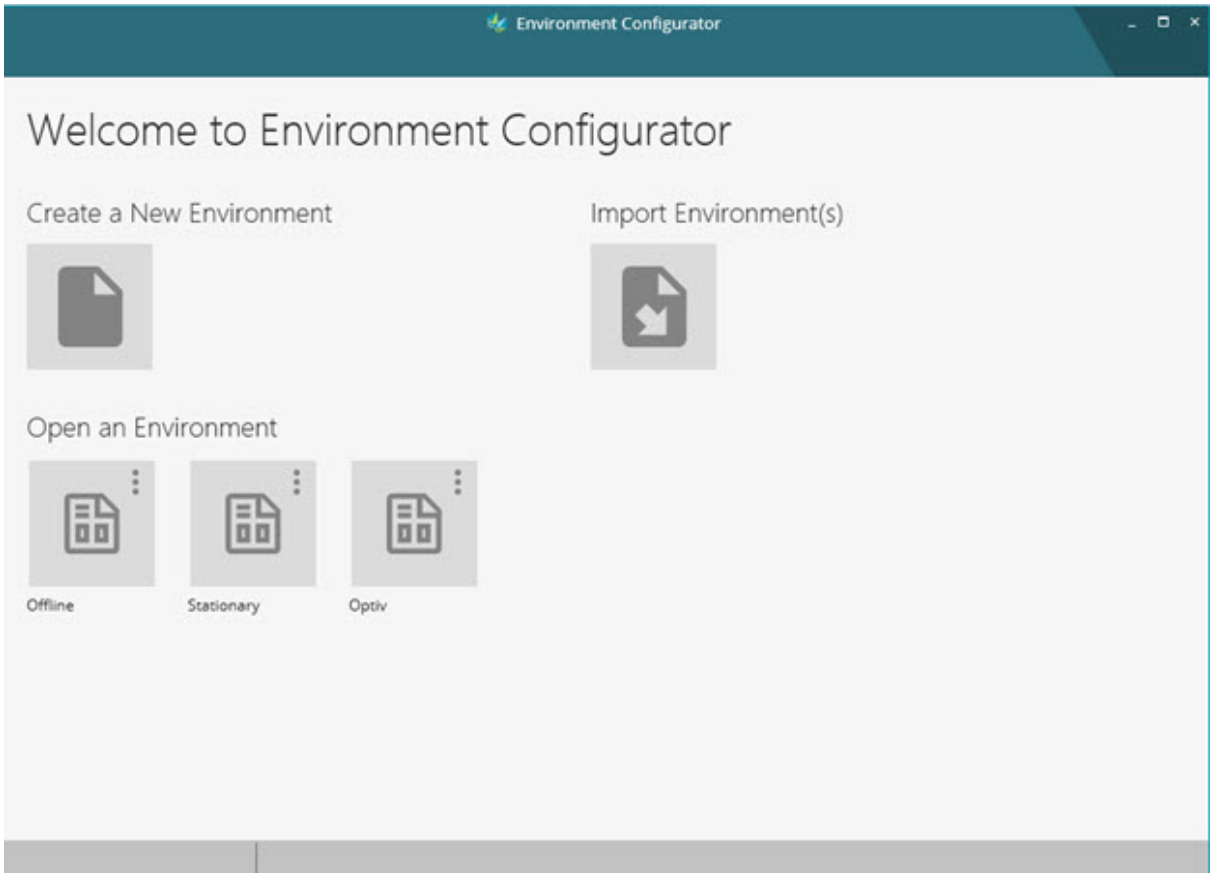
- For online stationary devices (CMM and Optiv), PC-DMIS only shows the environment that corresponds to the device. PC-DMIS selects it by default.
- For Portable devices, PC-DMIS shows all portable environments. You must select your environment.
- For offline, PC-DMIS shows Stationary, Optiv, and all Portable environment options. To select an environment, follow steps 1 through 3.

Creating and Editing an Environment

To create an environment, follow these steps:

You must run PC-DMIS as an administrator.

1. From the PC-DMIS Home page, select **Edit | Create/Edit Environment**. This opens the **Environment Configurator** dialog box.



Environment Configurator Dialog Box

   The dialog box has three areas that you can use to create a new environment, import an environment, or open an environment.

2. To create a new environment, click the **Create a New Environment** tile to open the **New Environment Name** dialog box.

3. From the **New Environment Name** dialog box, in the **Environment File** box, type a name.

4. Click **OK**. This opens the screen to define the hardware and PC-DMIS settings.

5. After you complete the definition of your new environment, click **Save**.

To edit or open an existing environment, follow these steps:

1. From the **Open Environment** area of the **Environment Configurator** dialog box, click a tile that you want to edit.

2. Make the changes in the **Hardware Settings** and **PC-DMIS Settings** tabs.

3. Click **OK**.

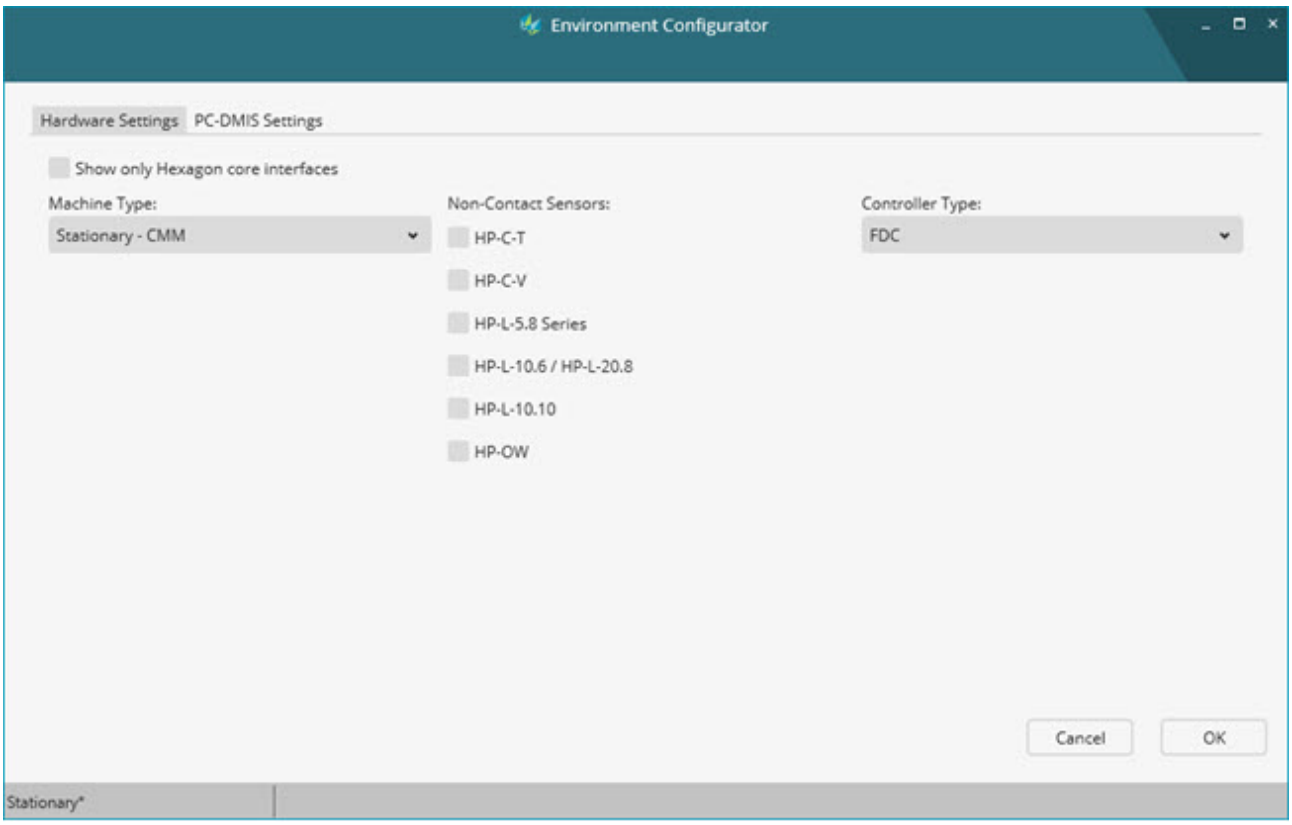
To import an existing environment that you generated from another online device, follow these steps:

You can use this option if you have a separate offline license that allows you to select an environment that corresponds to your machine.

1. From the **Environment Configurator** screen, click a tile that you want to edit under the **Import Environment(s)** area. This opens the **Select file(s)** dialog box.
2. From the **Select file(s)** dialog box, navigate to the desired folder and select an Environment Files (.json) file.
3. After you select a file, click **Open**. This adds the item under the **Open an Environment** area.
4. From the **Open an Environment** area, click a tile that you want to open. This opens the screen with the **Hardware Settings** and **PC-DMIS Settings** tabs.
5. From the **Machine Type** list on the **Hardware Settings** tab, select **Stationary - CMM** or **Stationary - Optiv**.
6. From the **Controller Type** list, select **Offline**.
7. Click **OK**. This adds the environment under the **Open an Environment** area.

Hardware Settings

From the **Hardware Settings** tab, define the **Controller** and **Sensor** settings.



Environment Configurator Dialog Box - Hardware Settings Tab

Show only Hexagon core interfaces - This check box filters the **Controller Type** list to display only those controllers that are manufactured by Hexagon.

Machine Type - This list lets you select the device type:

- Stationary - CMM
- Stationary - Optiv
- Portable

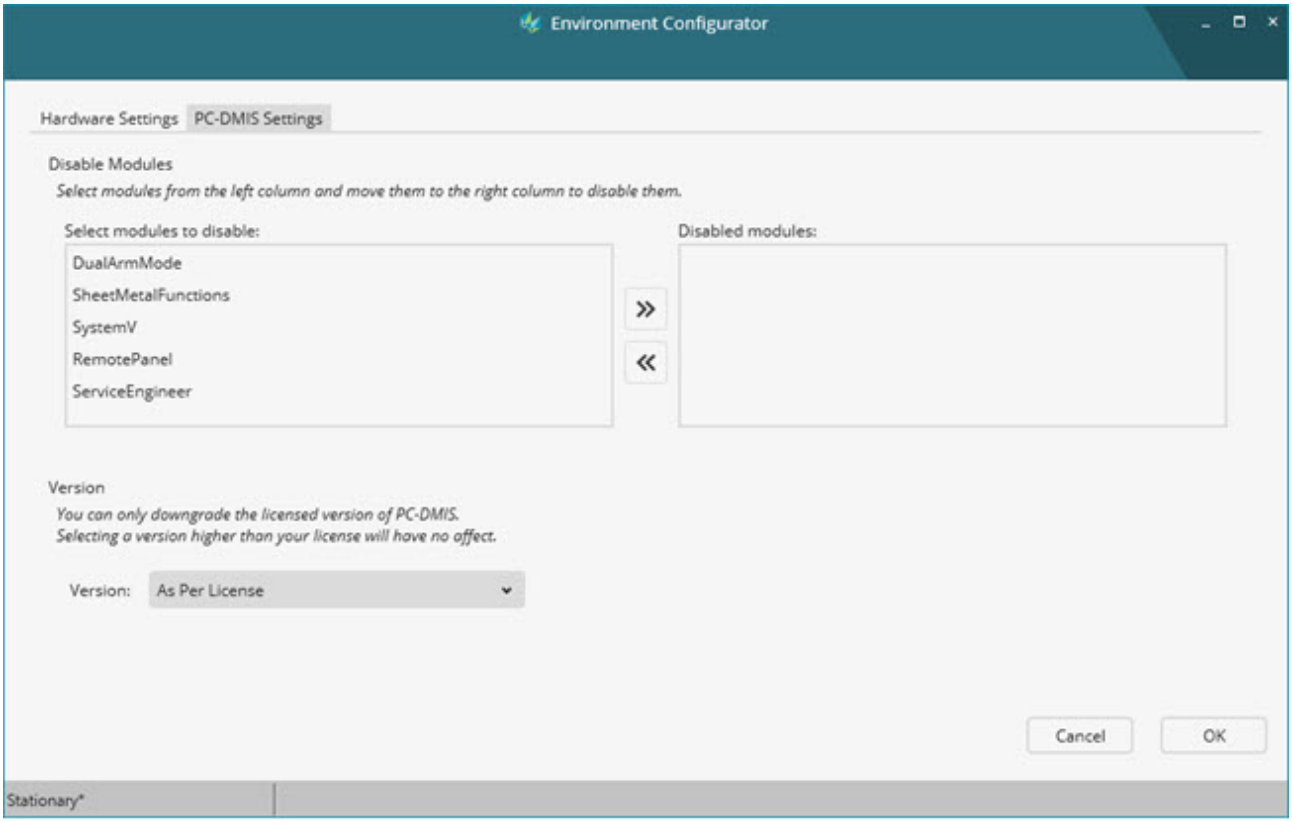
Based on what you select from the **Machine Type** list, PC-DMIS updates the list of **Non-Contact Sensors** and **Controller Type**.

When you select a combination of sensors that PC-DMIS doesn't support, PC-DMIS displays a message in the Environment Configurator's Status bar to indicate this.

PC-DMIS Settings

The **PC-DMIS Settings** tab lets you configure the options that you want to use to start PC-DMIS in a desired way. If you don't make any configuration changes, PC-DMIS starts with the default options.

With these settings, you can define **Startup Options** and **Version**.



Environment Configurator Dialog Box - PC-DMIS Settings Tab

Disable Modules

This area allows you to disable (and enable) application modules associated with your license.

You can disable license options that you don't need. To do this, follow these steps:

1. From the **Select modules to disable** list, select the modules that you want to disable.

   To select multiple modules, press Ctrl and then click more modules.

2. Click **Disable selected module(s)** to move the selected modules from the **Select modules to disable** list to the **Disabled modules** list.

To enable one or more modules, follow these steps:

1. From the **Disabled modules** list, select the modules that you want to enable.

   To select multiple modules, press Ctrl and then click more modules.

2. Click **Enable selected module(s)** to move the selected modules from the **Disabled modules** list to the **Select modules to disable** list.

Version

This area lets you select a specific level of PC-DMIS.

You cannot select a PC-DMIS level higher than your current license.

From the **Version** list, select a PC-DMIS license that you want to use to open your measurement routine.

The options available are **As Per License**, **Pro**, **Cad**, **Basic**, and **Premium**.

Navigating the User Interface

Navigating the User Interface: Introduction

This section introduces the main areas of the PC-DMIS user interface. This section also highlights ways to customize some of PC-DMIS's graphical options to best meet individual needs. (Additional information on these available options can be found in the "Editing the CAD Display: Introduction" section.)

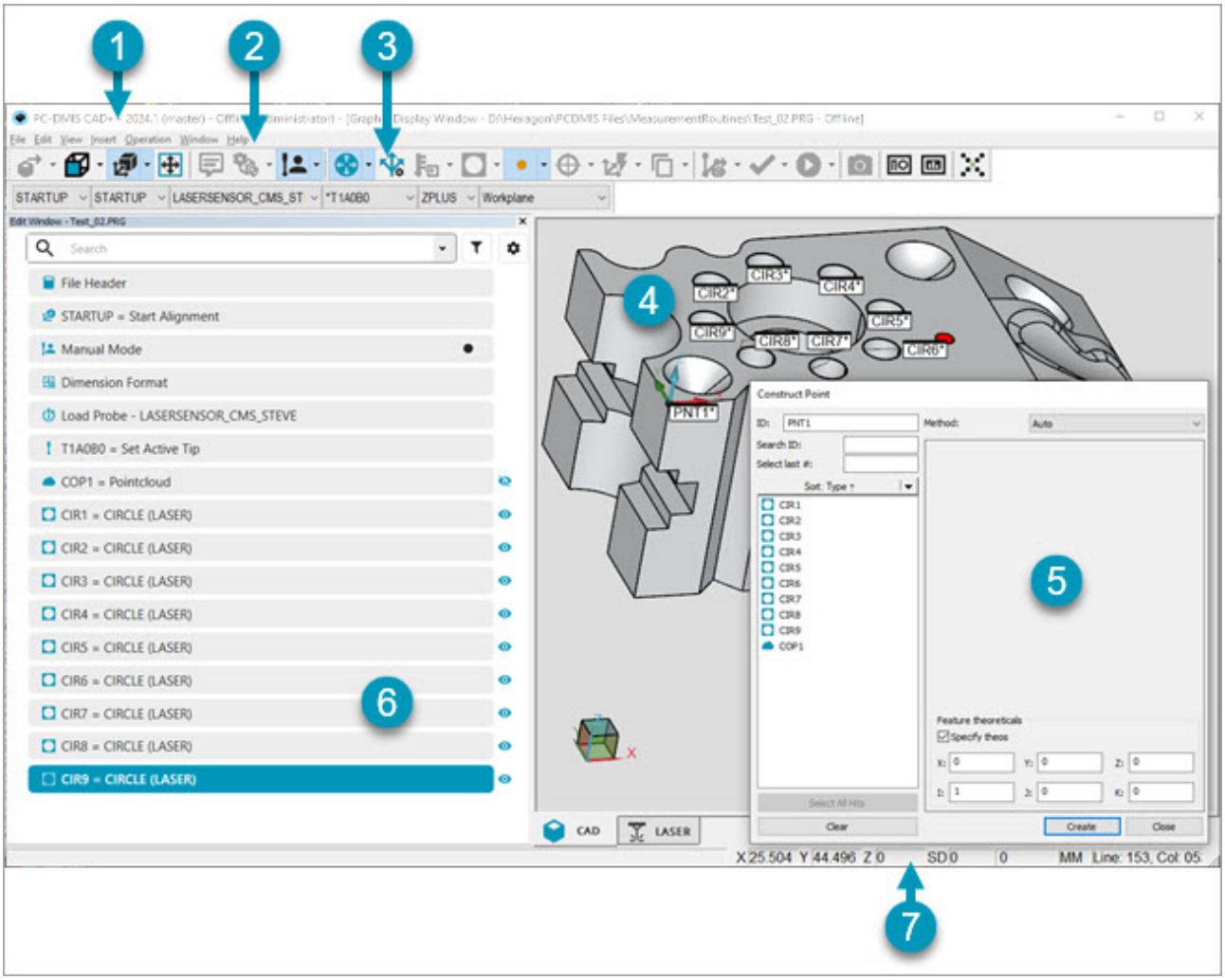
Each topic listed below illustrates a specific feature of the interface. The following are covered in this section:

- The User interface
- The Home Page
- The Menu Bar
- The Toolbar Area
- The Graphic Display Window
- The Status Bar
- Dialog Box Description
- Customizing the User Interface

PC-DMIS provides the flexibility for you to use the mouse or keyboard to enter commands or select options. PC-DMIS also provides shortcut menus and shortcut keys to frequently-used commands. See the "Using Shortcut Keys and Shortcut Menus: Introduction" section.

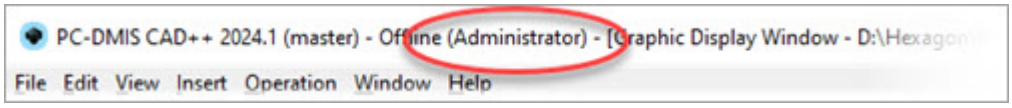
The User Interface

The PC-DMIS user interface is divided into several distinct areas, each of which fulfills a specific function or supplies needed information. A brief description of each screen area is listed below. (Additional information is located throughout the documentation as indicated.)



Example showing PC-DMIS user interface.

1. The title bar displays the PC-DMIS version information, its Offline or Online state, and if it's in administrator mode. If PC-DMIS is Online, it also displays the name of the machine.



2. The menu bar contains the menus and menu items available from the PC-DMIS application. For information, see "The Menu Bar".

3. The toolbar area contains toolbars with frequently used commands. You can right-click in the toolbar area to access these toolbars. See "The Toolbar Area".

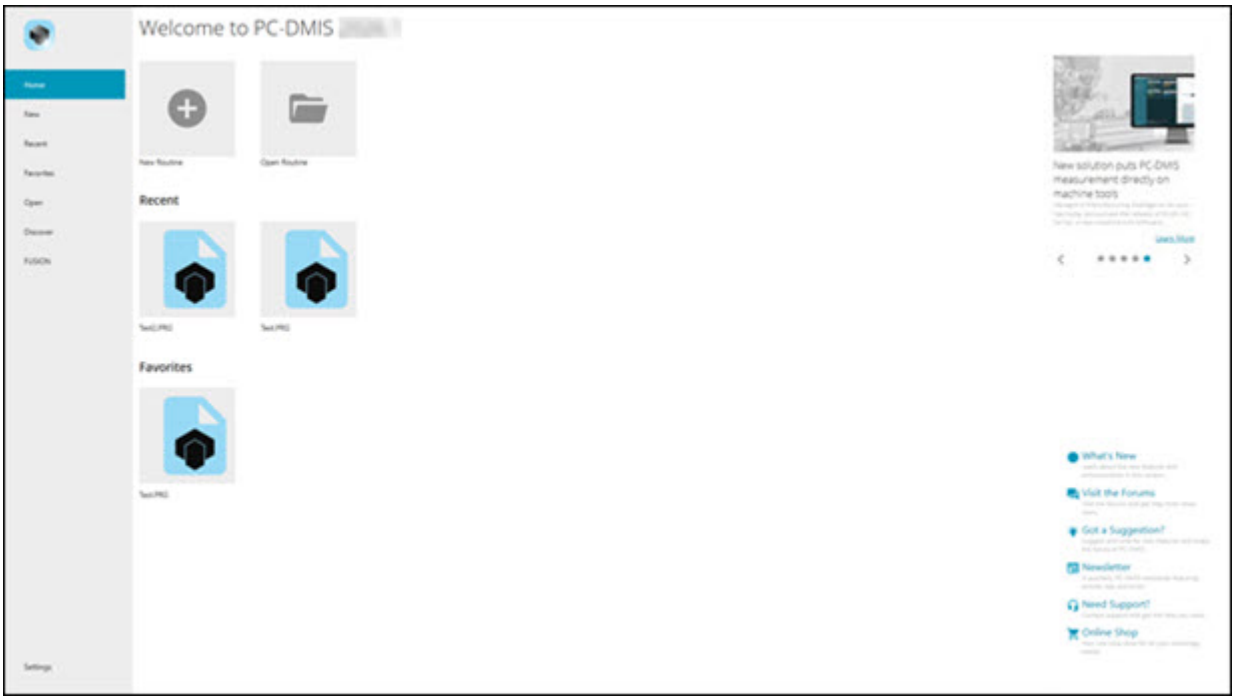
4. The Graphic Display window presents graphic representations of the part (the top view is shown in the illustration). The size of this area remains constant. However, you can divide the display area into as many as four different views of the part. For information, see "The Graphic Display Window".

5. Dialog boxes are the main communication channels between PC-DMIS and the user. Available features appear in the dialog box, and you can enter most input data through it. For information, see "Dialog Box Description".

6. The Edit window displays the measurement routine. This window allows you to access specific commands within the measurement routine and make changes to meet individual specifications. See the "Editing a Measurement Routine: Introduction" and "Using the Edit Window: Introduction" chapters for fully documented operations of the Edit window.

7. The Status bar displays information important to the current operation. This includes information such as the probe's current position, the probe calibration data, standard deviation of the last feature measured, the number of hits, and other information. It also contains a message area to display prompts and pertinent information about the operation in progress. For more information, see "The Status Bar".

The Home Page



The Home page is the first screen you see in PC-DMIS before you open a measurement routine. It's also the screen you see if you close all measurement routines but not the application. The Home page contains different views (such as **Home**, **Recent**, **Favorites**, **Open**, and **Discover**) that let you access measurement routines on your computer and perform other actions. The Home page also provides useful links to community sites, news, and product videos.

You can still use **File | Open** and **File | New** to open and create measurement routines.

By default, the Home page is visible when you start PC-DMIS and you don't load a measurement routine. If you don't see it, you may need to enable it. To do that, mark the **Show Home Page** check box in the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**). For information, see "Show Home Page" in the "Setting Your Preferences" chapter.

The Left Pane

From the Home page, the left pane lets you access these views:

- **Home** - This view provides tiles to create or open measurement routines. It also has its own **Recent** and **Favorites** embedded views. **Recent** here shows a row of tiles for the most recent measurement routines. **Favorites** here shows a row of tiles for the most recent items that you marked as favorites.
- **New** - This view shows templates from which you can create new measurement routines. When you first use PC-DMIS, this view contains these out-of-the-box templates:
  - **Blank Template** - This option creates a simple measurement routine, similar to the measurement routine creation from the **File | New** menu.
  - **Portable Execution Template** - This option creates a measurement routine intended for execution by portable devices.
  - **Q-DAS Export Template** - This option creates a measurement routine with pre-existing statistics and tracefield commands to export measurement info to the Q-DAS software.

  You can also modify any existing measurement routine and use it as a template. For more information, see the "Using Measurement Routine Templates" topic in the PC-DMIS Core documentation.
- **Recent** - This view shows tiles for all the recent measurement routines. It can show more than the single row of recent measurement routines available in the **Home** view.
- **Favorites** - This view shows tiles for all the items that you marked as favorites. It can show more than the single row of recent favorites available in the **Home** view.
- **Open** - This view shows tiles for any measurement routines or sub-folders within defined folders.

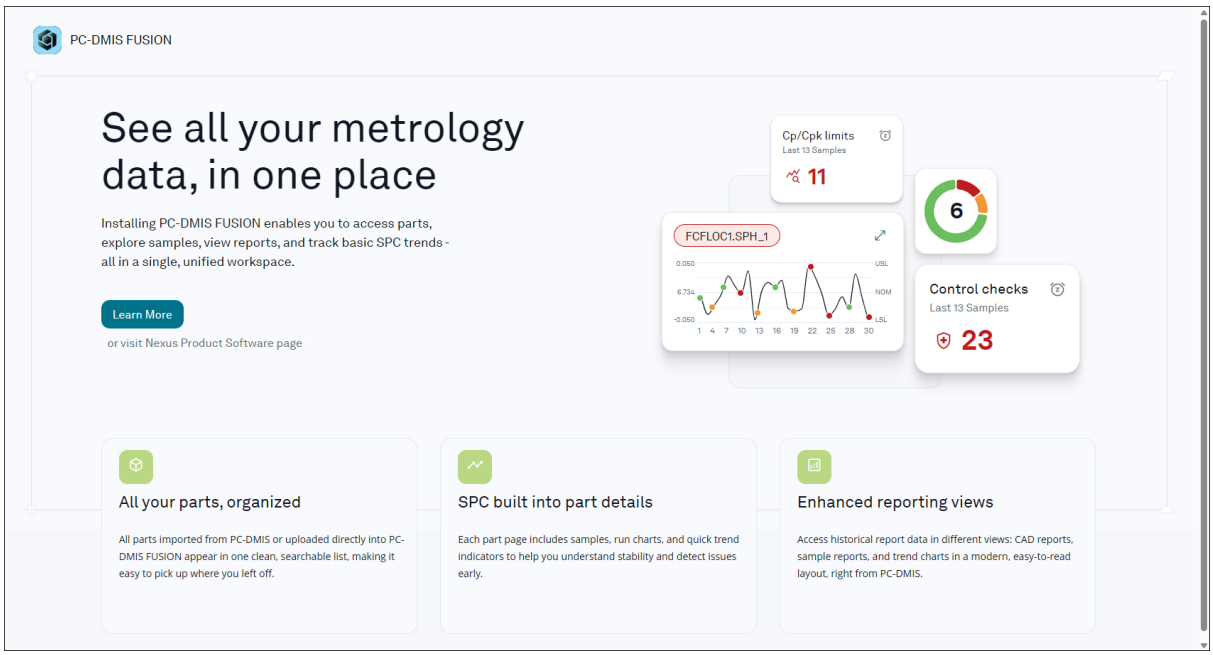
  By default, PC-DMIS shows the contents of this folder for the current version:

  C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2

You can define additional folders or hide folders from **Settings** if you run PC-DMIS as an administrator.

- **Discover** - With an Internet connection, this view shows these two tabs:
  - **Videos** - This tab provides a list of videos that showcase items, such as:
    - PC-DMIS product highlights
    - PC-DMIS how-to videos
    - Related products
  - **Example Routines** - This tab provides a list of example measurement routines with commands already in them to perform specific tasks or measurements. A descriptive picture on each tile shows the relative concept.
    - The measurement routines are grouped according to level. The level number represents the relative difficulty in the concepts involved.
    - Some items use automation scripting to accomplish their tasks. The descriptive pictures on the tiles use this small scroll icon in the bottom-right corner to indicate that scripting was used:

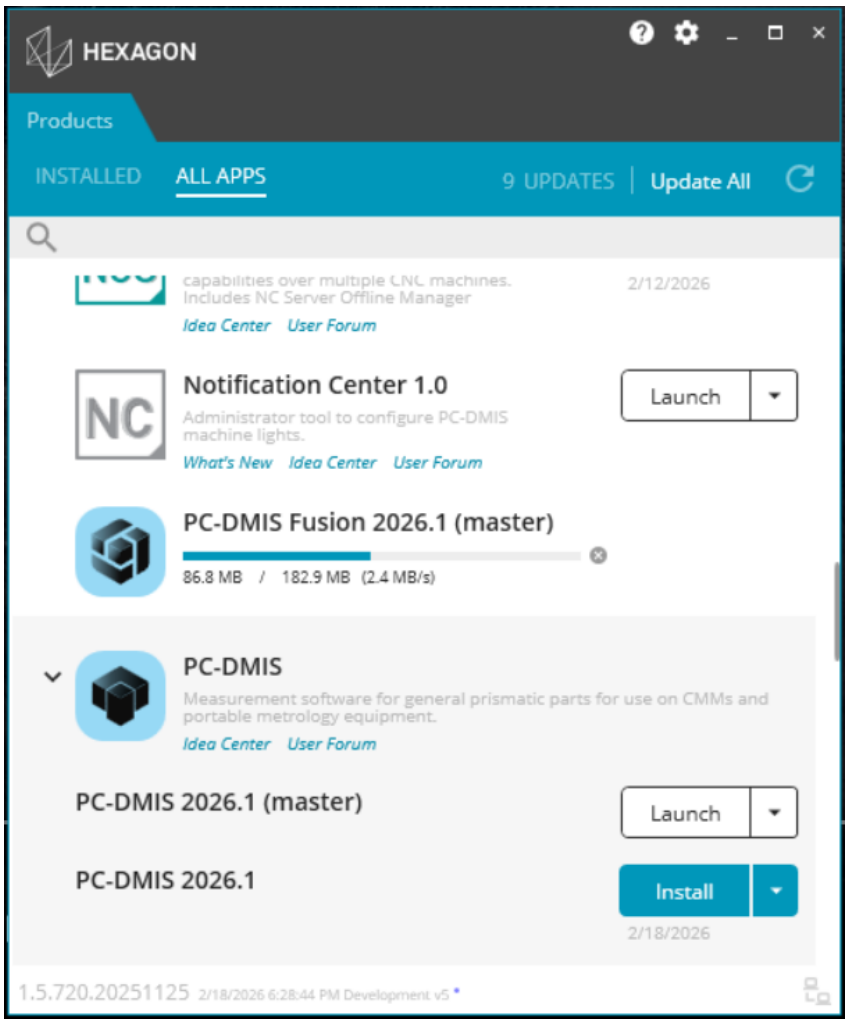
    - You can use the **...** menu on the tile to download measurement routines to the default measurement routine directory and add them to **Favorites**. You can also hover your pointer over measurement routines and use the download button ( ) to download them. Once downloaded, you can use the **Show in Folder Explorer** button ( ) to open the file in File Explorer.
- **FUSION** - This view initially shows a **Welcome** tab with information about the PC-DMIS FUSION application.



PC-DMIS FUSION brings all your metrology data together so you can:

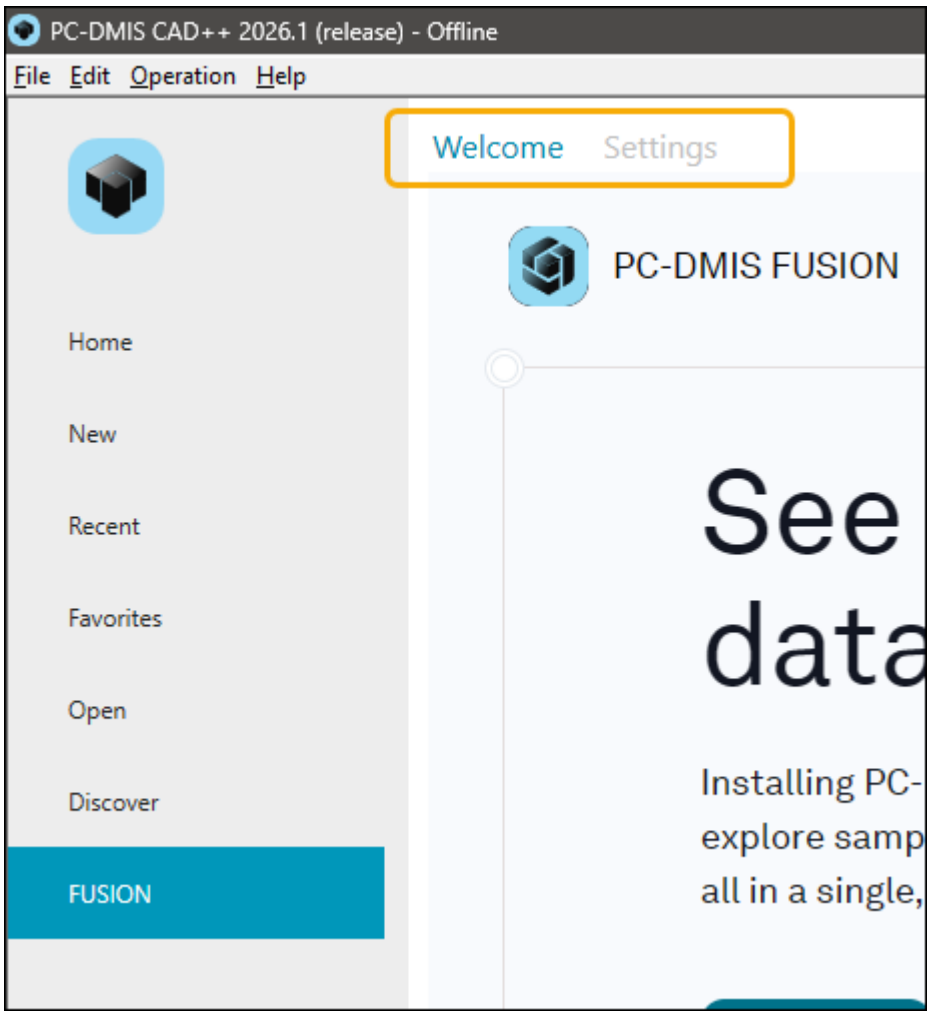
- Review PC-DMIS results and reports
- See sample-level details over time
- Explore SPC trends
- View part information in a central dashboard

To install PC-DMIS FUSION, from the **Welcome** tab, click **Learn More**. Once you have a license to use this application, you can download and install it from the Universal Updater application.



(For information on system requirements and installation instructions, see the PC-DMIS Fusion Release Notes.)

This installs the application on the same computer as PC-DMIS. Once you install PC-DMIS FUSION, on PC-DMIS, a **Settings** tab appears next to the **Welcome** tab within the **FUSION** view.



You can use the **Settings** tab to control how PC-DMIS sends your part data to the PC-DMIS FUSION application. PC-DMIS sends only what you choose to send, and all sent data remains on your computer. For more information, see "PC-DMIS FUSION Settings".

If PC-DMIS FUSION is not installed, only the **Welcome** tab is available.

Settings - This view shows the settings for the Home page.

The Right Pane

The right pane shows information and other options that change based on the current selected item.

Initial View

Initially, from the Home page's right pane, you can see links for these items. Most of these require an Internet connection to access:

- What's New

- Visit the Forums
- Got a Suggestion?
- Newsletter (available only if you have a current Software Maintenance Agreement)
- Need Support?
- Online Shop

Items for Measurement Routine

Once you select a measurement routine, the right pane changes to show a **Details** pane of the measurement routine's details:

At the top of the **Details** pane, you can see the routine's name and its thumbnail image. To the right of thumbnail image, the **Creation Date** shows the exact date and time that the measurement routine was created.

Change Thumbnail - This button appears over the measurement routine's thumbnail image. It allows you to change the thumbnail if you run PC-DMIS as an administrator. Note that the image is also linked with Inspect. This means if you change a thumbnail image in PC-DMIS, it also occurs in Inspect and vice versa.

Preview Pane - This area displays the CAD preview image if the measurement routine uses a CAD file. You can click **Set image as thumbnail** to create a thumbnail from the CAD preview image. You must run PC-DMIS as an administrator to change this thumbnail.

Favorite - In this area, you can click the star icon to add the measurement routine to favorites or to remove it from favorites.

Nickname - This box lets you type a nickname for the measurement routine. The actual file name in your folder structure is unaffected. The nickname only affects the name that you see in the **Favorite** view.

Open - This button opens the selected measurement routine.

Open Online - This option determines whether to open the measurement routine in Offline or Online mode. This option appears only if you have Online and Offline on your license.

Include CAD - This option determines whether you want to open the CAD file associated with the measurement routine.

Path - This text shows the folder path to the selected measurement routine. It is also a link that you can click on to open that routine's folder in the File Explorer and select the .prg file.

Open Folder Explorer - This button appears when you access the **Discover** tab from the left pane and then click on an example measurement routine. This button

opens File Explorer to the folder for the downloaded example measurement routine. The button becomes unavailable once you download the example measurement routine.

Items for Templates (New tab)

Once you select a template from the **New** tab, the right pane changes to shows details for the measurement routine you want to create. The **New Measurement Routine** dialog box from **File | New** contains this same information:

Nickname - This box defines the nickname for the measurement routine. By default, this matches the name of the template you selected. For more information, see "Nickname" above.

Part Name - This box defines the name of your part. The .prg file that PC-DMIS creates also uses this same name.

Location - This box defines the folder where you want to create the measurement routine.

Serial Number - This box defines the unique serial number for the part. This item is optional.

Revision Number - This box defines the revision number for the part or the measurement routine. This item is optional.

Units - This list defines whether the measurement routine uses millimeters or inches for the units of measure. The units you choose needs to match the units of the template file you selected.

Create - This button uses the above information and creates the measurement routine in the location folder.

Create Online - This option defines whether the measurement routine is an offline measurement routine (not connected to the machine) or an online measurement routine (connected to the machine).

- In an offline measurement routine, you can create the measurement routine and commands without a CMM, even if PC-DMIS is connected to a CMM. You define the features to measure with the CAD model.
- In an online measurement routine, you use the CMM's probe and jog box to teach PC-DMIS the features that you want to measure.

Items for Discover

Once you select a video from the **Videos** tab, the right pane changes to shows details for the selected video.

Launch - This button will launch the video in your default internet browser.

Settings

From the Home page, these are the available settings. To access the settings, you need to meet these conditions:

- Run PC-DMIS as an administrator
- PC-DMIS must not be in Operator Mode
- PC-DMIS must not be in Protected Mode

General Settings

Current Theme - This changes the look of the application between **Dark** and **Light**. You must re-start the application to see the new theme.

Internet Access - These two options let you customize Internet access for the Home page:

> **Allow Automatic Internet Access (News)** - This option enables the top-right news panel.
>
> **Allow Manual Internet Access (Links)** - This option enables all links to the Internet; these include the links on **Home** and the measurement routines and movies on **Discover**.

When disabled, the Home page has no access to items on the Internet.

Folder Explorer

PC-DMIS Routine Locations - In this section, you can click **Create New User Folder** to create new user-defined folder settings with pathways to folders with pre-defined or custom extensions. You can only make changes in this section if you run PC-DMIS as an administrator.

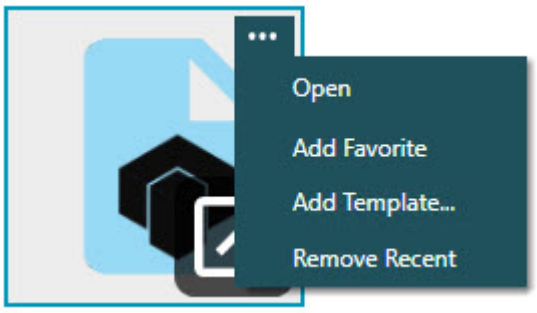
> **Label** - This option provides a simple label for the folder that appears under the folder in the other views. If you don't type anything here, the software uses the default file or folder name.
>
> **Path** - This option shows the pathway for the user-defined folder. You can define the pathway when you create it. If you want to change the path, you need to delete the specific user-defined folder setting and recreate it.
>
> **Extensions** - This box filters the files that PC-DMIS shows in the Home page. You can define this filter when you click **Create New User Folder**. You can add multiple filters as long as you separate them with commas. For example, you can type or select ***.prg** to show PC-DMIS measurement routines. You can type ***.*** to show all the files. If you want to change the extension, and you've already defined a user-defined folder setting, you need to delete the specific user-defined folder setting and recreate it.

- **Toggle Folder Visibility** - This option changes the visibility state of the user-defined folder. With this option, you can hide folders so they don't appear in **This PC.**

- **Delete This Setting** - This button appears next to user-defined folders. When you click it, PC-DMIS removes that user-defined folder from the **Settings** screen. The actual folder in Windows Explorer remains untouched.

Create New User Folder - This linked text adds a new user-defined folder setting that you can define.

The ... Menu



The **...** menu resides on top of measurement routine tiles on the Home page. To show the **...** menu, click on a measurement routine tile. This menu shows a list of common actions you can do with the measurement routine. Some of these options are identical to those contained on the **Details** pane. You may find these options useful if you don't want to use the **Details** pane to perform them:

- **Open** - This menu item opens the routine for edit or execution.
- **Add Favorite** - This menu item adds the routine to the **Favorite** view.
- **Add Template** - *This menu item only appears if you run PC-DMIS as an administrator.* It creates a template from the measurement routine. An **Add Template** dialog box opens so that you can type a name and description for the template. A template is helpful if you have commands you want to use over and over in new measurement routines. You can set up those commands in a measurement routine and create a template from it. For more information on this process, see "Using Measurement Routine Templates".
- **Remove Favorite** - This menu item removes the measurement routine from the **Favorite** view.
- **Remove Recent** - This menu item removes the measurement routine from the **Recent** view.
- **Edit Template** - *This menu item only appears if you run PC-DMIS as an administrator.* It shows an **Edit Template** dialog box so that you can modify a template description.

- **Remove Template** - This menu item removes the template from the **New** view.

Creating a Routine

1. From the Home page, click **Home**.
2. Click the **New Routine** tile to show the **New Measurement Routine** dialog box.
3. Fill out the dialog box and click **OK** to close the dialog box.

PC-DMIS also adds the routine to the **Recent** view.

Using Measurement Routine Templates

You can make a measurement routine, or part of a measurement routine, act as a measurement routine template. On the PC-DMIS Home page, you can then create other measurement routines from this measurement routine. This is useful when you frequently create the same commands over and over for new measurement routines.

Adding a Template from a Routine

1. Run PC-DMIS as an administrator.
2. From the Home page, from that measurement routine's tile, access the ... menu option.
3. Choose **Add Template** to show the **Add Template** dialog box.
4. In **Template Name**, give the template a descriptive name.
5. In **Description**, type a description of the template.
6. In **Units**, choose the measurement units of the template.
7. Click **OK** to create the template and add it into the **New** view on the left pane.

By default, the template includes all the commands from the first `LOADPROBE` and `TIP` commands to the end of the measurement routine.

PC-DMIS stores the template in a folder that matches your template name at this location:

C:\ProgramData\Hexagon\PC-DMIS\2026.2\Templates\

Controlling What Ends Up in a Template

If you want to include only parts of your measurement routine as template commands, you can control that with some special start and end Document comments.

1. Open a measurement routine that you want to use as your template.

2. Locate the commands that you want PC-DMIS to insert into new measurement routines.
3. Surround the commands you DO want to include with these Document comments:
    - #TEMPLATE_COMMAND_START_POINT#
    - #TEMPLATE COMMAND_END_POINT#

    > The commands between the template start and template end Document comments are what PC-DMIS adds into the new measurement routine when you create it from the template. PC-DMIS excludes commands that are *outside* of those comments.

4. Save and close the measurement routine.

Creating a Routine from a Template

1. From the Home page, from the left pane click **New** to show the available templates.
2. From the **New** view, choose the template from which you want to create a new measurement routine.
3. From the right pane, fill out the following:
    - In **Part Name**, type the name for the part. PC-DMIS uses this in the measurement routine's filename.
    - In **Location**, define where to store the measurement routine.
    - Fill out **Serial Number** and **Revision Number** as needed.
    - Select the **GD&T Standard** as needed.
    - In **Units**, select the same units as used by the template.
    - If PC-DMIS is in Online mode and you have the PC-DMIS functionality for offline measurement routines, decide if you want to connect the measurement routine to the machine or if it's an offline measurement routine. If you want it to connect to the machine, turn on the **Create Online** option. If you want an offline measurement routine, turn off the **Create Online** option.
4. Click **Create** to create the measurement routine.

The GD&T standard affects Size or Geometric Tolerance commands in your measurement routine. The standard you select does not affect legacy dimensions. You can change the selected GD&T standard at any time from the **Geometric Tolerances** tab of the **Setup Options** dialog box. However, because the setting applies to the entire measurement

routine, depending on the measurement routine's content at the time of change, it may behave differently.

For information on the **Geometric Tolerance** tab, see "Setup Options: Geometric Tolerances tab" in the "Setting Your Preferences" chapter.

Editing a Template's Description

1. Run PC-DMIS as an administrator.
2. From the Home page, from the left pane click **New** to show the available templates.
3. From the **New** view, choose the template whose description you want to modify.
4. From the **...** menu option, click **Edit Template**.
5. From the **Edit Template** dialog box, modify the **Description** and then click **OK**.

Editing a Template's Commands

1. In the File Explorer, navigate to this folder where PC-DMIS stores the templates:

   C:\ProgramData\Hexagon\PC-DMIS\2026.2\Templates
2. Locate the folder for your template. The folder name is the same as the template name.
3. Inside that folder, open either the **MM** or **Inch** folder.
4. Double-click the .prg to open it inside of PC-DMIS.
5. Modify and save that .prg.

Another way to modify the template is to modify the original measurement routine, and then re-create the template with the same name.

Removing a Template

From the **New** view, you can modify the template's description:

1. Run PC-DMIS as an administrator.
2. From the Home page, from the left pane click **New** to show the available templates.
3. From the **New** view, choose the template you want to delete.
4. From the **...** menu, click **Remove Template**.

Opening a Routine

1. From the Home page, click on one of these views: **Home**, **Recent**, **Favorites**, or **Open**.

2. Double-click on the tile that you want to open. You can also click once on a tile, and from the **...** menu, click **Open**.

PC-DMIS also adds the routine to the **Recent** view.

Removing a Routine from Recent

1. From the Home page, click the **Recent** view.
2. Select one or more routine tiles.
3. Click the **...** menu that appears in the upper right of the tile.
4. Click **Remove Recent** to remove the routine from the **Recent** view.

Adding Items to Favorites

1. From the Home page, select one or more measurement routines.
2. Click the **...** menu that appears in the upper right of one of the measurement routines.
3. Choose **Add Favorite**.
4. Click the **Favorites** view to see the newly-added item.

You can also use the **...** menu to add an item to **Favorites**. For information, see "The ... Menu".

Removing Items from Favorites

1. From the **Favorites** view on the Home page, select one or more routines.
2. Click the **...** menu that appears in the upper right of one of the routines.
3. Choose **Remove Favorite**.

You can also use the **...** menu to remove an item from **Favorites**. For information, see "The ... Menu".

Changing the Theme

1. From the left side bar on the Home page, click **Settings** to show the **Settings** screen.
2. From the **Settings** screen, change **Current Theme**, to either **Light** or **Dark**.
3. Restart PC-DMIS.

Adding Routine Locations

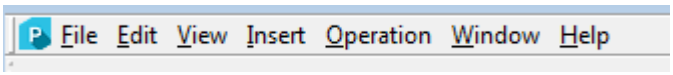
1. Run PC-DMIS as an administrator.
2. From the left side bar on the Home page, click **Settings** to show the **Settings** screen.
3. From the **Settings** screen, click **Create New User Folder** to define the new folder location.
4. For the new folder, define the **Label**, **Path**, and **Extension** values.
5. Click the **Confirm** button ( ) to accept your new location.
6. Click on the **Open** view to see the newly-added folder.

Disabling Home Page

1. Open or create a measurement routine.
2. Choose **Edit | Preferences | Setup** to show the **Setup Options** dialog box.
3. From the **Setup Options** dialog box, click the **General** tab.
4. From the list of check boxes on the **General** tab, clear **Show Home Page**.
5. Click **OK** to close the dialog box and accept your change.

The next time you close all your routines, you won't see the Home page.

The Menu Bar



The topics below give a brief description of each menu. For additional information, see the appropriate chapter as needed. PC-DMIS also provides a variety of shortcut menus. These menus are discussed in the "Using Shortcut Keys and Shortcut Menus: Introduction" chapter.

To get help on specific menu items, position your mouse pointer over the menu item, and press F1.

File

The **File** menu lets you open, save, and print files, access the Edit window, run measurement routines with the Execute Mode, and exit out of current measurement

routines. For more information, see the "Using Basic File Options: Introduction" and "Using Advanced File Options: Introduction" chapters.

You can control the number of files that PC-DMIS displays in the bottom half of the **File** menu. For details, see the "`MostRecentFileList`" topic in the PC-DMIS Settings Editor documentation.

Edit

The Edit menu allows you to edit your measurement routine preferences, your measurement routine, or the CAD display.

- To edit your preferences, see the "Setting Your Preferences: Introduction" chapter for complete information.
- To edit your measurement routine via the Edit window, see the "Editing a Measurement Routine: Introduction" and "Using the Edit Window: Introduction" chapters for complete information.
- To edit the display of the CAD drawing, see the "Editing the CAD Display: Introduction" chapter for complete information.

View

The view menu allows you to access all the windows, editors, and toolbars available to PC-DMIS. See the "Using Other Windows, Editors, and Tools: Overview" chapter for complete information.

Insert

You can access the bulk of a measurement routine's commands from the **Insert** menu.

Most of the chapters that comprise this documentation come from the options and submenus displayed on this menu.

- You can find the information for the options on the **Hardware Definition** submenu in the "Defining Hardware" chapter.
- You can find the information for the options on the **Feature** submenu in these chapters: "Creating Auto Features", "Creating Measured Features", "Constructing New Features from Existing Features", "Creating Generic Features".
- You can find the information for the options on the **Alignment** submenu in the "Creating and Using Alignments" chapter.
- You can find the information for the options on the **Dimension** submenu in the "Using Legacy Dimensions" chapter.

- You can find the information for the options on the **Scan** submenu in the "Scanning your Part" chapter.
- You can find the information for the options on the **Move** submenu in the "Inserting Move Commands" chapter.
- You can find the information for the options on the **Flow Control Command** submenu in the "Branching by Using Flow Control" chapter.
- You can find the information for the options on the **Statistics Command** submenu in the "Tracking Statistical Data" chapter.
- You can find the information for the options on the **Report Command** submenu, including the ability to add Diminfo and Pointinfo boxes onto your CAD display and into your measurement routine, in the "Inserting Report Commands" chapter.
- You can find the information for the options on the **File I/O Command** submenu in the "Using File Input / Output" chapter.
- The options on the **Modules** submenu represent the toolkit modules installed with PC-DMIS.
- You can find the information for the remaining options on the **Insert** menu in the "Using Expressions and Variables " chapter and the "Adding External Elements" chapter.

Operation

The items on the **Operation** menu are discussed in various topics of the documentation.

Window

The **Window** menu allows you to manage the display of all open measurement routines. See the "Navigating and Displaying Multiple Windows" chapter.

Help

The **Help** menu provides these menu items:

Search - This item opens the introduction page for the PC-DMIS Getting Started help. From there, you can search across the entire PC-DMIS Help content. To access the PC-DMIS Help, select this item from the **Help** menu, or press F1 at any time. The first time you run the Help after installation, it may take several seconds. Thereafter it opens more quickly.

Using Help - This item opens the PC-DMIS help file to the "Using the Help: Introduction" chapter. The topics there provide hints and tips about searching for and locating information in the help.

Offline Help - This item determines whether PC-DMIS shows the offline help whenever you access the Help content. A check mark next to this menu item means PC-DMIS uses the offline help even if you have Internet access. No check mark means PC-DMIS uses the online help. If you don't have Internet access and you don't have the offline help installed, PC-DMIS shows a message to inform you that you need to either connect to the Internet or install the offline help.

Release Notes - This item shows the readme.pdf that ships with this version of the software. The readme.pdf contains information about this version of PC-DMIS.

About - This item opens the **About** dialog box. This dialog box gives you specific version and license information. It also provides a unique **Machine ID** for your computer, which may be used in some status and notification messages.

Selecting Options from the Menu Bar with the Keyboard

To open a menu and browse through the options using the keyboard, press the Alt key followed by the underlined letter in the name of the menu to be opened.

For example, you can see that the "V" in the V**iew** menu option is underlined. This means you can press the Alt + V keys to open the **View** menu. You can use a similar approach to select options within the **View** menu. Simply press the underlined letter that corresponds with the menu option. (i.e., Cut: press the "T" key).

Menus also can be opened using the arrow keys.

To do this:

1. Press the Alt key to switch to the menu bar.
2. Use the Left or Right Arrow keys to move to the desired menu.
3. Use the Up or Down Arrow keys to select an option.
4. When the desired option is highlighted, press the Enter key.

Shortcut keys also have been assigned to many of the options. The shortcut key is shown to the right of the command or option.

See the "Using Shortcut Keys and Menus" chapter for more information.

The Toolbar Area

PC-DMIS provides you with a variety of toolbars composed of frequently used commands. These toolbars can be accessed in one of these two ways.

1. Select the **View | Toolbars** submenu and select a toolbar from the menu provided.
2. Right-click on PC-DMIS's **Toolbar** area and select a toolbar from the shortcut menu provided.

Once you have a toolbar in the toolbar area, you can move the toolbar by first clicking on the space between the left or right edge and a button and then dragging the toolbar to a different location.

For in-depth information on all of PC-DMIS's toolbars, see the "Using Toolbars" chapter.

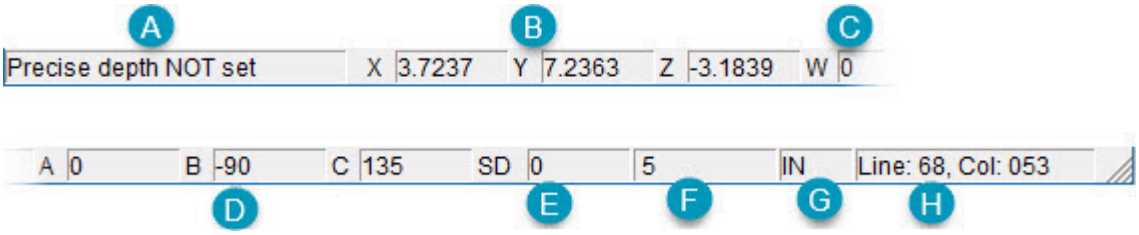
The Graphic Display Window

The Graphic Display window, quite simply, displays graphics. This window is opened by default when you create or access a measurement routine. To hide or view this window, select the **View | Graphic Display Window** menu option. This window displays the following:

- A multi-paned graphical representation of an imported CAD part, the probe, and any simulated machines or fixtures.
- The measured, auto, and constructed features and their ID labels.
- The scans, dimensions, GD&T callouts, and text box displays.

The Graphic Display window is an integral part of PC-DMIS. For information on editing the Graphic Display window and altering the screen display, see "Editing the CAD Display" chapter. For information on how to set up the Graphic Display window, see "Setting Up CAD and Graphics". For information on how to use the Graphic Display window, see the topics below.

The Status Bar



The Status bar displays the following:

A. **Message Box:** The left-most box displays instructions and other information regarding the current procedure or command.

B. **X, Y, and Z boxes:** These boxes show the X, Y, and Z position of the probe. Clicking on any of the three coordinate displays within the Status bar will cause the **Auto Move Point** dialog box to appear. (See "Inserting a Move Point Command" in the "Inserting Move Commands" chapter for additional information regarding DCC Move Point.)

C. **W:** Angle of Rotary Table.

D. **A, B, and C boxes**: These boxes show the A, B, and C wrist angles for a three-axis wrist probe such as a CMS probe. Your wrist probe may only support two axes. In that case, this portion of the Status bar only shows A and B.

E. **SD:** This shows the standard deviation of the last feature measured.

F. **# of Hits:** This shows the number of hits currently taken.

G. **IN/MM:** This shows the current unit of measurement used for the measurement routine.

H. **Line:** This box displays a number that indicates the current line where the cursor is located in the Edit report. **Col:** This text displays a number that indicates the current column where the cursor is located in the Edit report.

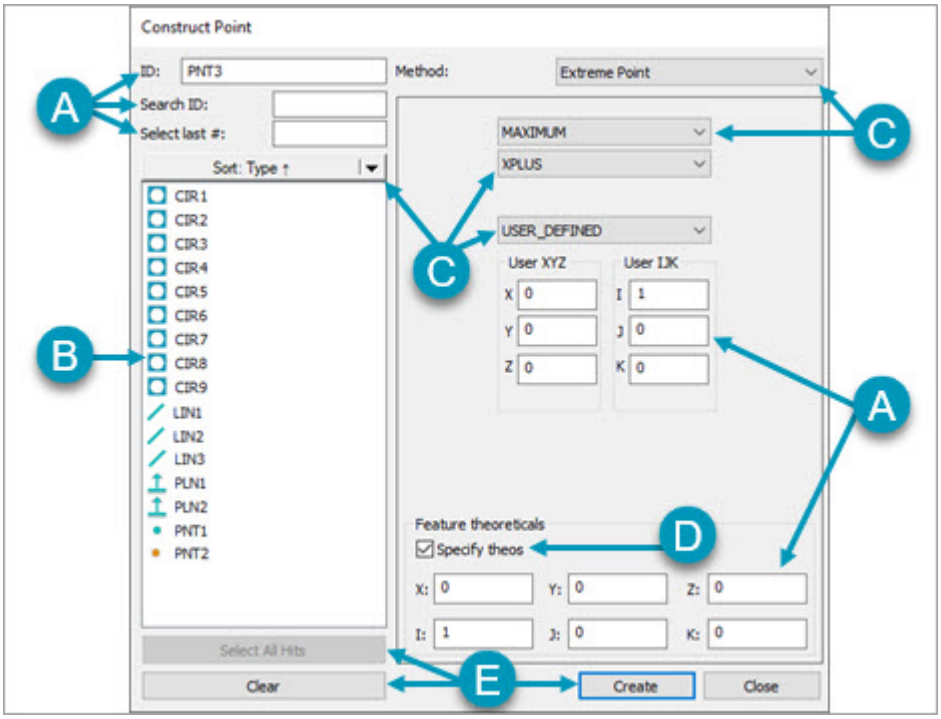
Status Bar Manipulations

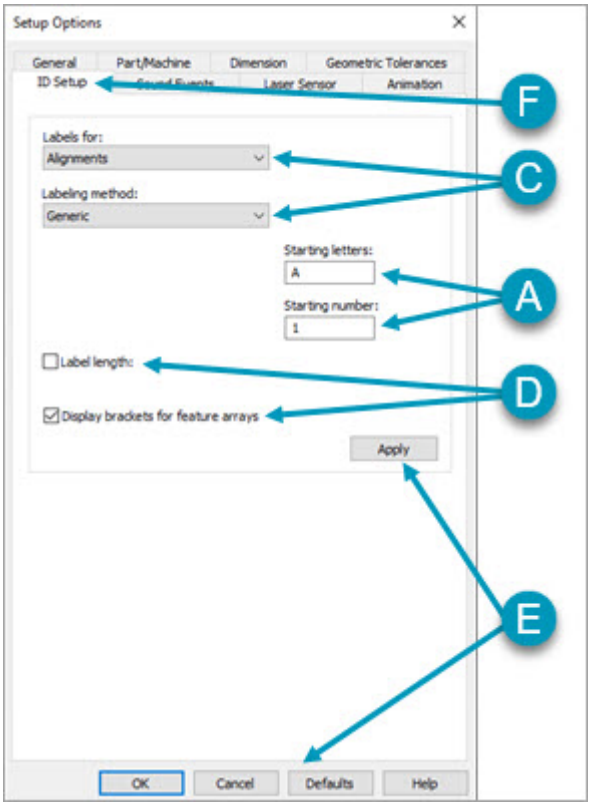
- To increase the size of the Status bar, select **View | Status Bar | Large**. Due to its increased size, this large Status bar only displays the **Message**, the **XYZ**, the **SD**, the **# of Hits:**, the **Line:**, and the **Col:** boxes.
- To put the Status bar back to its normal size, select **View | Status Bar | Normal**.
- To hide or show the Status bar, select **View | Status Bar | Hide Status Bar(s)**.

Dialog Box Description

A dialog box provides various boxes and buttons (called controls) that you can select to tell PC-DMIS how to carry out a command. Sometimes a dialog box shows the default settings already selected, which if satisfactory, can be left as they are. After you modify a dialog box, click **OK** to carry out the command. To cancel the command and close the dialog box, click the **Cancel** button.

The graphics illustrate some features common to most dialog boxes in PC-DMIS.





A - Edit Boxes or Fields

B - List Box

C - Drop Down List Box

D - Check Boxes

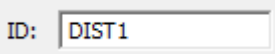
E - Command Buttons

F - Tabs

Edit Box

An edit box allows you to type in a desired value or name. In this documentation, edit boxes are simply referred to as "boxes". Edit boxes common to many of PC-DMIS's dialog boxes are listed here:

ID= [Feature]



This box shows the Feature or Dimension identification or name for the features or dimensions to be created.

To change the ID:

1. Select the previous ID.
2. Type a new ID.
3. Press the Tab key.

For the rules that apply when you type an ID, see "Setup Options: ID Setup tab" in the "Setting Your Preferences" chapter.

For a Dimension ID:

- You can change only the number portion of the ID and reset the dimension counter to a desired count.
- You can change the text portion of the ID and set it as the default text for subsequent dimensions. (See the **ID Setup** tab of the **Setup Options** dialog box in the "Setting Your Preferences" chapter.)

Search ID

In the **Search ID** box, you can type the feature (or dimension) IDs to use in the current operation. You can use this box to search using meta-character search options. The feature or dimension IDs will be highlighted in the appropriate **Feature List** box (or **Dimension List** box).

If you type a feature's ID in the **Search ID** box and press the Enter key, PC-DMIS selects that feature from the **Feature List** or **Dimension List** box.

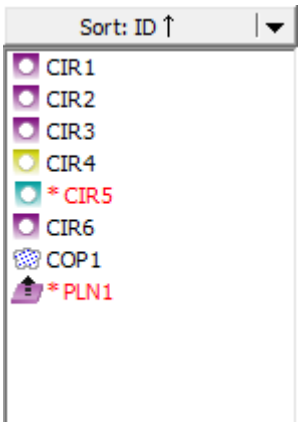
Select Last

In the **Select Last #** box, you can select the last of a certain number of features (or dimensions). For example, if you wanted to select the last four features that were created, simply type **4** and press the TAB key. PC-DMIS will highlight the features selected in the **Feature List** box.

List Box

A list box is similar to an edit box; however, changes cannot be made to the listed values. This documentation refers to either "list" or "box" for the list box. Lists common to many of PC-DMIS's dialog boxes are listed here:

Feature List box



Feature List Box

A feature list box contains a list of the available features within a measurement routine. PC-DMIS uses the selected features to define alignments, construct new features, create dimensions, and perform other tasks. You can select features by following the instructions in the "Selecting Features Using the Graphic Display Window" topic in the "Editing the CAD display" chapter.

You can also use the Graphical Hit Selection method to select inputs to your alignment, construction, or dimension. For details, see "Graphical Hit Selection Method".

You can add new features while the dialog box is open if you use the QuickFeature functionality. PC-DMIS selects the features in the list as you create them. For more information on creating QuickFeatures, see "Creating QuickFeatures" in the "Creating Auto Features" chapter.

A feature displayed in red text with an asterisk (*) to the left of it represents a feature that has been skipped during measurement routine execution.

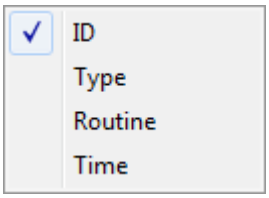
For details on how PC-DMIS reports dimensions that use skipped features during execution, see the "Reporting a Dimension Using a Feature Skipped During Execution" topic in this documentation.

You can deselect selected (highlighted) features with the **Clear** or **Clear List** buttons.

If you are attaching an external measurement routine (see "Attaching an External Measurement Routine" in the "Adding External Elements" chapter), PC-DMIS also lists the variable (or pointer) used to reference the attached measurement routine. A plus sign (+) appears to the left of the pointer. Clicking this plus sign expands or collapses a view of all the features in the attached measurement routine.

Sorting the Features

Clicking the drop-down arrow button located at the top of the **Feature List** box displays the various sorting methods available as shown below.



Feature List Sort Menu

Once a selection is made, the list is automatically sorted based on the selected option. As new items are added to the list, clicking on the **Sort** button will re-sort the list based on the currently sorted options shown on the button. Re-clicking the button will sort the list in reverse as shown be the up or down arrow to the right of the displayed sorting method.

Sorting Options Defined

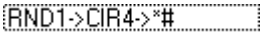
- **ID** - Sorts the list alphabetically based on feature ID.
- **Type** - Sorts by feature type. In this case, the feature's ID is used as a secondary sort criteria.
- **Routine** - Sorts the list based on the order the feature appears in the measurement routine.
- **Time** - Sorts the list based on the times the features were last measured.

Dimension List box

The **Dimension List** box contains a list of all the dimensions that were created within a measurement routine. You can edit or analyze a dimension by selecting the desired dimension in the list box and selecting the desired changes or options in the associated dialog box. You select multiple dimensions at a time by holding down Ctrl or Shift and clicking on another dimension.

Inside the **Edit Dimension Info** dialog box, if the dimension already has a Dimension Info box associated with it, PC-DMIS shows an asterisk (*) symbol next to the dimension ID in the list. If the dimension info is hidden in all current views, it adds a number sign (#) symbol next to the ID.

For example, suppose you have a roundness dimension named RND1 for a circle named CIR4. If the roundness dimension already had a DIMINFO box associated with it, yet that DIMINFO box was hidden from all views in the Graphic Display window, the ID in the list would look like this:



Example showing asterisk and number sign symbols.

Check Boxes

Click on check boxes to turn options on or off. The option is considered "on" when a check mark appears inside the box. You can select multiple check boxes.

Command Buttons

A command button carries out an action. The command buttons that are common to most dialog boxes in PC-DMIS are listed here:

Apply - This button applies your selections to the dialog box but keeps the dialog box open for further changes.

OK - This button saves, applies the changes, and closes the dialog box. On some dialog boxes, this button inserts commands into the Edit window.

Cancel - This button ignores any changes and closes the dialog box.

Delete - This button removes items (such as created features, dimensions, probe files, alignment information, and so on) from certain list boxes. In the **Delete CAD** dialog box, the **Delete** command button deletes selected nominal features. For more information, see "Deleting CAD".

Last - This button selects the last item in the associated list box.

Last Two - This button selects the last two items in the associated list box.

Select All - This button selects all the items in the associated list box.

Defaults - This button updates the default settings of several parameters. When you create a new measurement routine, it reflects any changes that are stored as the default values.

The default values are stored in the JSON file. You can update these parameters through the appropriate dialog box or use the *PC-DMIS Settings Editor* to edit the setting entry itself. This application is included with your PC-DMIS install set. For more information, see "Modifying Setting Entries".

- If you click the **OK** button without clicking the **Defaults** button, the defined parameters apply only to the active measurement routine and will not affect the entries.
- If you change a parameter and then click the **Defaults** button, PC-DMIS updates the entries to redefine the default to the current entry.

The **Defaults** button allows you to store and apply new default values to new measurement routines created later on.

Recall - This button calls back the original factory settings as long as the original values haven't changed through clicking the **Defaults** button. In that case, PC-DMIS returns the stored values in the JSON file.

For information on how to use the **Machine Options** dialog box to restore original factory settings by reading them from the controller, see "Setting Up the Machine Interface" in the "Setting Your Preferences" chapter.

Create - This button applies changes made in certain dialog boxes and creates features, dimensions, scans, and so on. It then places the created item into the Edit window and the Graphic Display window.

The **Create** button won't close the dialog box. To close the dialog box, click the **Close** button.

Close - This button closes the dialog box. The **OK, Apply**, or the **Create** button doesn't automatically close the dialog box with some dialog boxes (where you may want to select further options).

Clear - This command button clears the highlight of any selected item from the list boxes. In some cases, this button completely removes items from certain list boxes (such as the AB positions from the **New Angles List** box).

In some dialog boxes and tabs, the **Clear** button clears any values that were entered or changed before either the **Apply** or the **OK** button was clicked and restores the "built in" values in PC-DMIS. If you have used the **Default** button to store default values, then PC-DMIS restores those values.

Undo - This button reverses changes that you made using the **Apply** button (or the **Generate** button in DCC scans). Once you click the **OK** button, you cannot reverse the changes.

Help - This button opens the online Help topic associated with the current dialog box.

Dialog Box Tabs

Some dialog boxes contain tabs. These tabs act as identifying markers much like a folder's name in a filing cabinet would. Click on a tab to bring a desired window or options in front of the other tabs. Any options or commands associated with that tab are then displayed.

Drop-Down List Box

A drop-down list box works like menus. Click on the drop-down arrow icon to the right of the current choice, or press the Alt key and then the Up or Down arrow key on the keyboard consecutively. Press F4 to display the available options. Press the key(s) again to close the list. The software shows the current choice in the highlighted box.

Accessing Dialog Box Options

You can use the mouse or keyboard to access the options within a dialog box. To use the mouse to access an option, simply point to the desired option and click.

The keyboard offers a variety of ways to move between available options:

- Press the **Tab** key to move to the next available option.
- Press **Shift + Tab** to move to the previous available option.
- Press the **Arrow** key to display a current drop-down list.
- Press the **Enter** key to select the current command.
- Press the **Space Bar** to select the current check box or button.

Scrolling the Edit Window

The Edit window can be moved in the same manner as a dialog box. See "Moving Dialog Boxes".

The Edit window has scroll bars for viewing additional data in the window.

To scroll:

1. Place the mouse pointer on one of the scroll arrows.
2. Click or hold the left mouse button.

PC-DMIS scrolls the text in the scroll arrow's pointed direction. For example, to scroll down, select the down scroll arrow with the pointer and click the left mouse button.

See the chapters "Editing a Measurement Routine" and "Using the Edit Window" for complete information regarding the Edit window.

Moving Dialog Boxes

You can use the mouse to move a dialog box or window.

To move a dialog box or window:

1. Place the mouse pointer over the title bar.
2. Hold down the left mouse button.
3. Drag the dialog box or window to the desired position in the Graphic Display window.
4. Release the mouse button.

PC-DMIS moves the dialog box or window to its new location.

PC-DMIS stores the last position of the various PC-DMIS dialog boxes. When you re-open the dialog box, it opens the dialog box at the last used location, even across multiple monitors.

If you move PC-DMIS to another monitor, PC-DMIS resets the dialog box positions. This means it opens the dialog boxes in their default position on that monitor.

You can dock some dialog boxes and other interface elements inside the rest of the user interface. You may need to undock these items before you move them. For details on how to dock and undock UI elements, see "Docking and Undocking User Interface Elements".

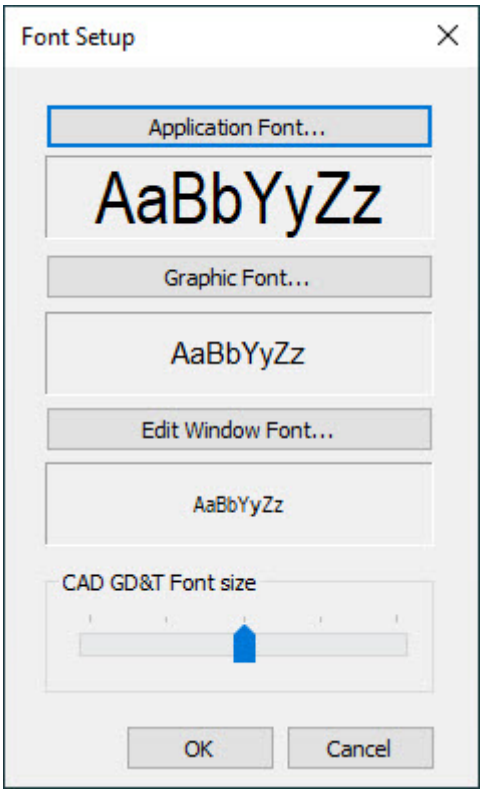
Customizing the User Interface

You can reorganize menus, add your own programs to menus, or create new menus and options altogether. PC-DMIS also allows you to link commands native to PC-DMIS and customized commands to toolbars. Instructions on how to modify the user interface are in the following topics:

- Restoring the Default User Interface
- Customizing User Interface Fonts
- Customizing Menus
- Customizing Toolbars
- Customizing Shortcut Keys

The organization of this documentation is based on the default user interface shipped with this version PC-DMIS. Be aware that customizing the user interface may make existing documentation difficult to follow.

Customizing User Interface Fonts



Font Setup Dialog Box

To modify font attributes, select the **Edit | Preferences | Fonts** menu option to open the **Font Setup** dialog box. Use this dialog box to change the font and its attributes (such as the font size and style) for these areas of the main PC-DMIS user interface:

- The Status Bar
- The Graphic Display window
- The Edit window

To change the font:

1. Click one of these buttons to open a standard **Font** dialog box:
    - **Application Font** - This changes the font of the Status Bar in the main application.
    - **Graphic Font** - This changes the font for text in the Graphic Display window. This affects feature labels, Dimension Info and Point Info text boxes, and so on. The **Color** list on the **Font** dialog box that appears sets the background color for CAD GD&T characteristic IDs. For information on these characteristic IDs, see "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.
    - **Edit Window Font** - This changes the font for the Edit window. For a recommended font for the Edit window, see the note below.
2. From the **Font** dialog box, use the lists and select your font changes.
3. Click **OK** to close the **Font** dialog box.
4. From the **Font Setup** dialog box, click **OK** to accept your change.

You can use any font available on your computer system; however, some fonts don't have a uniform spacing. This can make some components more difficult to read. For the Edit window text, we recommend that you use a monospace, fixed-width font, such as the *Courier New* font. Otherwise, characters, lines, and headings may not line up as expected.

CAD GD&T Font size

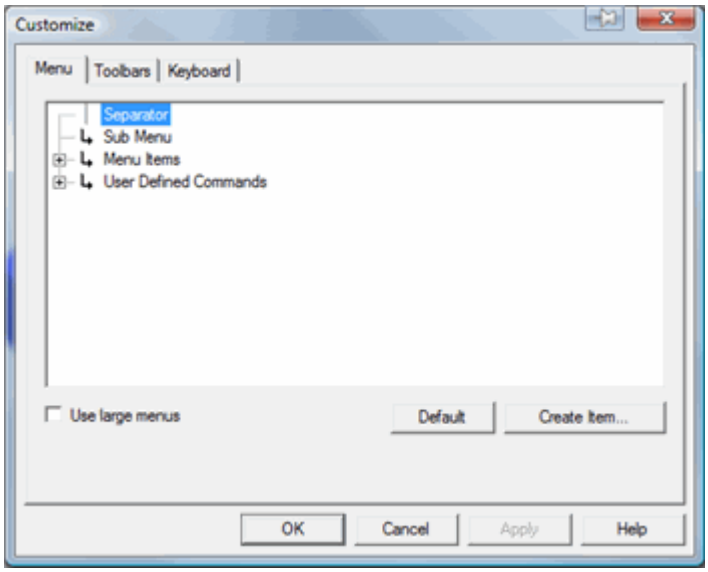
This slider scales the size of any GD&T CAD elements that appear in the Graphic Display window. Each increment above or below the midpoint increases or decreases the font by 10%. The font size changes dynamically as you adjust the slider. Click **OK** to keep the changed font size.

Customizing Menus

To customize the menus on the menu bar:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option.

2. Select the **Menu** tab.



Customize Dialog Box - Menu Tab

You can use this tab to add new menu items, move existing menu items, or remove menu items from the menu bar altogether.

You can use the **Use large menus** check box to show all menu items in a larger font. PC-DMIS will need to restart to apply the large menu sizes.

To Move a Menu Item

To move a menu item to a new location on the menu bar:

1. Select the **View | Toolbars | Customize** menu option to open the **Customize** dialog box, and then click the **Menu** tab.

2. Navigate to the menu item you want to move on PC-DMIS's actual menu bar (not the tree view menu list in the **Menu** tab).

3. Select the desired menu item.

4. Drag it to the new position. As you drag the item through the menu structure, PC-DMIS displays a small red arrow to indicate where you want to add the item.

5. Release the mouse button. The menu item is moved from its original location to the new location.

6. Click **OK** to accept your changes and close the **Customize** dialog box.

To Remove/Restore a Menu Item

To Remove a Menu Item from the Menu Bar:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and click the **Menu** tab.
2. Navigate to the menu item you want to remove on PC-DMIS's actual menu bar (not the tree view menu list in the **Menu** tab).
3. Select the desired menu item.
4. Drag it off of the menu bar area.
5. Release the mouse button, and the menu item is removed from the menu.
6. Click **OK** to accept your changes and close the **Customize** dialog box.

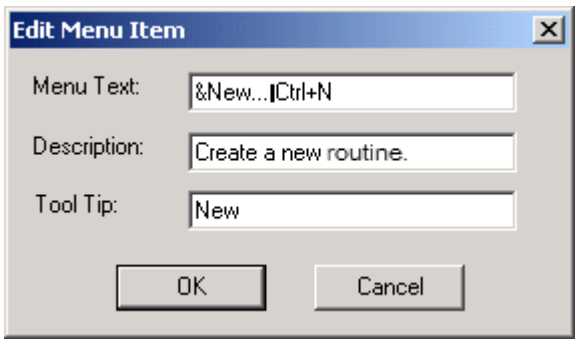
To Restore a Removed Menu Item:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and click the **Menu** tab.
2. Navigate in the **Menu** tree view list to the menu item you want to restore.
3. Select the menu item and drag it up onto PC-DMIS's actual menu bar. As you drag the item through the menu structure, a small red arrow will appear indicating where the item will be added when you release the mouse button.
4. Release the mouse button to insert the menu item.

To Edit an Existing Menu Item

You can edit existing menu items, including menu item names, descriptions, and tooltips. To do this:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option.
2. With the dialog box open, click on PC-DMIS's menu bar.
3. Select a menu and then right-click on the menu item you want to change on PC-DMIS's actual menu bar (not the tree view menu list in the **Menu** tab). The **Edit Menu Item** dialog box appears.



Edit Menu Item Dialog Box

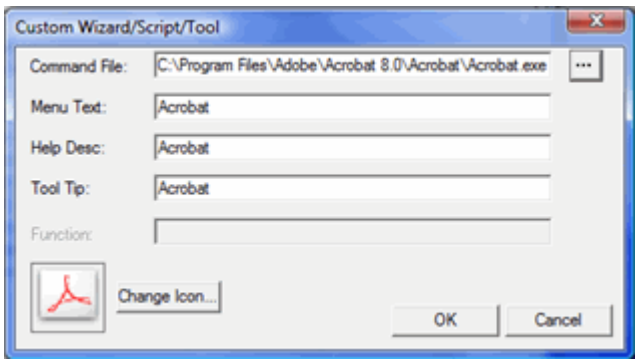
4. Modify the menu item using this dialog box and then click **OK**.
5. Click **OK** on the **Customize** dialog box.
6. Verify your change.

In the **Menu Text** box, the letter that follows the ampersand sign (&) indicates that the letter is used with the Alt key to quickly select that menu item. For example, when you press the key combination Alt + F + N, the **New Measurement Routine** dialog box opens from the **File | New** menu option. Note that you can also access this dialog box with the Ctrl + N key combination.

To Add a New Menu Item

You can create your own custom menu item and tie an external program or batch file to it. To do this:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and select the **Menu** tab.
2. Click the **Create Item** button. The **Custom Wizard / Script / Tool** dialog box appears.
3. Click the **'…'** button. An **Open** dialog box appears.
4. From the **Files of Type** list, select the type of file you want to insert.
5. Navigate to the directory containing the custom application, batch file, BASIC script, and so forth.
6. Select the file to insert and then click **Open**. The **Open** dialog box closes, and the **Custom Wizard / Script / Tool** dialog box displays the default information associated with the custom program you selected.



Example of a sample application in the Custom Wizard/Script/Tool dialog box.

Command File:
This box shows the pathway to the command.

Menu Text:
This box allows you to specify the name used on the menu for this item.

Help Desc:
This box allows you to define information that PC-DMIS displays in the Status bar when you move your mouse over the item.

Tool Tip:
This box allows you to specify the tooltip text displayed for this item when you hover your mouse over it. This only works if the custom item is added to a toolbar, not a menu.

Function:
This box remains unavailable for selection unless you are using a custom BASIC script. Once the box becomes enabled, you can use it to define the function or subroutine that runs from the loaded .bas file. Type the routine name without parentheses. For example, if you type **TestFunction**, PC-DMIS launches the TestFunction routine when the BASIC script is executed.

Change Icon:
This button allows you to change the icons displayed for this menu item.

7. Make any additional changes to this dialog box.
8. Click **OK** to accept your changes. PC-DMIS inserts the newly created command into the **User Defined Commands** tree view list in the **Menu** tab of the **Customize** dialog box.
9. Expand **User Defined Commands** tree view list.
10. Select the newly created command and drag it up onto PC-DMIS's actual menu bar. As you drag the item through the menu structure, a small red arrow will appear indicating where the item will be added when you release the mouse button.
11. Release the mouse button to insert the command.

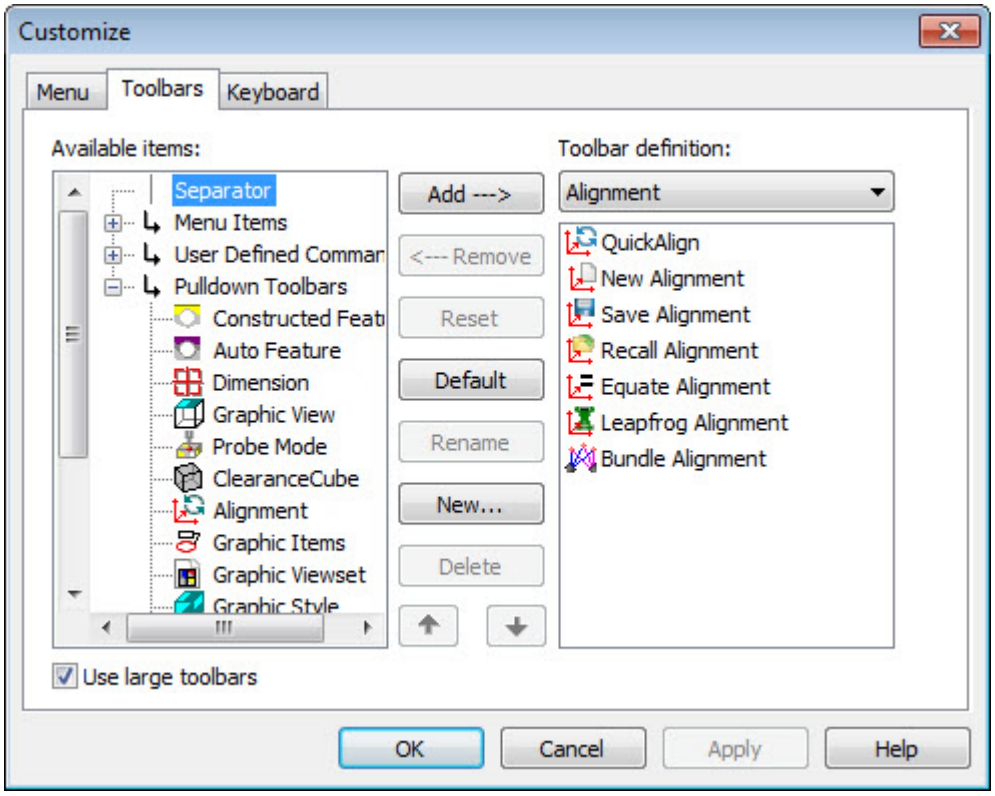
12. Close the **Customize** dialog box. The newly inserted menu item now resides in your menu bar.

Customizing Toolbars

To customize the toolbars that appear on the toolbar menu:

1. Select the **View | Toolbars | Customize** menu option to access the **Customize** dialog box.

2. Select the **Toolbars** tab.



Customize Dialog Box - Toolbar Tab

You can use this tab to add new toolbars and new toolbar icons, add new dropdown toolbars, remove custom toolbars, remove toolbar icons, and reorder toolbar icons on existing toolbars.

You can use the **Use large toolbars** check box to show all toolbar icons in a larger size. This change happens automatically when you click **OK** or **Apply**. PC-DMIS does not need to restart.

To Modify an Existing Toolbar

You can modify existing toolbars as described below. First, access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and click the **Toolbars** tab.

To add a menu item to a toolbar:

1. Select the toolbar from the **Toolbar definition list**.
2. Find the item you want to add in the **Available items** list.
3. Select it, and click the **Add** button. The item appears on the desired toolbar.
4. Click **Apply** and then **OK** to accept your changes.

To remove a toolbar icon:

1. Select the toolbar from the **Toolbar definition** list.
2. Find the item in the list to remove (right side of dialog).
3. Select it and click the **Remove** button.
4. Click **Apply** and then **OK** to accept your changes.

> Existing toolbars must have at least one icon on them. If you remove the last icon and click **Apply** or **OK**, the icon still remains on the toolbar.

To erase any changes:

Click the **Reset** button. As long as the **OK** or **Apply** buttons haven't been clicked, the toolbar reverts to its previous state.

To remove all customized toolbars and restore the layout shipped with this version:

Click the **Default** button.

To create a new toolbar item:

Click **New**. The **Custom Wizard / Script / Tool** dialog box opens. Select an item to add.

This documentation covers the process of adding custom items in the "To Add a New Menu Item" topic above.

To rearrange the item order on a toolbar:

1. Select the toolbar from the **Toolbar definition** list.
2. Select the item that you wish to move.
3. Click the up arrow or down arrow button to move the item up or down respectively among the other toolbar icons in the list.

To rename a custom toolbar:

1. Select the toolbar from the **Toolbar definition** list.

2. Click **Rename**. A **New Toolbar Name** box appears.
3. Type a name in the box and click **OK**.

PC-DMIS also allows another way to rearrange or delete items from any custom toolbar on the fly. This is by using the Shift key. Consider the following:

To delete or rearrange a toolbar's icons:

1. Press and hold the Shift key.
2. Click and drag an icon. The icon becomes movable.
3. If *rearranging* the icon, drag it to a new spot on the toolbar.
4. If *deleting* the icon, drag it outside of the toolbar's boundaries.
5. Release the mouse button. The change is made.

Since existing toolbars must have at least one icon on them, if you remove the last icon in this fashion, it only remains removed until you restart PC-DMIS. After restarting, the icon reappears.

To add a separator:

1. Press and hold the Shift key.
2. Click and drag an icon to move it.
3. Drag it a small distance to the right (not far enough to rearrange its position).
4. Release the mouse button. The software places a separator to the left of the icon.

To delete a separator:

1. Press and hold the Shift key.
2. Click and drag the icon to the right of the separator. The icon becomes movable.
3. Drag it on top of the separator to its left.
4. Release the mouse button.

To Create a Custom Toolbar

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and select the **Toolbars** tab.
2. Click **New** and select **Toolbar**. An input box appears requesting a toolbar name.

3. Type the name of the toolbar and click **OK**. Your new toolbar now appears in the **Toolbar Definition** list. You can now modify the toolbar to accept new toolbar icons (see "To Modify an Existing Toolbar").

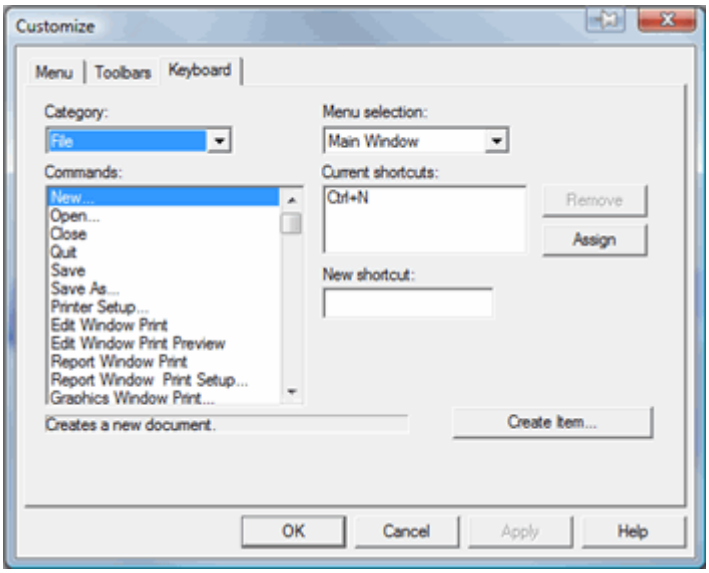
To Delete a Custom Toolbar

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and select the **Toolbars** tab.
2. Select the toolbar from the **Toolbar Definition** list.
3. Click the **Delete** button.
4. Click **Apply** to apply your changes.

Customizing Shortcut Keys

To customize the shortcut keys used for menu items and commands in PC-DMIS:

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option.
2. Select the **Keyboard** tab.



Customize Dialog Box - Keyboard Tab

You can use this tab to modify the shortcut keys used to call commands within PC-DMIS and add new shortcut keys for custom menu and toolbar items.

To Add Shortcut Keys to a Command

1. Access the **Customize** dialog box by selecting the **View | Toolbars | Customize** menu option, and select the **Keyboard** tab.

2. Select the command that you want to modify by using the **Category** and **Menu Selection** lists. Once you have selected a command in the **Commands** list, the current shortcut key(s), if any are assigned to that command, will be displayed in the **Current Shortcuts** box.
3. Highlight the command for which you will assign new shortcut keys.
4. Click in the **New Shortcut** box.
5. Type the keys you want to assign to this command. The keys you pressed appear as you type them.
6. Click the **Assign** button, and the new shortcut keys appear in the **Current Shortcuts** box with any other shortcut keys currently assigned to the command.
7. Click **Apply** to accept your changes.

As in the other tabs, you can use the **Create** button to first create a command and tie it to a custom program. You can then click **User Defined** from the **Menu Selection** list to select this command. All user-defined commands appear in the **Commands** list.

To Remove Shortcut Keys from a Command

1. To open the **Customize** dialog box, select the **View | Toolbars | Customize** menu option, and click the **Keyboard** tab.
2. Select the command you wish to modify with the **Category** and **Menu Selection** lists. Once you select a command in the **Commands** list, the software displays the current shortcut key or keys in the **Current Shortcuts** box if any are assigned to that command.
3. Select the command from which you want to remove the shortcut keys.
4. Select the shortcut key to remove.
5. Click the **Remove** button.
6. Click **Apply** to accept your changes.

Restoring the Default User Interface

To remove all intentional custom user interface changes at once, follow this procedure:

This removes changes you made with the **Customize** menu option. (**Customize** is visible when you right-click on the toolbar area.)

1. Close PC-DMIS.
2. Navigate to the directory where you installed PC-DMIS.
3. Access the directory that corresponds to your Windows user profile.
4. Delete the file named menu_xxx.dat where xxx refers to the three letter code for the language you're running. For English, delete menu_eng.dat.
5. Restart PC-DMIS. It will use the default user interface.

For information on how to customize menus, toolbars, and shortcut keys, see "Customizing Menus", "Customizing Toolbars", and "Customizing Shortcut Keys".

Docking and Undocking User Interface Elements

Some user interface elements (such as toolbars and some dialog boxes) can dock themselves into the Edit window or the toolbar area when you drag and drop the item on top of one of those areas. If you don't want this, hold down the Ctrl key while you release the mouse button. The toolbar or dialog box then hovers over the dockable background instead.

Some items, such as the Edit window, contain a right-click menu item (**Docking View**) that allows you to dock or undock that item. If the right-click menu item is not selectable, close any open dialog boxes and try again.

You may need to cascade (**Window | Cascade**) or tile (**Window | Tile Horizontally** or **Window | Tile Vertically**) your windows and then click and drag a window's title bar to float one over another.

Using Basic File Options

Using Basic File Options: Introduction

PC-DMIS provides you with many options to manipulate measurement routines and manage your essential files. As with most Windows-based programs, you can perform basic file management using standard Windows dialog boxes; these include creating, opening, copying, renaming, and deleting files. Additionally, through PC-DMIS you can perform more advanced operations, such as importing and exporting CAD data, or executing completed measurement routines.

This chapter specifically deals with Basic File Options. For more advanced options, see the "Using Advanced File Options" chapter.

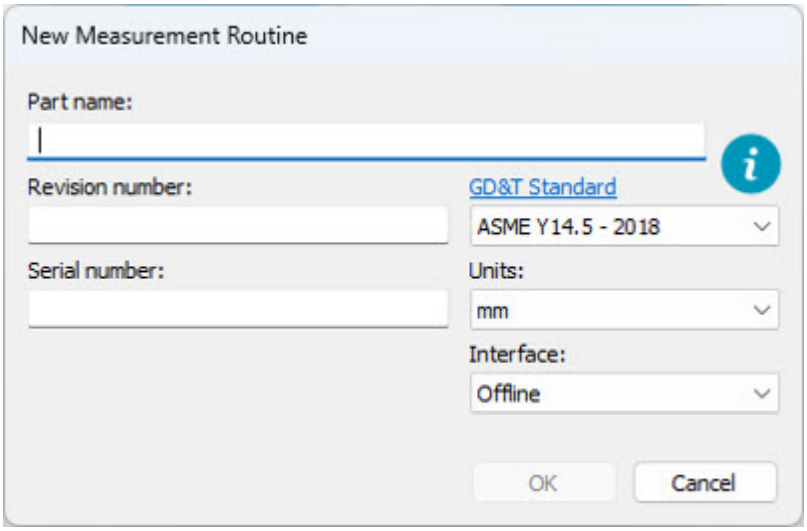
The options discussed in this chapter include:

- Creating New Measurement Routines
- Opening Existing Measurement Routines
- Saving Measurement Routines
- Working with Measurement Routine Archives
- Performing File Operations
- Connecting to Teamcenter
- Changing the Language
- Printing the Graphic Display Window
- Printing the Report Window
- Closing or Quitting a Measurement Routine
- Exiting PC-DMIS

Creating New Measurement Routines

If you don't have any existing measurement routines, you can create a new measurement routine with the **New Measurement Routine** dialog box. You can access this through the **File | New** menu option.

The **File | New** and **File | Open** menu options are unavailable while a probe change is in progress. This protects your hardware by preventing you from opening or creating a measurement routine that could power on a laser sensor and damage the probe mounted on the head.



New Measurement Routine Dialog Box

Here, you can create a new measurement routine. You can type a part name, revision number, serial number and select the GD&T standard, measurement units, and CMM interface type that you want to use. Once you click **OK**, PC-DMIS creates the measurement routine.

PC-DMIS only requires the **Part Name** box to have a value to create a new measurement routine. Information for the **Revision Number** and **Serial Number** boxes is optional.

By default, PC-DMIS selects the **ASME Y14.5 - 2018** standard from the **GD&T Standard** list. You can modify this to a different standard.

The GD&T standard affects Size or Geometric Tolerance commands in your measurement routine. The standard you select does not affect legacy dimensions. You can change the selected GD&T standard at any time from the **Geometric Tolerances** tab of the **Setup Options** dialog box. However, because the setting applies to the entire measurement routine, depending on the measurement routine's content at the time of change, it may behave differently.

For information on the **Geometric Tolerance** tab, see "Setup Options: Geometric Tolerances tab" in the "Setting Your Preferences" chapter.

For details on using Geometric Tolerances, see the "Using Geometric Tolerances" chapter in the PC-DMIS Core documentation.

For details on using legacy dimensions, see the "Using Legacy Dimensions" chapter in the PC-DMIS Core documentation.

You can also create new measurement routines from the Home page. You can even create measurement routines from templates so that the measurement routines come populated with those commands you use most frequently. You can also choose a destination folder for measurement routines that you create from a template. For information, see "The Home Page" and "Using Measurement Routine Templates" in the "Navigating the User Interface" chapter.

Creating a Measurement Routine Package

Select **File | Package** to create a .zip file that contains the measurement routine and its dependent files.

For more details about the Package Measurement Routine functionality, see the "Package Measurement Routine" chapter in the PC-DMIS Toolkit Module documentation.

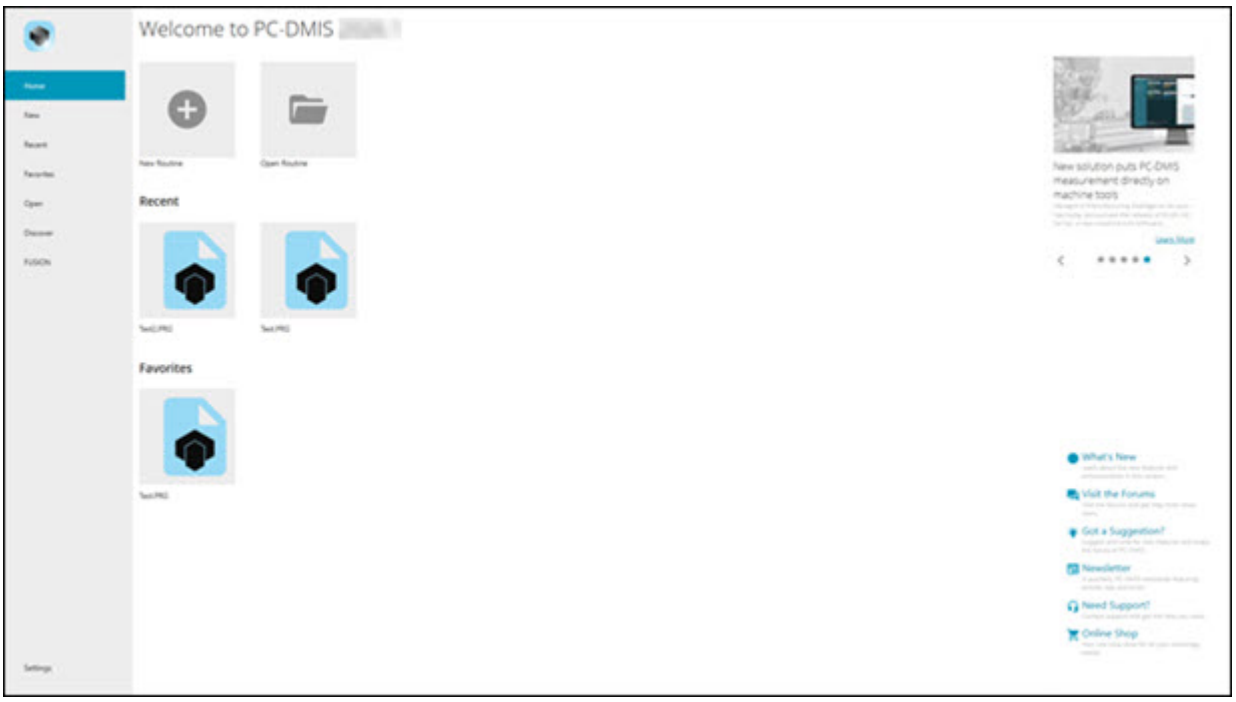
Opening Existing Measurement Routines

You can open measurement routines from the Home page or from the **File | Open** menu option.

The **File | New** and **File | Open** menu options are unavailable while a probe change is in progress. This protects your hardware by preventing you from opening or creating a measurement routine that could power on a laser sensor and damage the probe mounted on the head.

Using the Home Page

The Home page is discussed in the "Home Page" topic of the "Navigating the User Interface" chapter:

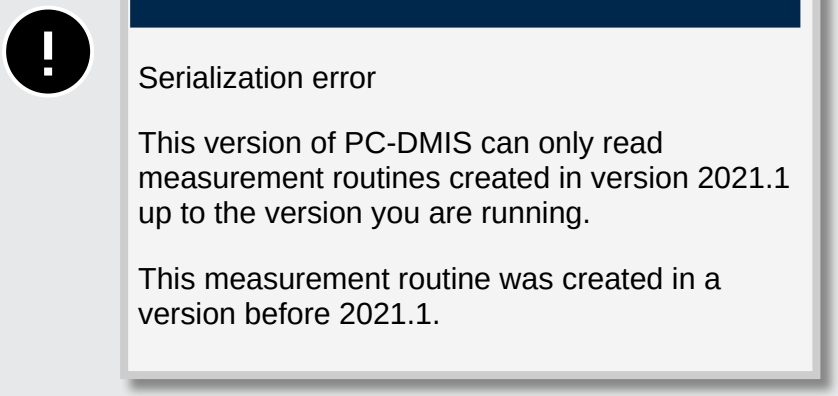


Home Page

Valid Versions

You can only open a measurement routine (.prg) made in valid versions of PC-DMIS. For this version of PC-DMIS, if you attempt to open a measurement routine created before PC-DMIS version 2021.1, PC-DMIS shows an error message.

Example error message when you try to open an invalid measurement routine file:



Using the File Open Dialog Box

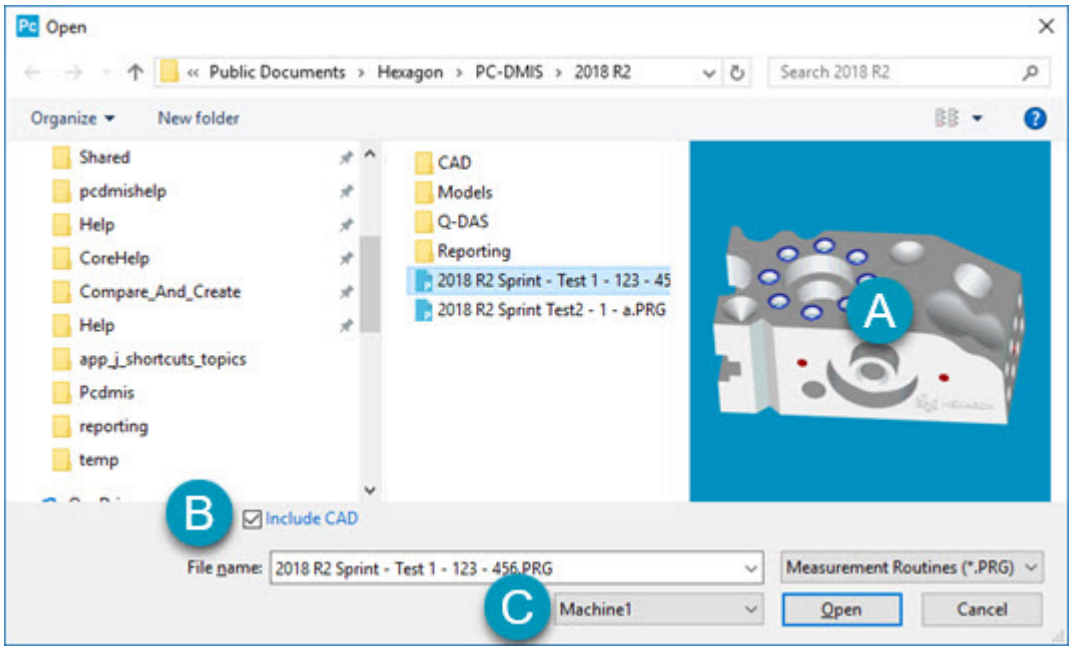
You can use the **Open** dialog box to locate and open a measurement routine.

To open the **Open** dialog box, select **File | Open**.

When you open a measurement routine that was created in a prior version, be aware of the following:

- If you quit the routine (**File | Quit**), any changes you made in this version are lost. The routine remains unchanged and you can still open it in the prior version of PC-DMIS.
- If you save (**File | Save**) or close (**File | Close**) the routine, or if you close PC-DMIS (close button in the top right of the application window), PC-DMIS saves the routine using this version's schema. Once that happens, it will no longer open in the prior version.
- If you wish to save your changes, and you need the routine to remain available in the prior version, choose **File | Save As** and be sure to select the prior version beneath the **Save as type** drop-down list. For more information, see "Save As".

This dialog box is a standard Windows **Open** dialog box with the following additions:



Open Dialog Box

A. **Preview** - This area of the dialog box shows the CAD image for the last view of your part in the Graphic Display window (without labels). If your part doesn't have the CAD data, the image displays measured geometry. If you set a thumbnail from "The Right Pane" of the Home page, PC-DMIS displays the thumbnail as the preview image.

B. **Include CAD** - This check box determines whether to load the CAD model when you open the measurement routine. If a CAD model is part of the measurement routine that you are opening, click the **Include CAD** check box. If you clear the check box, PC-DMIS does not load the CAD model when you open the measurement routine.

C. **Interface** - This list displays the online machines in your system as well as an **Offline** option.

- If PC-DMIS is running in Online mode, you can select the machine interface to use for this measurement routine (if you have multiple CMMs).
- If PC-DMIS is running in Offline mode, you can select the **Offline** machine option.

The **Open** dialog box automatically opens each time you start PC-DMIS so that you can open an existing measurement routine. However, if you want to turn off this feature, you can clear the **Show Startup Dialog** check box on the **General** tab of the **Setup Options** dialog box.

Probe Conversion Messages When Opening

When you open a measurement routine from a previous version, you may receive a warning message that asks you if you want to convert probe files for any probes used in the measurement routine to the latest format.

If you plan on using the probe with older versions of PC-DMIS, you may not want to update the probe file. However, note that until you update the probe file to the latest format, some newer functionality for that probe may not be available in this version of PC-DMIS.

If you continue to receive this message whenever you open your measurement routine, even after you click **Yes**, you should perform a **File | Save As** operation, and save the measurement routine so that it is compatible with this version of PC-DMIS. For more information, see the "Save As" topic.

Subroutine Names

If you open a measurement routine from a previous version that contains a subroutine with a name greater than 180 characters, the name of the subroutine is automatically shortened to the first 180 characters.

You can also open new measurement routines from the Home page. For information, see "The Home Page" in the "Navigating the User Interface" chapter.

Saving Measurement Routines

You can select the **File | Save** option to save any work that has been done in the current measurement routine. The first time you save a file, the **Save As** dialog box appears. This dialog box allows you to choose where to save your measurement routine (see the "Save As" topic for additional information).

If you have already saved a measurement routine to a previous version of PC-DMIS (from the **Save As** dialog box), PC-DMIS displays a message on subsequent saves. This

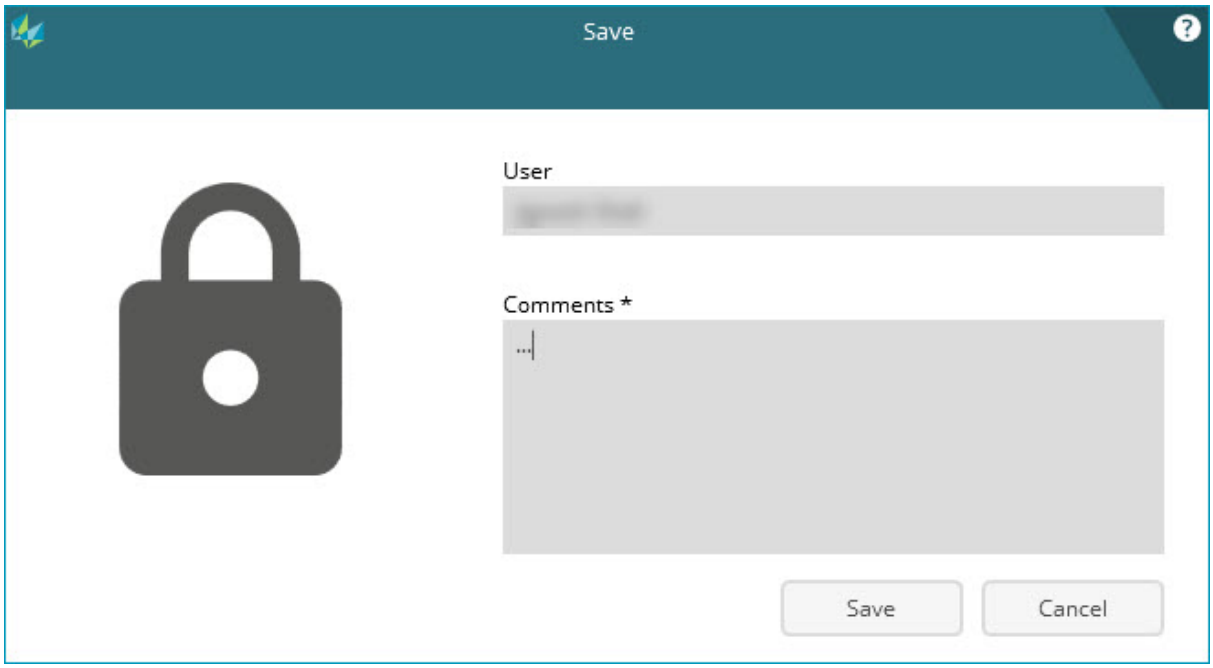
message asks if you want to save the measurement routine to the old version selected previously. You can choose these options:

- **Yes** - PC-DMIS saves the measurement routine to the old version.
- **No** - PC-DMIS saves the measurement routine to the current version.
- **Cancel** - PC-DMIS ends the save process without saving.

You cannot save a certified measurement routine to an older version of PC-DMIS.

If you want to stop displaying this message, see "Warnings" in the "Setting Your Preferences" chapter.

When you save a certified measurement routine, PC-DMIS displays the **Save** dialog box.



Save Dialog Box

The **Save** button saves all the changes that you made to the measurement routine with comments in the .protect file.

For information on certified measurement routines, see "Certify".

Your measurement routines are automatically archived during certain events. For more information, see "Working with Measurement Routine Archives".

Save As

When you select the **File | Save As** option, or save a measurement routine for the first time, the **Save As** dialog box appears.

- You can use the **File name** option in this dialog box to save the current measurement routine under a new file name.
- You can also save the measurement routine under an older version of PC-DMIS.

To save the measurement routine, click the **Save** button.

To close the dialog box without saving the measurement routine, click the **Cancel** button.

Certified Measurement Routine

When you click the **Save** button on the **Save As** dialog box to save a certified measurement routine to the current version of PC-DMIS, the software displays a message. The message asks if you want to keep the certification on the new measurement routine before the **Save As** dialog box closes.

You can choose these options:

- **Yes** - PC-DMIS saves the measurement routine and keeps the certification on the new measurement routine. It also generates a .protect file and continues to track the changes from the previous .protect file.
- **No** - PC-DMIS saves the measurement routine but doesn't keep the certification on the new measurement routine.
- **Cancel** - PC-DMIS ends the save as process without saving.

Saving to an Older Version of PC-DMIS

You cannot save a certified measurement routine to an older version of PC-DMIS.

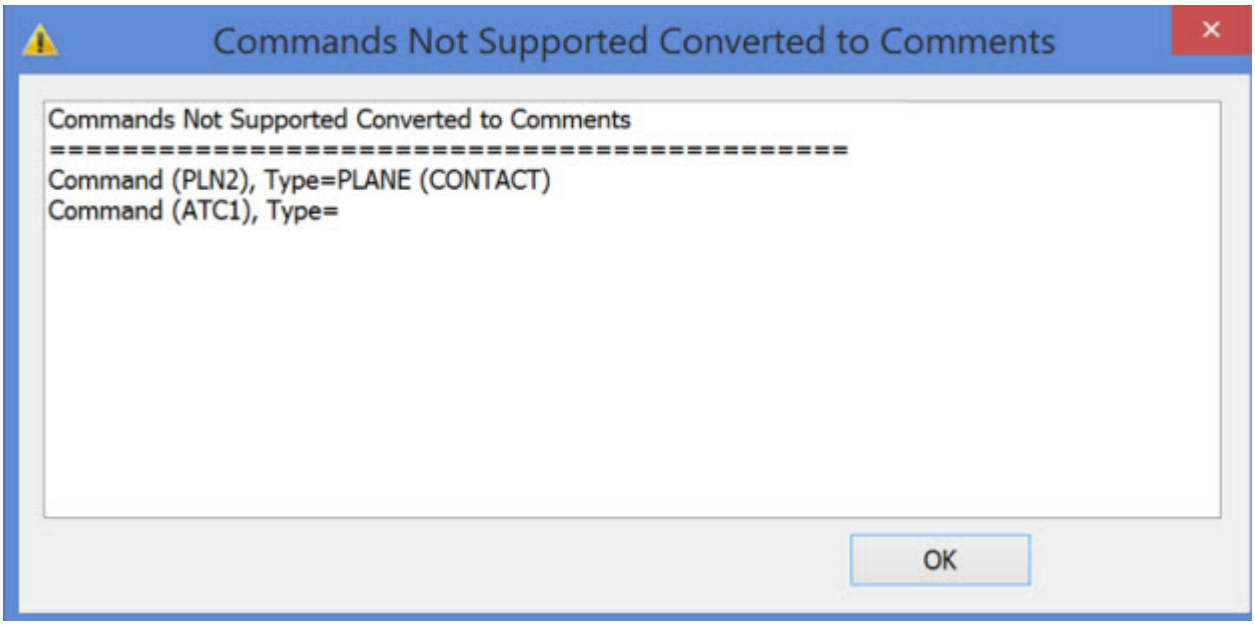
From the list of version numbers, you can choose to save the measurement routine in a format that the current version and previous versions of PC-DMIS can use. Available versions include 2021.1 and above.

- If you try to save a measurement routine to an older version of PC-DMIS, and if the measurement routine contains commands that the older version does not support, PC-DMIS displays a caution that it will convert those commands into comments.
- If you try to save a certified measurement routine to a previous version of PC-DMIS, PC-DMIS displays a message that you can only save certified measurement routines to the current version.

To cancel saving this measurement routine, click **Cancel**.

To continue to save the measurement routine to the older version, click **Save**.

PC-DMIS converts the non-supported commands into comments and displays the **Commands Not Supported Converted to Comments** dialog box. This dialog box lists the non-supported commands that PC-DMIS converted to comments. For example:



Commands Not Supported Converted to Comments Dialog Box

You must click **OK** to acknowledge that you have noted this message.

If the `SaveAsWarningLogEnabled` entry is enabled, PC-DMIS automatically creates a log file that contains the contents of the message. The log file is named:

<name>_Save_As_Log_<YYYYMMDDHHMMSS>.log

Where <name> is the name of the measurement routine, and <YYYYMMDDHHMMSS> is the date and time.

YYYY is the year.

MM is the month.

DD is the day.

HH is the hour.

MM are the minutes.

SS are the seconds.

The log file is in the same folder as the measurement routine.

You can disable the creation of the log file. For help, see the "SaveAsWarningLogEnabled" topic in the Settings Editor documentation.

About Schema Numbers and Saving to an Older Version

Each new version of PC-DMIS has new capabilities and stores new commands into a measurement routine that older versions cannot understand. Thus, each measurement routine has an internal schema number assigned to it that's associated with a specific version of PC-DMIS. By default, a measurement routine is assigned the current version's schema number when you choose **File | Save**. This number ensures that only that version (or a later version) can open that measurement routine.

For example, if you have a measurement routine that was saved in version 2011, it will have the schema number of 5008. If you try to load that measurement routine into version 2010, it will be incompatible because the older version cannot handle changes that were made in PC-DMIS 2011.

You can, of course, choose to save the measurement routine back to the version 2010 format. This drops the internal schema number for the measurement routine down to 4555, which enables you to then open it in version 2010. However, the newer commands won't be used. Instead, PC-DMIS stores the unsupported commands inside of DOCUMENT comments.

For more information, see "$$Document" in the "Using the Edit Window" chapter.

Custom reports, templates, label templates, and other reporting modifications are NOT backward compatible.

Saving a CAD File as a Reference

To save a CAD file as a reference to the original CAD file, select the **Save CAD as reference** check box. You can use this option to reference your CAD model instead of copying a new one. This action can help save disk space.

This option becomes available for selection whenever you have a CAD model loaded for your measurement routine and access the **Save As** dialog box.

If the software cannot find the original referenced CAD file when you try to open the measurement routine (due to a CAD file rename, deletion, or relocation), PC-DMIS prompts you to locate the original CAD file.

Working with Measurement Routine Archives

PC-DMIS provides some basic measurement routine archiving. This archiving replaces the automatic file save functionality that existed in some older versions of PC-DMIS.

Two copies of your measurement routine are automatically created in the background during these events:

- <name>.prg~ is created just before you save the measurement routine (where <name> is the measurement routine name).
- <name>.prg^ is created just before you load the measurement routine (where <name> is the measurement routine name).

> The above PRG~ and PRG^ backup files have the hidden attribute set and won't be visible in Windows Explorer unless the "Show hidden files" option is set.

If you try and open a corrupted measurement routine, PC-DMIS asks if you want to open the copy made before the last save (<name>.prg~). If you choose **Yes**, PC-DMIS renames the corrupt measurement routine to <name>.prg.tmp (by default, these files have the hidden attribute set to be hidden and won't be visible in Windows Explorer unless the "Show hidden files" option is set correctly). It renames <name>.prg~ to the current measurement routine name. If that backup was also corrupted, PC-DMIS asks if you want to restore from the backup that was made just before you loaded the measurement routine during your last session (<name>.prg^). It then follows the same process of renaming and copying.

> As with any backup/restore process, you may lose your latest changes when restoring.

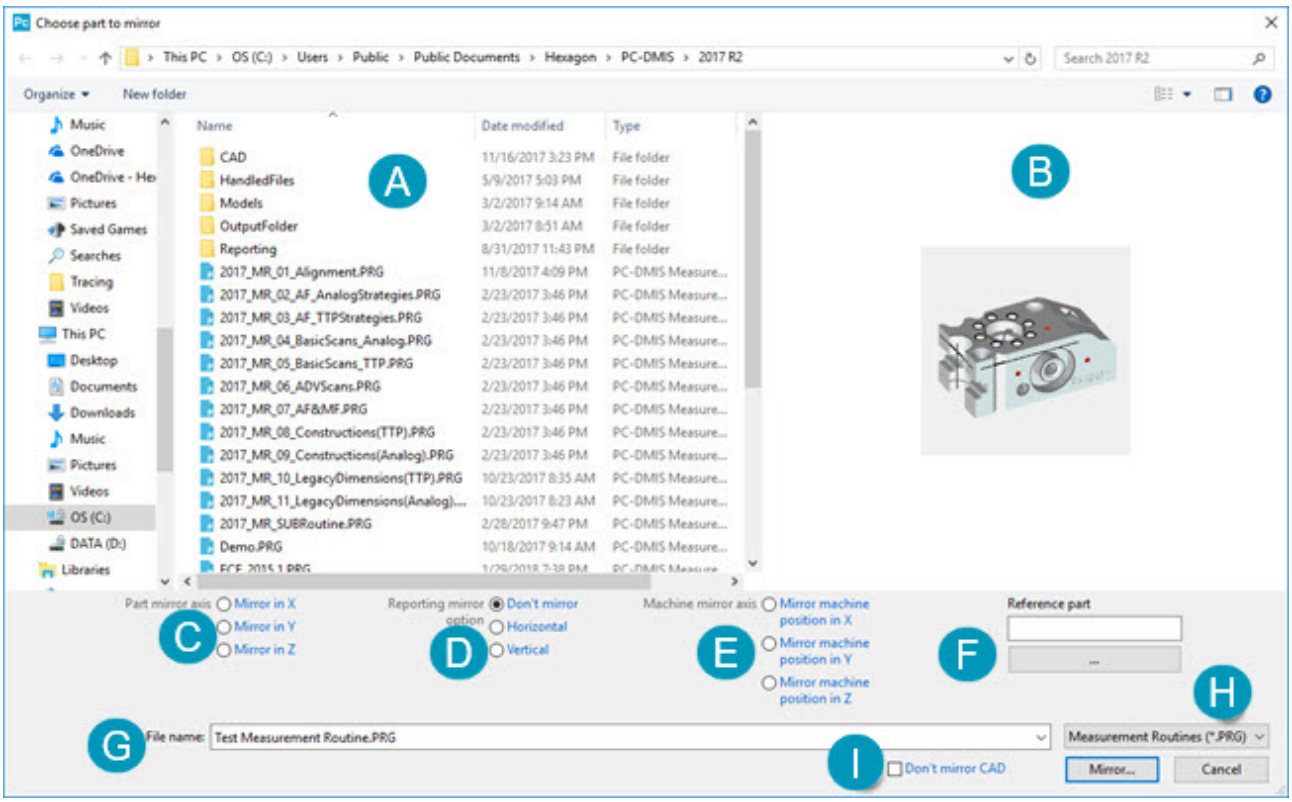
You can disable or enable this functionality by using the `DocumentRecovery` entry in the PC-DMIS Settings Editor.

Performing File Operations

Using PC-DMIS's file operations, you can mirror, copy, delete, rename, and certify measurement routine files.

Mirror

The **Mirror** command (**File | Operations | Mirror**) lets you create a mirror image copy in the X, Y, or Z axis of a measurement routine. The command opens the **Choose part to mirror** dialog box.



Choose part to mirror Dialog Box

You can only mirror measurement routine (.prg) and CAD files made in valid versions of PC-DMIS. For this version of PC-DMIS, if you attempt to mirror either of these file types created before PC-DMIS version 2021.1, PC-DMIS shows an error message.

Example error message when you try to mirror an invalid measurement routine file:

PC-DMIS Message

X Mirrored Copy of <measurement routine file> (Schema: 1215).

Serialization error

This version of PC-DMIS can only read measurement routines created in version 2021.1 up to the version you are running.

This measurement routine was created in a version before 2021.1.

Where <measurement routine file> is the name and location of the file you are trying to open.

The following description describes the items available for selection on the dialog box:

A. **Measurement Routines** - This area displays the available measurement routines in the current directory.

B. **Preview** - This area shows the CAD image for the last view of your part in the Graphic Display window (without labels). If your part doesn't have any CAD data, the image displays measured geometry. This area only shows images for .prg files.

C. **Part mirror axis** - This area defines the axis in which you want to mirror the measurement routine:

- **Mirror in X** - This option mirrors the part in the X axis.
- **Mirror in Y** - This option mirrors the part in the Y axis.
- **Mirror in Z** - This option mirrors the part in the Z axis.

D. **Reporting mirror option** - This area defines how PC-DMIS should mirror CAD images in reports:

- **Don't mirror** - This option does not mirror at all.
- **Horizontal** - This option mirrors CAD images horizontally.
- **Vertical** - This option mirrors CAD images vertically.

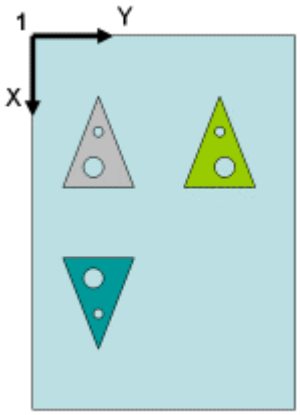
This area affects reporting objects, such as CadReportObjects, CadImageObjects, Analysis objects, and SectionCutObjects, as well as any defined viewsets in the measurement routine.

E. **Machine mirror axis** - This area defines the orientation of the mirrored part on the virtual machine:

- **Mirror machine position in X** - This option mirrors the virtual machine's X axis.
- **Mirror machine position in Y** - This option mirrors the virtual machine's Y axis.
- **Mirror machine position in Z** - This option mirrors the virtual machine's Z axis.

PC-DMIS mirrors the part at the same location as the original part. You can use the **Machine mirror axis** area to define the orientation of the mirrored part on the virtual machine. You only need to shift the part on the machine to the place it should get measured.

For a description of how these options position your part, consider this diagram:



1 - The machine's origin

- Original part

- Part mirrored in Y with mirror machine position in Y

- Part mirrored in Y with mirror machine position in X

F. **Reference part** - This box pulls out any quick fixture alignment data from the reference part and uses that information in the mirroring operation.

   If you use this area to define a reference part, then PC-DMIS does not use the **Machine mirror axis** area. Instead, the location and orientation of the mirrored part are loaded from the existing (and already mirrored) reference measurement routine.

   This option is especially useful if you have multiple measurement routines that belong to the same physical part and need to get mirrored. In this case, you would only need to define the location and orientation of the mirrored part in the first mirrored measurement routine. You could then select the first mirrored measurement routine as the reference part to mirror other measurement routines.

G. **File name** - This box defines the measurement routine to mirror.

H. This file type list filters the display of the current directory to only show measurement routine files (.PRG).

I. **Don't mirror CAD** - This check box determines whether the resulting CAD is mirrored. If you select this check box, the resulting CAD is not mirrored. The check box is initially cleared when you first start PC-DMIS. Afterward, this check box retains the state of what was applied in the last mirroring.

   You can use the PC-DMIS Settings Editor to change the `DoNotMirrorCAD` entry's value in the **Option** section.

To Mirror a Measurement Routine:

1. If you use an infinite wrist in the measurement routine, you should first set the `AlternateTipMirror` entry to TRUE. This entry is in the **USER_Option** section of the PC-DMIS Settings Editor. This entry helps PC-DMIS choose the correct AB angle in cases where the probe head offers more than one possible combination of AB angles to produce the mirrored tip vector.

2. Select **File | Operations | Mirror** to open the **Choose part to mirror** dialog box.

3. Select the measurement routine file to mirror.

4. From the **Part mirror axis** area, select the axis in which you want to mirror.

5. From the **Reporting mirror option** area, choose how you want to mirror CAD images in reports.

6. From the **Machine mirror axis** area, choose how you want to position the mirrored part on a virtual machine displayed in the Graphic Display window.
7. If you have an already mirrored reference part, use the **Browse** button in the **Reference part** area to select that part. This box pulls out any quick fixture alignment data from the reference part and uses that information in the mirroring operation.
8. Click the **Mirror** button to open the **Save mirrored part as** dialog box with the file name of the measurement routine that you selected to mirror.
9. Navigate to the directory where you want to save the measurement routine and click the **Save** button. PC-DMIS mirrors the measurement routine and saves it in that directory. Note that it may take a little while to copy the necessary files.

PC-DMIS also attempts to mirror your probe's AB tip angles. If your probe does not have the exact mirrored tip angles already defined and calibrated, PC-DMIS may choose the closest calibrated tip angles. It chooses closely-calibrated tip angles if the calibrated tip angles exist and are within the wrist angle amount specified in the **Wrist warning delta** box on the **Part/Machine** tab of the **Setup Options** dialog box. If no closely-calibrated AB tip angles exist, then PC-DMIS creates perfectly-mirrored yet non-calibrated AB tip angles. See the "Setup Options: Part/Machine tab" topic in the "Setting Your Preferences" chapter.

Copy

The **Copy** command allows you to copy all of the files that are associated with a measurement routine.

To copy a measurement routine, perform the following steps:

1. Select the **File | Operations | Copy** menu item to open the **Copy part files from** dialog box.
2. Navigate through the directory structure and select a measurement routine.
3. Click the **Copy** button to display the **Copy part files to** dialog box.
4. Click on the measurement routine that you want to copy so that its file name and extension appear in the **File name** box.
5. Navigate to the folder to which you want to copy the measurement routine.
6. Click the **Save** button.

These files are copied:

- .prg
- .cad

- Any file with the same base name and an extension in the range .001, .002, …, .999
- <*name of measurement routine*>.MiniRoutines.xml, if it exists
- <*name of measurement routine*>.QdasData.xml, if it exists

Delete

The **Delete** command allows you to delete all of the files that are associated with a measurement routine.

To delete a measurement routine:

1. Select the **File | Operations | Delete** menu item to open the **Delete part files** dialog box.
2. Choose the measurement routine to delete.
3. Click the **Delete** button. PC-DMIS asks if you are sure that you want to send the files to the Recycle Bin.
4. Click the **Yes** button.

These files are deleted:

- .prg
- .cad
- Any file with the same base name and an extension in the range .001, .002, …, .999.
- <*name of measurement routine*>.MiniRoutines.xml, if it exists
- <*name of measurement routine*>.QdasData.xml, if it exists

Deleted files are sent to the Recycle Bin.

Rename

The **Rename** command allows you to rename all of the files that are associated with a measurement routine.

You can only rename measurement routine (.prg), probe (.prb), and report (.rpt) files made in valid versions of PC-DMIS. For this version of PC-DMIS, if you attempt to rename any of these file types created before PC-DMIS version 2021.1, PC-DMIS shows an error message.

Example error message when you try to rename an invalid measurement routine file:

PC-DMIS Message

<measurement routine file>: Could not Rename File.

Unable to rename this Measurement Routine because it was created with a PC-DMIS version before 2021.1.

Where <measurement routine file> is the name and location of the file you are trying to open.

To rename a measurement routine:

1. Select the **File | Operations | Rename** menu item to open the **Rename part files from** dialog box.
2. Select the measurement routine to rename.
3. Click the **Rename** button to open the **Rename part files to** dialog box.
4. Type in the new name for the file in the **File name** box.
5. Click the **Save** button. The name change is applied immediately.

These files are renamed:

- .prg
- .cad
- Any file with the same base name and an extension in the range .001, .002, …, .999
- <*name of measurement routine*>.MiniRoutines.xml, if it exists
- <*name of measurement routine*>.QdasData.xml, if it exists

Certify

The **Certify** command lets you certify measurement routines and add related comments. You can select the **File | Operations | Certify** menu option, or from the **File Operations** toolbar, click the **Certify** button ( ).

A certified measurement routine is one validated by a programmer (a member of the PC-DMIS-Programmer group). This measurement routine meets internal requirements for use in production. The Protect Viewer application lets you control and track modifications to this measurement routine, such as:

- Any internal organizational requirements
- Specific externally imposed process certification requirements (such as ISO and FDA)

The Protect Viewer installer, included with the PC-DMIS installer, creates the **PC-DMIS-User** and **PC-DMIS-Programmer** groups in the **Local Users and Groups** folder on your computer.

- The **PC-DMIS-User** group's users can submit or cancel the submission of measurement routines to the programmer for certification. They can also edit measurement routines.
- The **PC-DMIS-Programmer** group's users can edit, certify, and reject measurement routines. They cannot edit the measurement routine created by the **PC-DMIS-User** group's users.

If you are not a member of either group, you can open, view, and execute certified measurement routines in Protected Mode. However, you cannot certify or edit certified measurement routines.

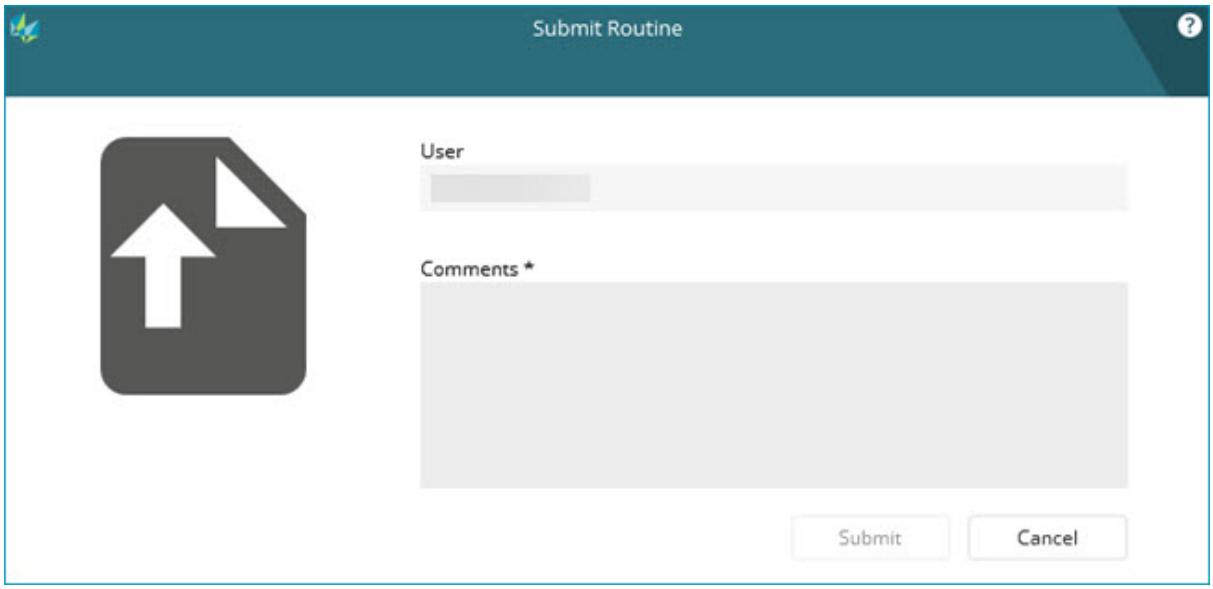
When the PC-DMIS Protect license option is enabled, operators (users who are not members of the PC-DMIS-Programmer or PC-DMIS-User group) can execute only certified measurement routines. To let operators also execute uncertified measurement routines, set the `RestrictCertifiedMRExecutionToOperators` entry in the **Option** section of the PC-DMIS Settings Editor to **FALSE**.

If you are a member of both the **PC-DMIS-User** and **PC-DMIS-Programmer** groups, PC-DMIS considers you as a user.

To modify the **PC-DMIS-User** or **PC-DMIS-Programmer** groups (such as adding or removing users), contact your computer administrator.

PC-DMIS-User Specific Dialog Boxes

- **Submit Routine** - This dialog box lets you submit a measurement routine and generate a .protect file. From the **File Operations** toolbar, click the **Submit** button ( ) to open the **Submit Routine** dialog box.



Submit Routine Dialog Box

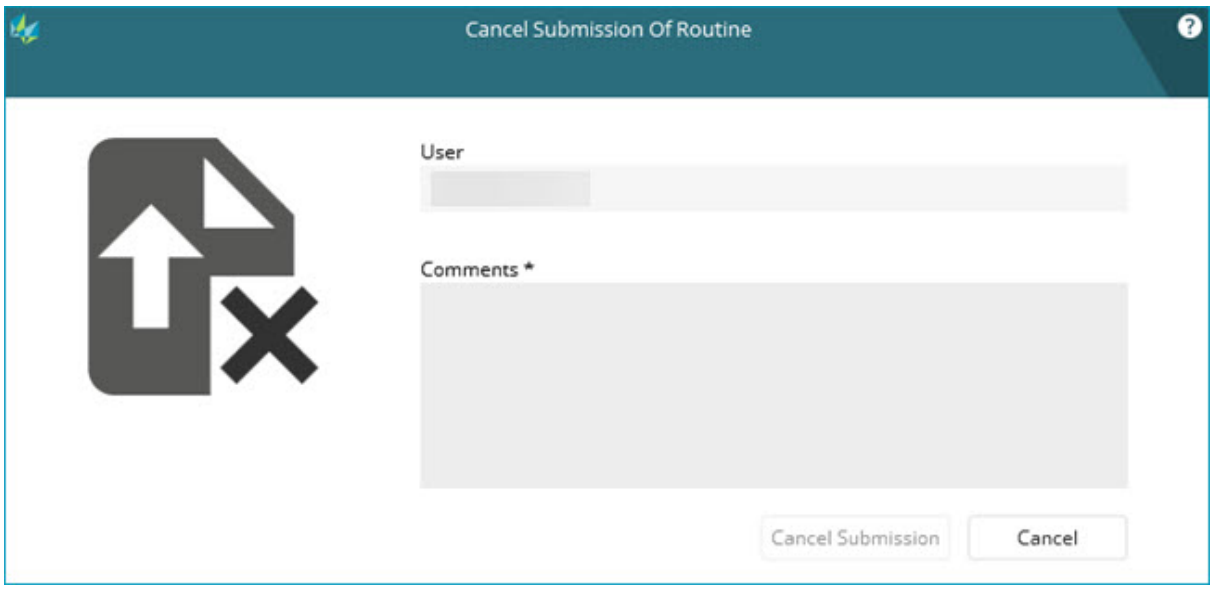
User - This box displays your network username.

Comments - This box lets you add comments before you submit a measurement routine for certification.

Submit - This button submits and saves the measurement routine, closes the dialog box, and generates a .protect file in the same folder as the measurement routine. The file name is <measurement routine file name>.PRG.protect, where <measurement routine file name> is the name of the measurement routine file. The default folder location is: "C:\Users\Public\Documents\Hexagon\PC-DMIS\2026.2".

Cancel - This button ignores any changes and closes the dialog box.

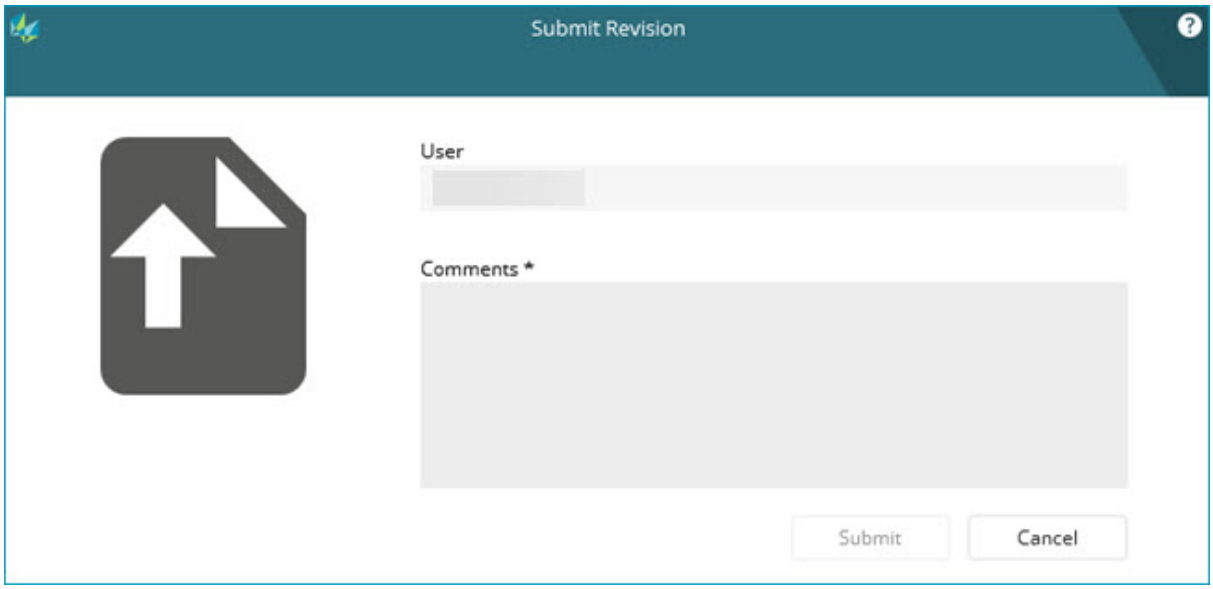
- **Cancel Submission Of Routine** - This dialog box lets you cancel the submission of a measurement routine. From the **File Operations** toolbar, click the **Cancel Submission** button ( ) to open the **Cancel Submission Of Routine** dialog box.



Cancel Submission Of Routine Dialog Box

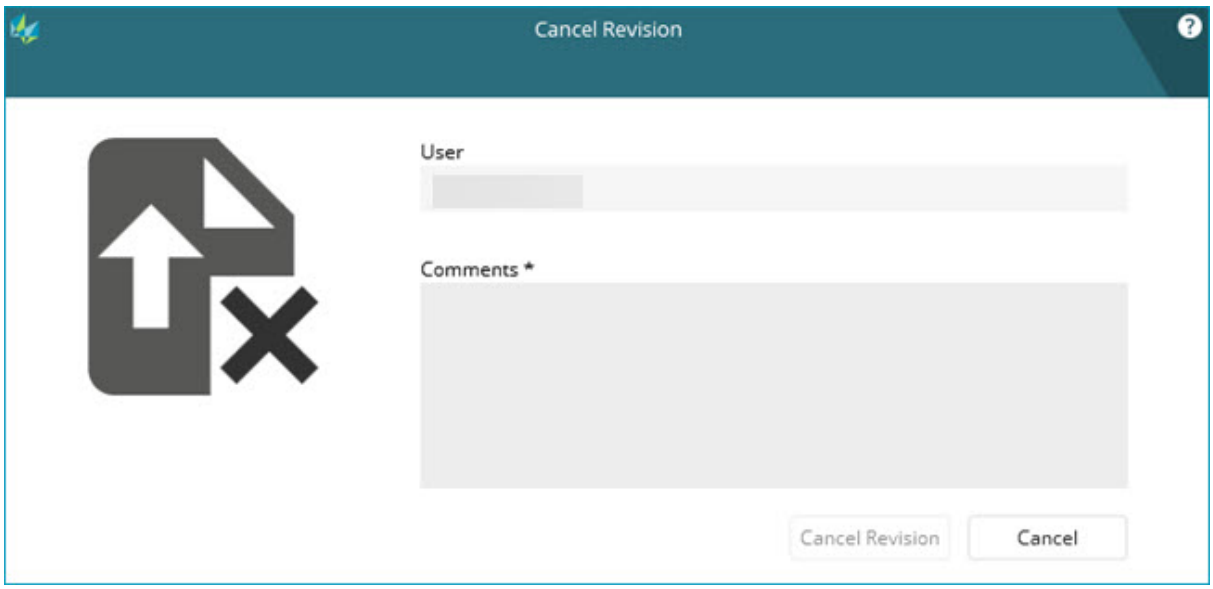
Cancel Submission - This button cancels the submission of the measurement routine. The user must resubmit the measurement routine for certification.

- **Submit Revision** - This dialog box lets you submit a revised certified measurement routine to a member of the PC-DMIS-Programmer group for recertification. From the **File Operations** toolbar, click the **Submit** button ( ) to open the **Submit Revision** dialog box.



Revision Dialog Box

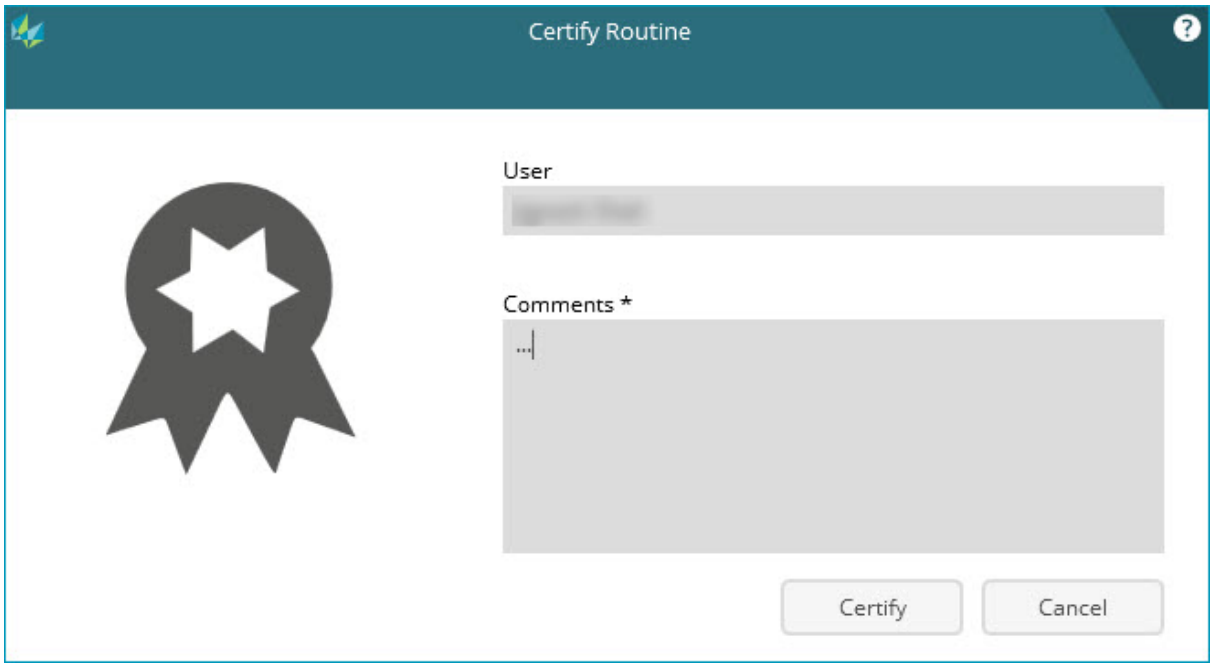
- **Cancel Revision** - This dialog box lets you cancel the submission of a revised measurement routine for recertification. From the **File Operations** toolbar, click the **Cancel Submission** button ( ) to open the **Cancel Revision** dialog box.



Revision Dialog Box

PC-DMIS-Programmer Specific dialog boxes

- **Certify Routine** - This dialog box lets you certify the measurement routine and generate a .protect file if you certify it without a submission from a user. Select the **File | Operations | Certify** menu option or, from the **File Operations** toolbar, click the **Certify** button ( ) to open the **Certify Routine** dialog box.



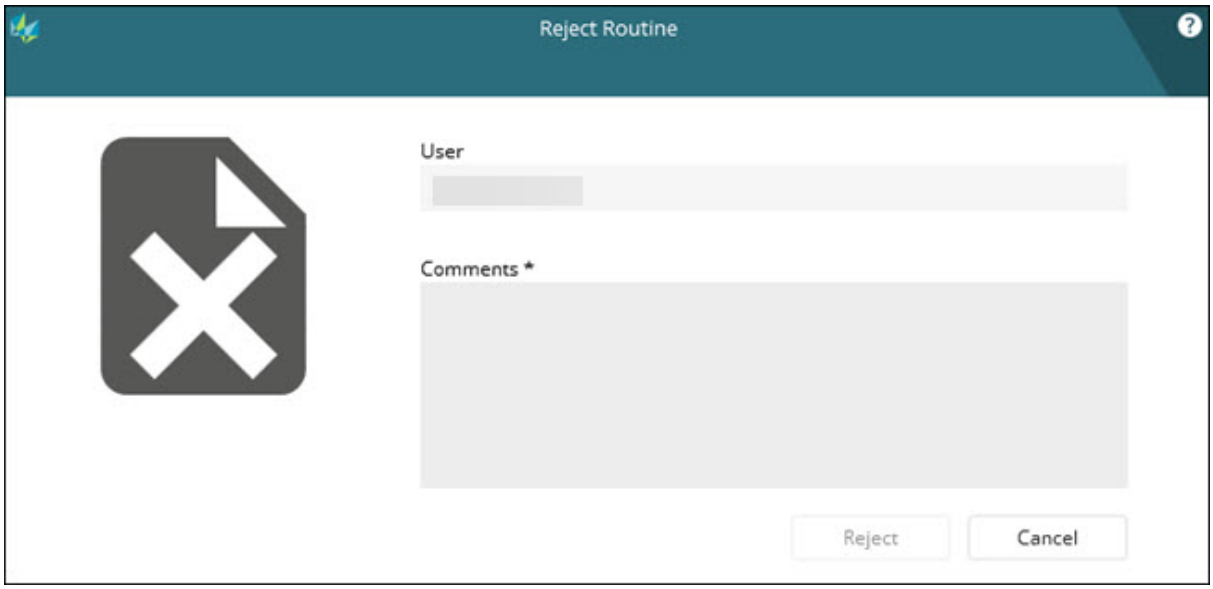
Certify Routine Dialog Box

Certify - This button certifies the measurement routine, saves it, and closes the dialog box. If you certify the measurement routine without a submission from the user, clicking this button generates a .protect file in the same folder as the measurement routine. The file name is <measurement routine file name>.PRG.protect, where <measurement routine file name> is the name of the

measurement routine file. The default folder location is: "C:\Users\Public\Documents\Hexagon\PC-DMIS\2026.2".

- **Reject Routine** - This dialog box lets you reject a submitted measurement routine.

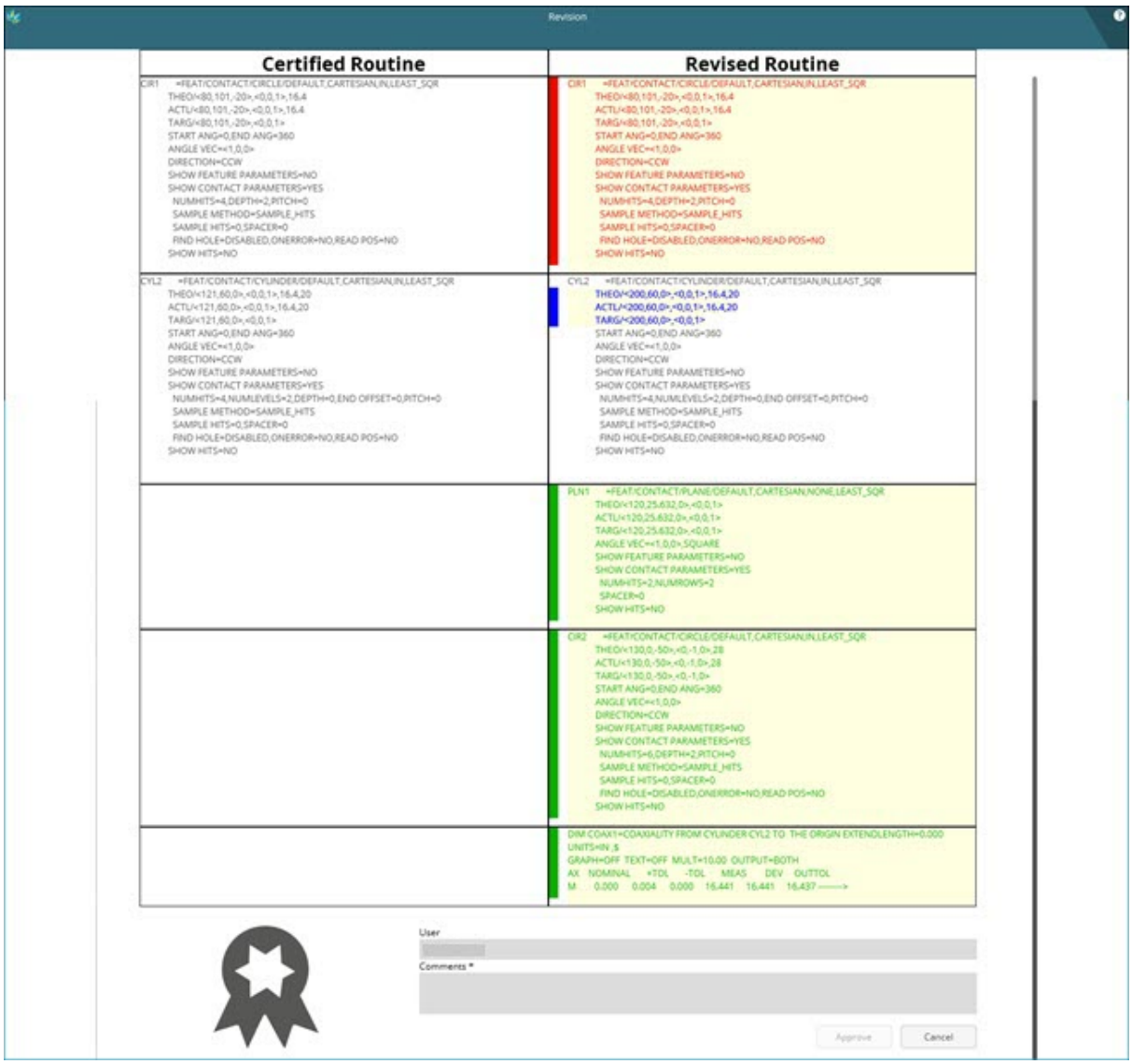
  From the **File Operations** toolbar, click the **Reject** button ( ) to open the **Reject Routine** dialog box. This button is only available when a user submits a measurement routine for certification.



Reject Routine Dialog Box

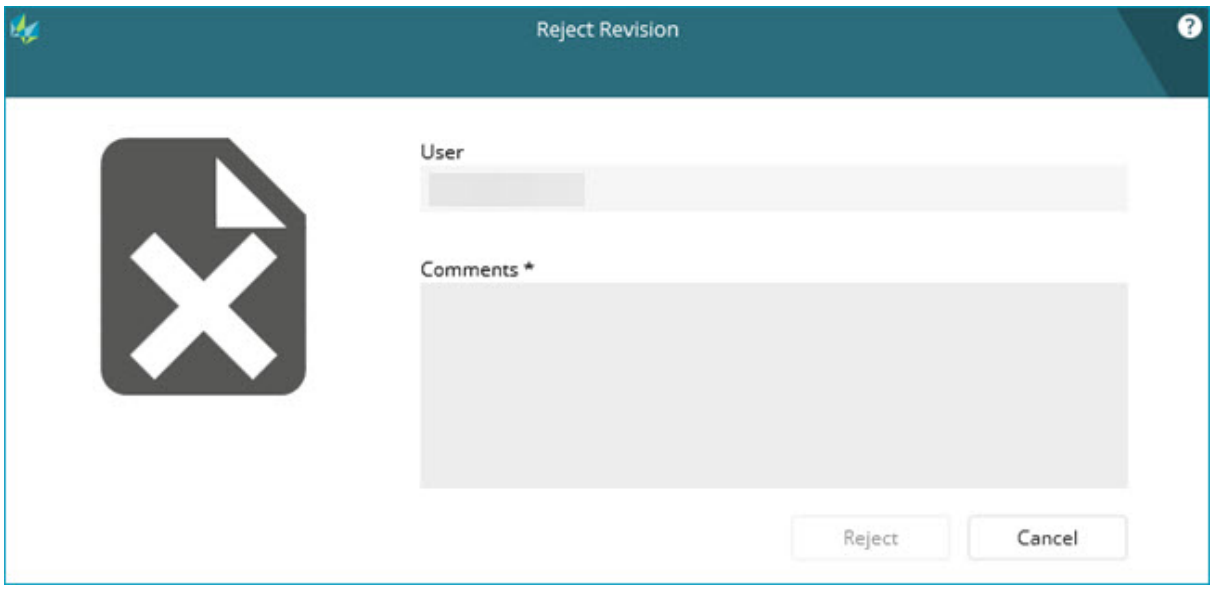
Reject - This button rejects the submission of the measurement routine. The user needs to resubmit the measurement routine for certification.

- **Revision** - This dialog box displays the **Certified Routine** and **Revised Routine** columns to review the changes made after certification. From the **File Operations** toolbar, click the **Recertify** button ( ) to open the **Revision** dialog box.



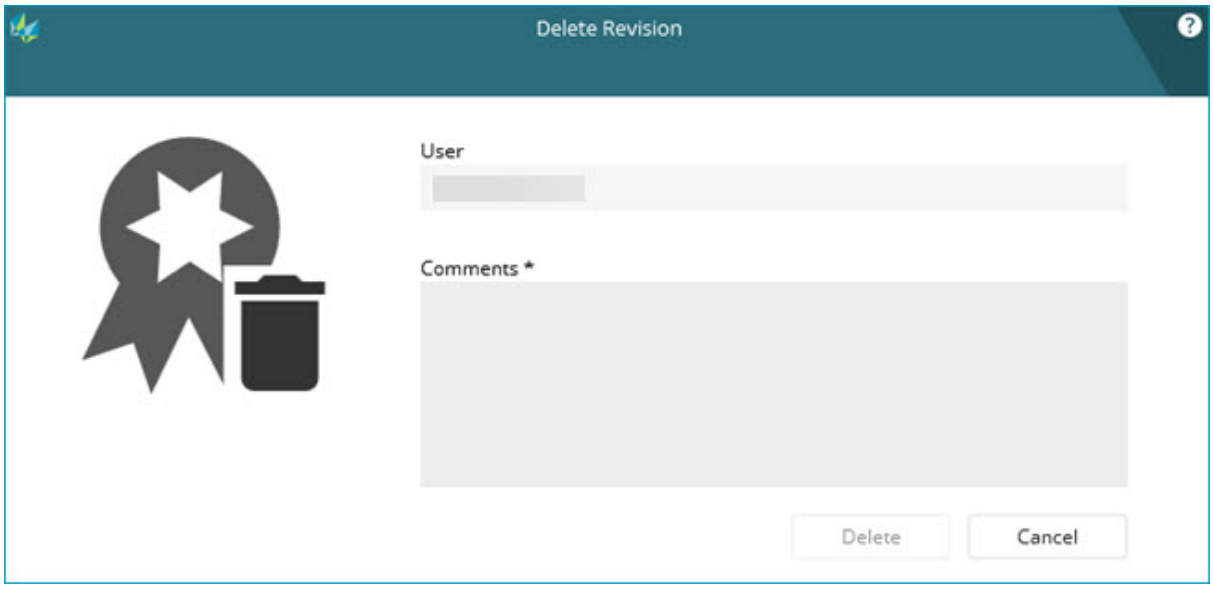
Revision Dialog Box

- **Reject Revision** - This dialog box lets you reject the changes made after the certification without deleting them. From the **File Operations** toolbar, click the **Reject** button ( ) to open the **Reject Revision** dialog box.



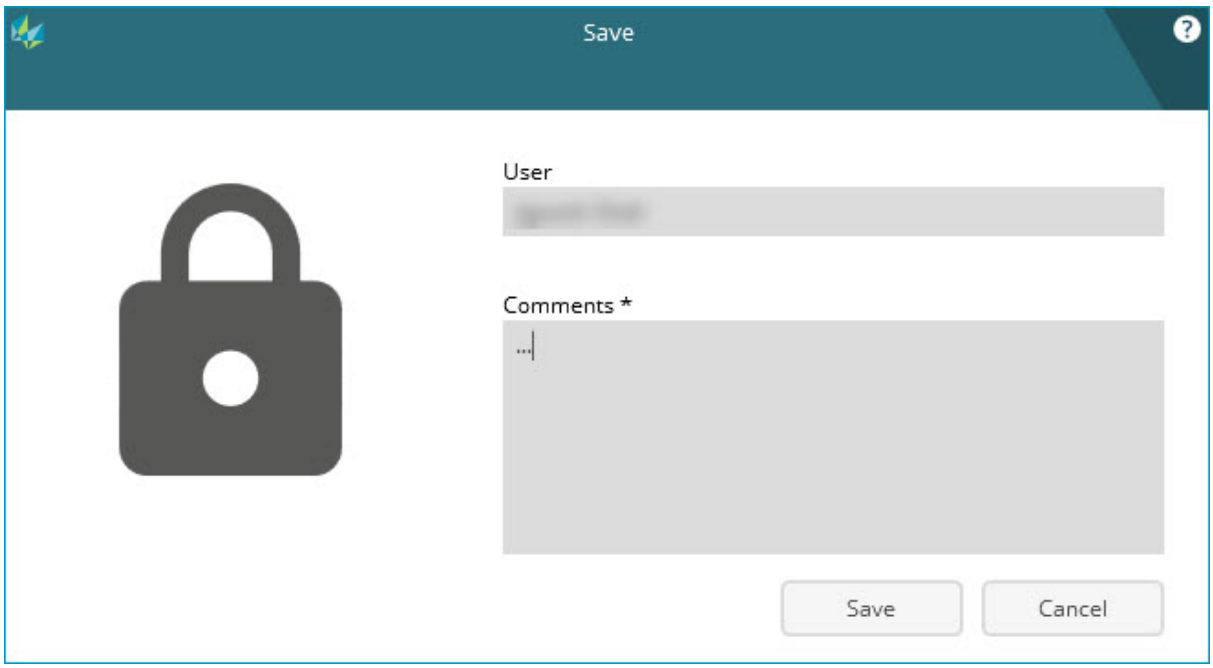
Revision Dialog Box

- **Delete Revision** - This dialog box lets you delete the changes made after the certification. From the **File Operations** toolbar, click the **Delete** button ( ) to open the **Delete Revision** dialog box.



Delete Revision Dialog Box

- **Save** - This dialog box appears only for the user who can edit the measurement routine after certification. After certification of the measurement routine, PC-DMIS displays the **Save** dialog box whenever you save any changes. This dialog box only appears if you want to save the changes to the measurement routine related to a feature or dimension.



Save Dialog Box

Save - This button saves the comments and changes to the .protect file.

When you open a certified measurement routine in PC-DMIS 2018 R2 and earlier, the certification is not available. The measurement routine opens like a usual measurement routine.

Adding a User or a Group to the PC-DMIS-Programmer Group

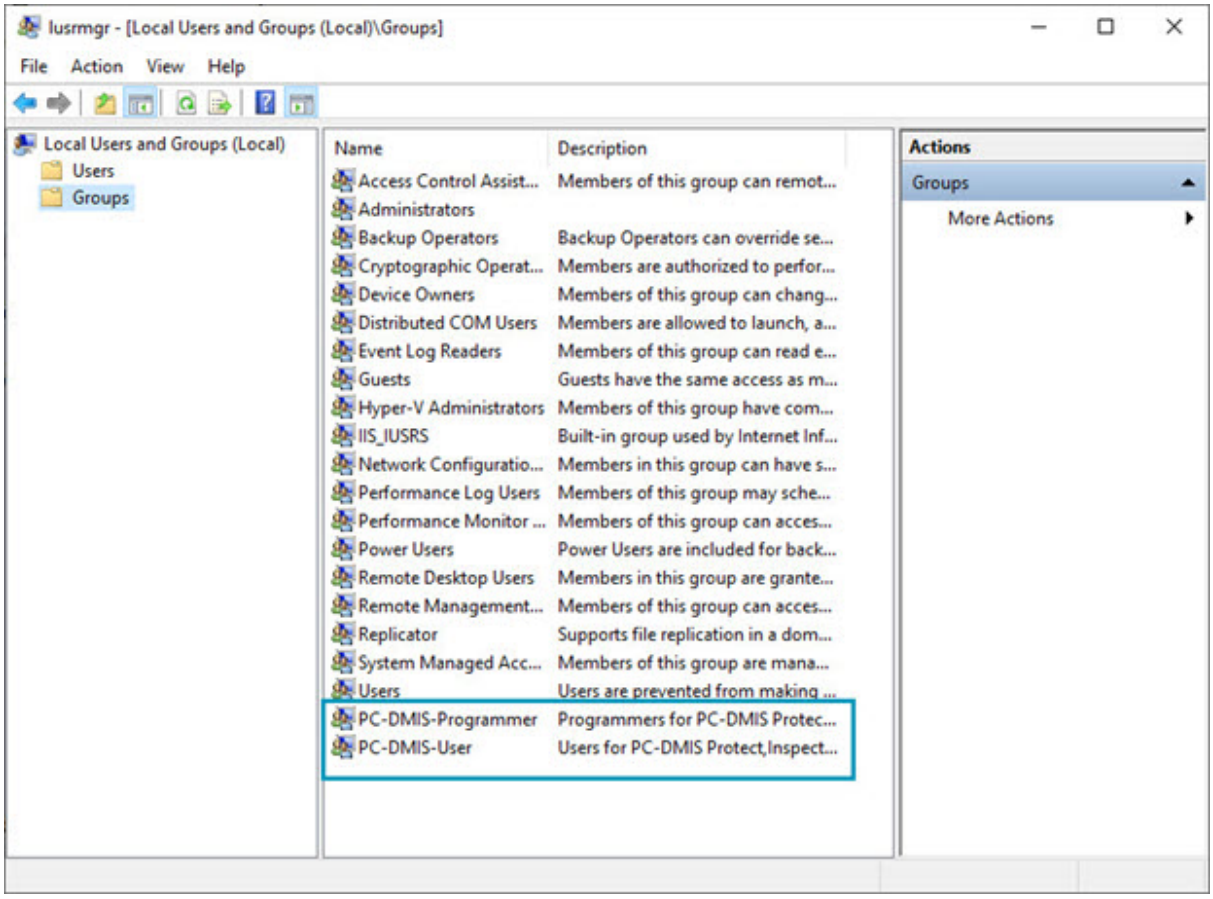
To add a user or a group to the PC-DMIS-Programmer group, follow these steps:

Adding a user or a group of users to the PC-DMIS-Programmer group makes them a metrology administrator.

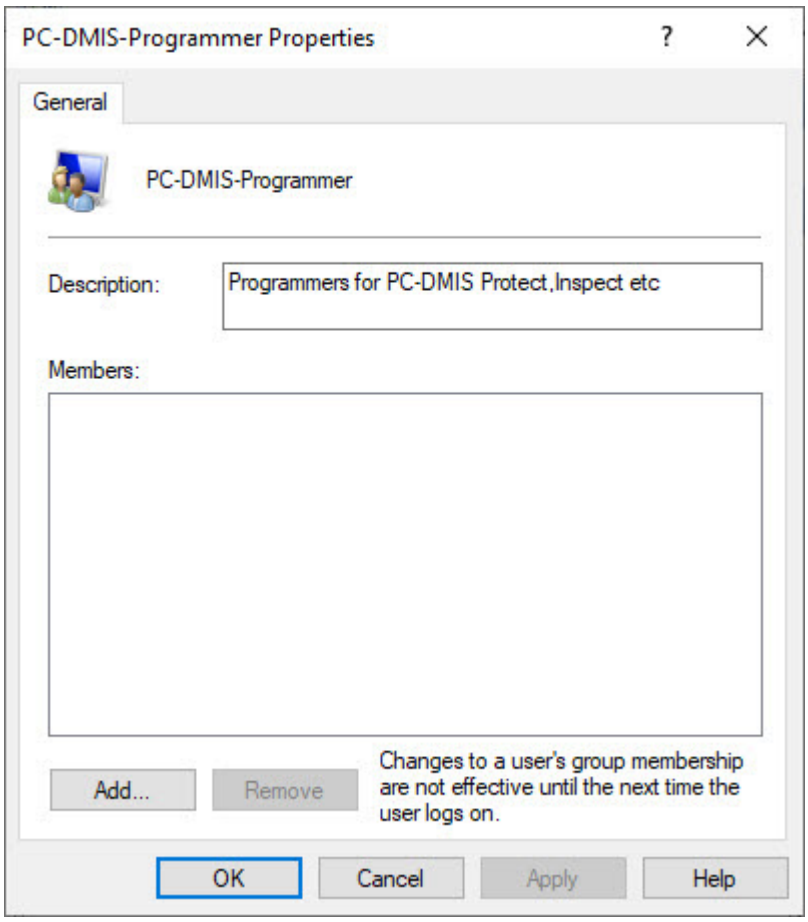
Prerequisites: Administrator access to the computer.

1. Install the Protect Viewer installer (included with the PC-DMIS installer). This creates the **PC-DMIS-User** and **PC-DMIS-Programmer** groups in the **Local Users and Groups** folder on your computer.
2. Open the **Edit local users and groups** dialog box. You can do this in two ways:
   - Type **local** in the search box on the taskbar and select **Edit local users and groups** from the results.
   - Press Windows + R key combination, type **lusrmgr.msc** in the **Run** dialog box, and click **OK**.

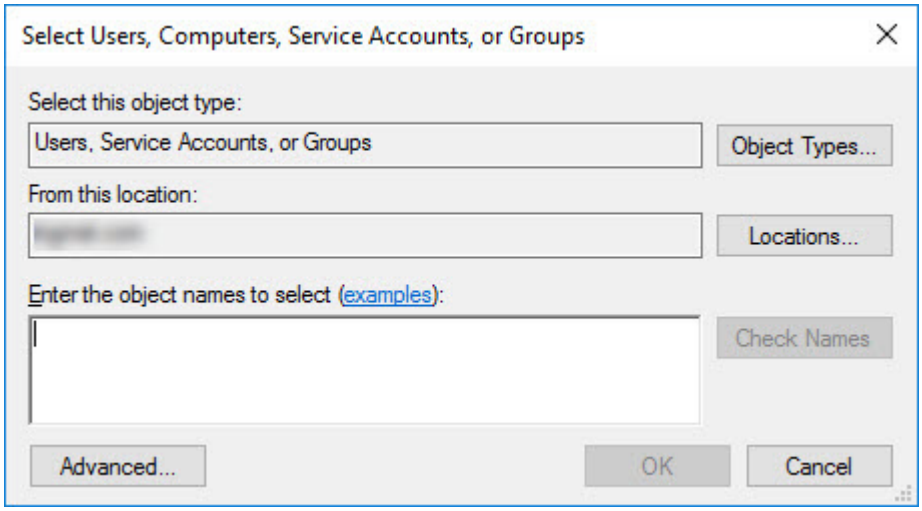
3. From the left pane, click **Groups** to display the list of groups on your computer, including the **PC-DMIS-User** and **PC-DMIS-Programmer** groups.



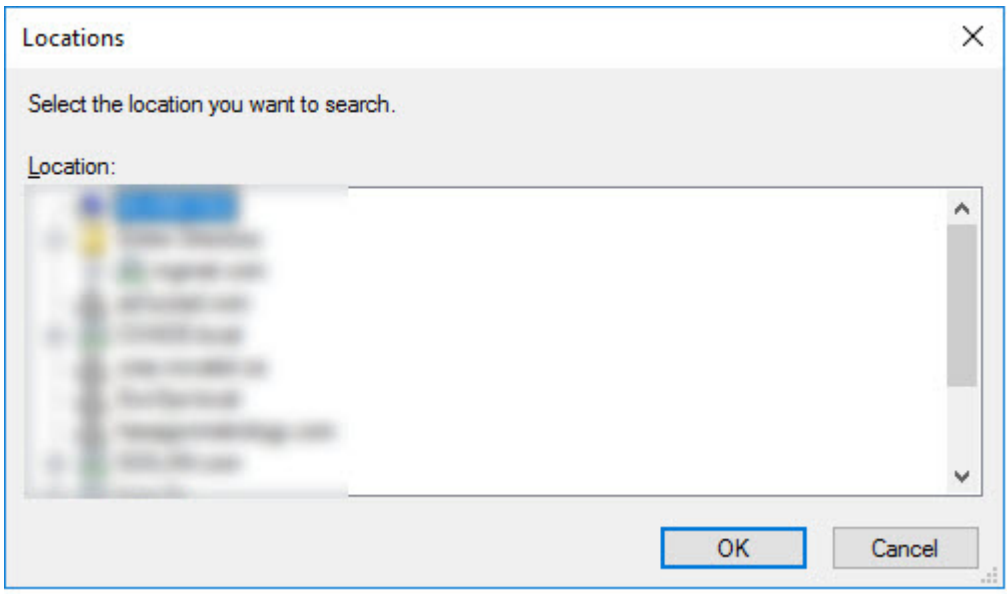
4. Double-click **PC-DMIS-Programmer** to open the **PC-DMIS-Programmer Properties** dialog box.



5. Click **Add** to open the **Select Users, Computers, Service Accounts, or Groups** dialog box.

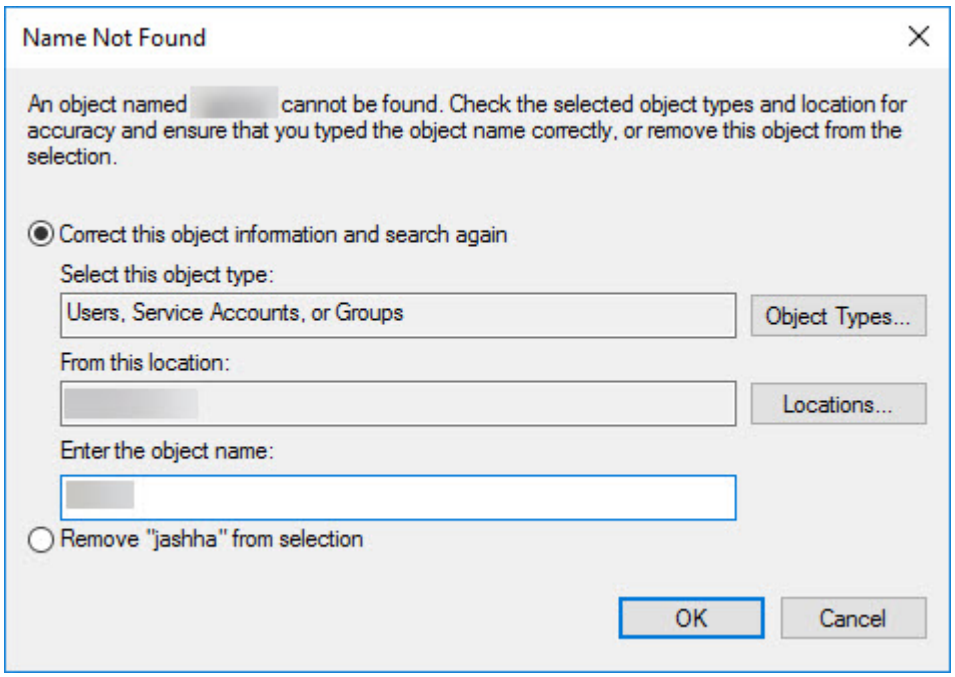


6. Click **Locations** to open the **Locations** dialog box. You can connect the account as a network user or a local user.



- To connect the account as a network user, select the network location and click **OK**.
- To connect the account as a local user, select the computer name and click **OK.**

7. In the **Enter the object names to select** box of the **Select Users, Computers, Service Accounts, or Groups** dialog box, type your user name and click **Check Names**. If the user account does not appear and your computer is part of a company network, verify your network connection. If you are outside of the company network, connect through a VPN.
   - If you are connected to a network, the box displays an entry with the network's name.
   - If you are not connected to a network, the **Name Not Found** screen appears.



> You can also add a local group or network group to the PC-DMIS-Programmer group. This adds all the members in that local or network group to the PC-DMIS-Programmer group.

8. Click **OK** to close the **Select Users, Computers, Service Accounts, or Groups** dialog box. The **Members** area in the **PC-DMIS-Programmer Properties** dialog box displays the user name or group that you added in step 6.
9. Click **OK** to close the **PC-DMIS-Programmer Properties** dialog box.
10. To apply the changes, you must sign out of Windows and sign back in.

You have successfully added the user or group to the PC-DMIS-Programmer group. You can follow the same steps to add a user or group to the PC-DMIS-User group.

> Whenever you make changes to the PC-DMIS-User or PC-DMIS-Programmer group, you must sign out of Windows and sign back in again to apply those changes. If you add a network group to the PC-DMIS User or PC-DMIS Programmer group, connect to the network to apply subsequent modifications.

Adding Azure AD (Microsoft Entra ID) Users to PC-DMIS-Programmer Group Offline

To add Azure AD (Entra ID) users to local groups without an Active Directory connection or VPN connection, open Windows PowerShell in administration mode and type this command: `net localgroup "PC-DMIS-Programmer" /add "AzureAD\user@domain.com"` .

One Level Authentication

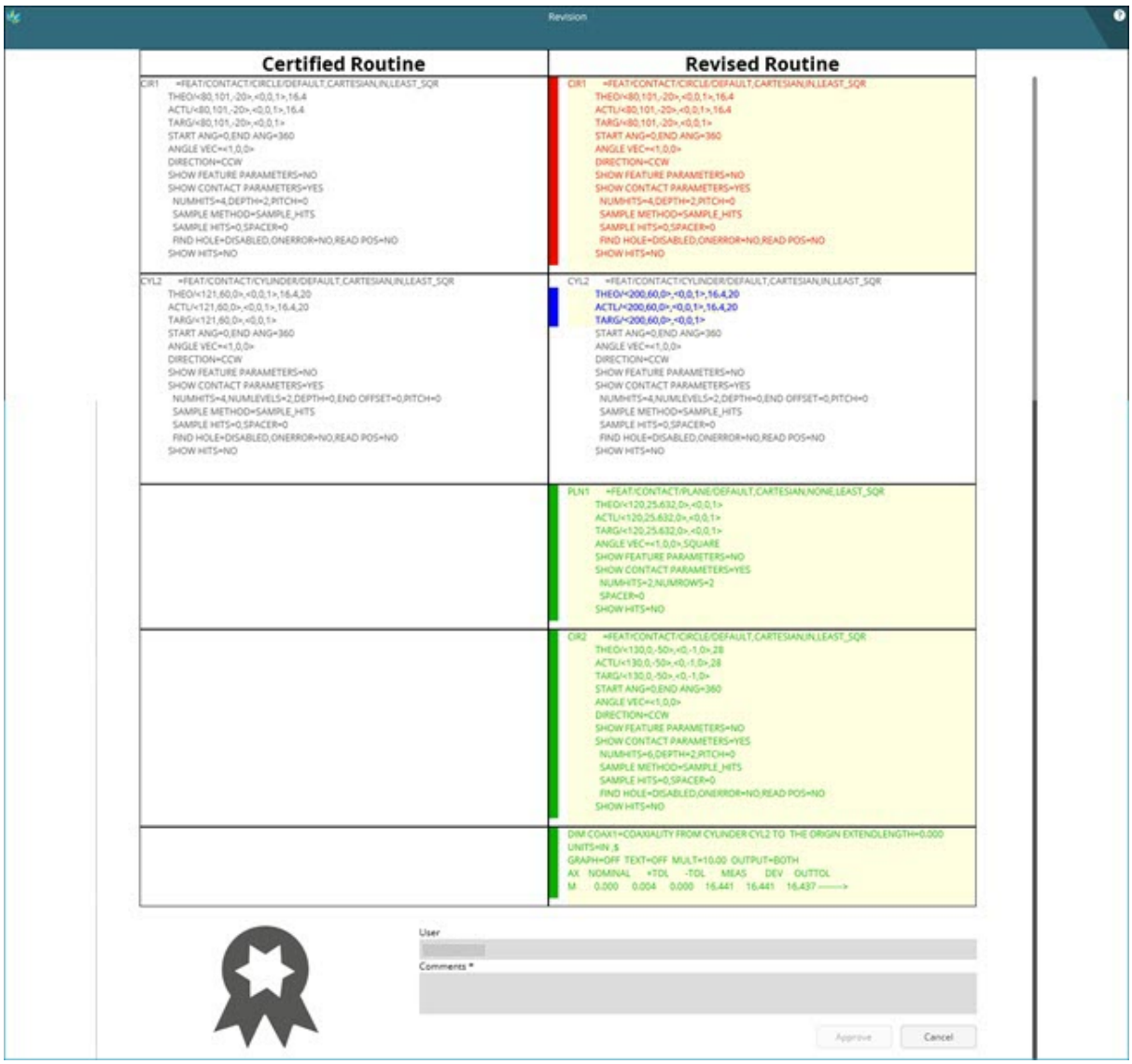
This procedure applies to members of the **PC-DMIS-Programmer** group.

To certify, edit, view, or recertify a measurement routine, follow these steps:

1. Create a measurement routine.
2. Add features, dimensions, and other necessary elements to the measurement routine.
3. From the **File Operations** toolbar, click the **Certify** button ( ) to open the **Certify Routine** dialog box.
4. In the **Comments** box, type your comments and click **Certify** to certify the measurement routine. PC-DMIS immediately opens the measurement routine in Protected Mode.

   > PC-DMIS displays a lock icon at the top of the Edit window, indicating that the measurement routine cannot be edited.

5. To edit the certified measurement routine, from the **File Operations** toolbar, click the **Edit Certified Routine** button ( ). PC-DMIS opens the measurement routine in Edit Mode.
6. Edit your measurement routine.
7. From the **File Operations** toolbar, click the **Recertify** ( ) button to open the **Revision** dialog box. This dialog box displays the **Certified Routine** and **Revised Routine** columns to review the changes made after the certification of the measurement routine.



Revision Dialog Box

8. After you review the changes, click **Approve** to recertify the measurement routine. PC-DMIS immediately adds the changes to the certified measurement routine and opens the recertified measurement routine in Protected Mode.

> After you submit a measurement routine for certification, PC-DMIS generates and assigns a .protect file to it. This file tracks changes and holds the current status of the measurement routine's certification process. If you move or delete the .protect file, you will lose all tracked changes, and no member of the PC-DMIS-Programmer group will be able to recertify the measurement routine.

Two Level Authentication

To certify a measurement routine, follow these steps:

1. As a member of the PC-DMIS-User group:
    a. Create a measurement routine.

b. Add features, dimensions, and other necessary elements to the measurement routine.

c. From the **File Operations** toolbar, click the **Submit** button ( ) to open the **Submit Routine** dialog box. In the **Comments** box, type your comments and click **Submit** to submit the measurement routine for certification.

> To cancel the submission, from the **File Operations** toolbar, click the **Cancel Submission** button ( ) to open the **Cancel Submission of Routine** dialog box. In the **Comments** box, type your comments and click **Cancel Submission**.

d. After you submit the measurement routine for certification, select the **File | Exit** menu option.

2. As a member of the PC-DMIS-Programmer group:

a. Open the submitted measurement routine.

b. Select the **File | Operations | Certify** menu option or from the **File Operations** toolbar, click the **Certify** button ( ) to open the **Certify Routine** dialog box.

c. In the **Comments** box, type your comments and click **Certify** to certify the measurement routine. PC-DMIS immediately opens the measurement routine in Protected Mode.

> After certification, PC-DMIS always opens the measurement routine in Protected Mode for all users.

To recertify a measurement routine, follow these steps:

1. As a member of the PC-DMIS-User group:

a. Open the certified measurement routine. PC-DMIS opens the measurement routine in Protected Mode.

b. From the **File Operations** toolbar, click the **Edit Certified Routine** button ( ) to open the measurement routine in Edit Mode.

If you are a member of the PC-DMIS-Programmer group and you open the same measurement routine, PC-DMIS displays a message stating that you are not the creator of the measurement routine and that the measurement routine has not been submitted to you for certification.

c. Edit the measurement routine.

d. From the **File Operations** toolbar, click the **Submit** button ( ) to open the **Submit Revision** dialog box. In the **Comments** box, type your comments and click **Submit Revision** to submit the measurement routine for recertification.

- To cancel the submission, from the **File Operations** toolbar, click the **Cancel Submission** button ( ) to open the **Cancel Revision** dialog box. In the **Comments** box, type your comments and click **Cancel Revision**.
- To view the certified measurement routine, from the **File Operations** toolbar, click the **View Certified** button ( ) to open the certified measurement routine in Protected Mode.

e. After you submit the measurement routine for recertification, click the **File | Exit** menu option.

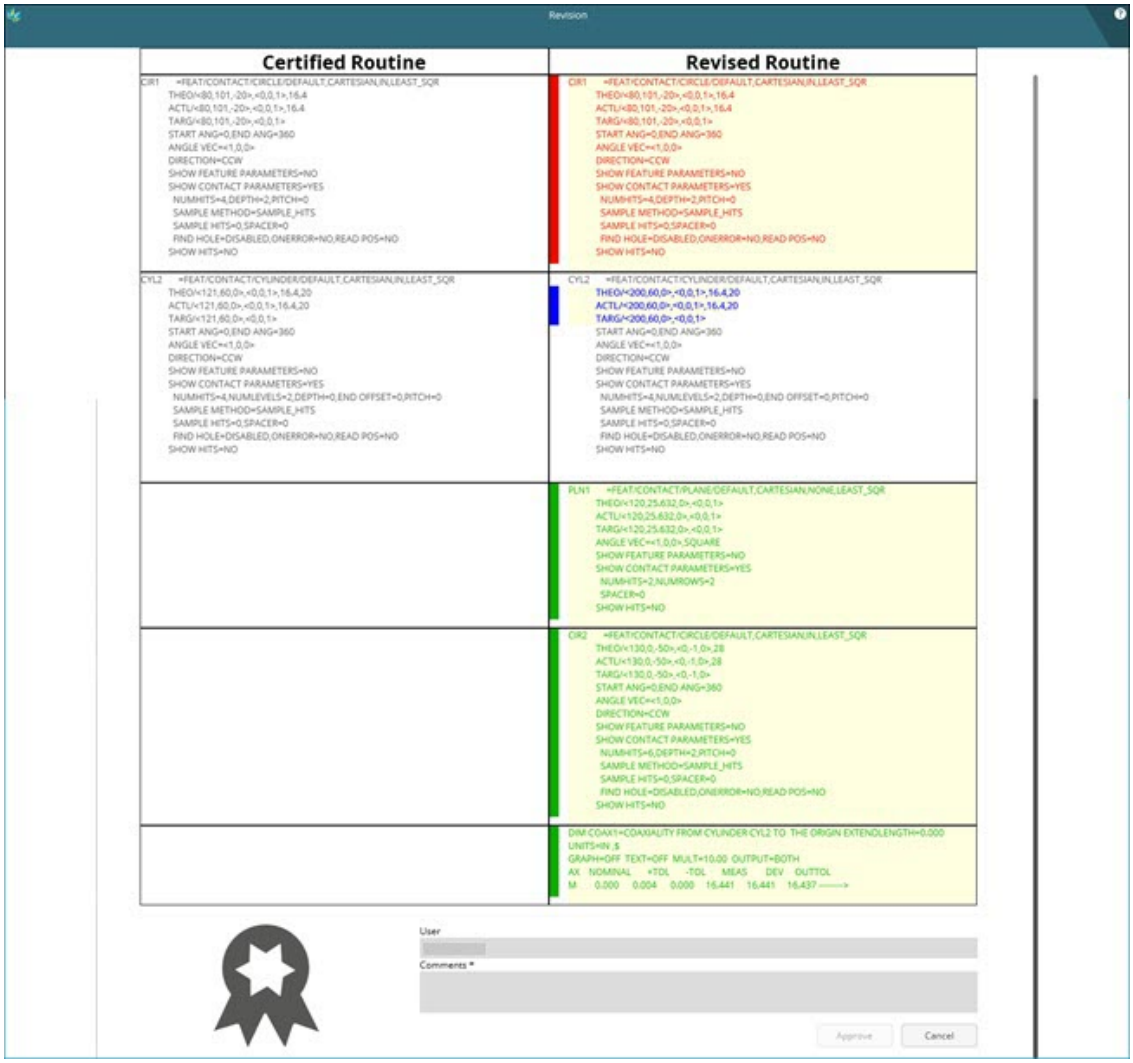
2. As a member of the PC-DMIS-Programmer group:

a. Open the measurement routine. PC-DMIS opens the measurement routine in Protected Mode.

b. From the **File Operations** toolbar, click the **Edit Certified Routine** button ( ) to open the measurement routine again in Protected Mode and to enable the **View Certified**, **Recertify**, **Reject**, and **Delete** options in the **File Operations** toolbar.

c. From the **File Operations** toolbar, click the **Recertify** button ( ) to open the **Revision** dialog box. The dialog box displays the **Certified Routine** and **Revised Routine** columns to review the changes made after the certification of the measurement routine.

- To reject the changes, from the **File Operations** toolbar, click the **Reject** button ( ).
- To delete all the changes after the last certification, from the **File Operations** toolbar, click the **Delete** button ( ).



Revision Dialog Box

d. After you review the changes, type your comments in the **Comments** box and click **Approve** to recertify the measurement routine. PC-DMIS immediately opens the measurement routine in Protected Mode.

After you submit a measurement routine for certification, PC-DMIS generates and assigns a .protect file to it. This file tracks changes and holds the current status of the measurement routine's certification process. If you move or delete the .protect file, you will lose all tracked changes, and no member of the PC-DMIS-Programmer group will be able to recertify the measurement routine.

Edit Certified Measurement Routines

Local Environment

You can edit certified measurement routines only if you are the creator of the measurement routine or a member of the PC-DMIS-User group.

To edit a certified measurement routine, follow these steps:

1. If you are a member of the PC-DMIS-User group and the creator of a certified measurement routine, PC-DMIS opens the certified measurement routine in Protected Mode.
2. From the **File Operations** toolbar, click the **Edit Certified Routine** button ( ) to open the measurement routine in Edit Mode.
3. Make the necessary edits to your measurement routine.

If you are a member of the PC-DMIS-Programmer group and you open the same measurement routine, PC-DMIS displays a message stating that you are not the creator of the measurement routine and that it has not been submitted to you for certification. However, if a member of the PC-DMIS-User group submitted the measurement routine for certification, clicking the **Edit Certified Routine** button ( ) opens the measurement routine in Protected Mode and enables the **Recertify**, **Reject**, and **Delete** buttons in the **File Operations** toolbar.

For members who are not part of either the PC-DMIS Programmer or PC-DMIS User group, PC-DMIS always opens the certified measurement routine in Protected Mode. In Protected Mode, you can only execute measurement routines. You cannot disable the mode. For information on Protected Mode, see the "Using Protected Mode to Protect Measurement Routines from Edits" topic in the PC-DMIS Core documentation.

Teamcenter Integrated Environment

> You can edit certified measurement routines only if you are a member of the PC-DMIS-Programmer group.

To edit a certified measurement routine, follow these steps:

1. Open the measurement routine. For information on how to open a measurement routine from Teamcenter, see the "Step 2 - Open a Measurement Routine" topic in PC-DMIS DCI documentation.
2. From the **File Operations** toolbar, click the **Edit Certified Routine** button ( ) to open the measurement routine in Edit mode.
3. Make the necessary edits to your measurement routine and do one of the following:
   - **Save Changes** – Click **File | Save** to save the changes. The measurement routine remains uncertified. To view or continue working with the saved changes, you must open the measurement routine in Edit mode.
   - **Re-certify the Measurement Routine**
     A. Click the **Recertify** button ( ) to open the **Revision** dialog box. Review the changes by comparing the **Certified Routine** and **Revised Routine** columns.
     B. Enter comments, if required, and click **Approve** to open the **Teamcenter Save As** dialog box.

        For information about the options in the **Teamcenter Save As** dialog box, see the "Step 4 - Save a Measurement Routine" topic in the PC-DMIS DCI documentation.
     C. Select the required fields and click **OK**.

Procedure to Certify and View the .protect File

The following procedure certifies a measurement routine and generates a .protect file. You can use the Protect Viewer application to view this file. For information about the Protect Viewer application, visit the PC-DMIS Help library page. Select your specific version of PC-DMIS help and then navigate to the Protect Viewer documentation.

Prerequisites:

- To certify a measurement routine, you must be a member of the PC-DMIS-Programmer group and have the Protect Viewer option in your license enabled.

- To view the .PRG.protect file, you must have the Protect Viewer application installed on your computer.

To certify and view the .PRG.protect file, follow these steps:

1. Create a measurement routine.
2. Select the **File | Operations | Certify** menu option or from the **File Operations** toolbar (**View | Toolbars | File Operations**), click the **Certify** button ( ) to open the **Certify Routine** dialog box.
3. Type your comments in the **Comments** box and click **Certify**.
   - **Local Environment** - PC-DMIS certifies and saves the measurement routine and generate a .PRG.protect file in the same folder as the measurement routine. The file name is <measurement routine file name>.PRG.protect, where <measurement routine file name> is the name of the measurement routine file. The default folder location is: "C:\Users\Public\Documents\Hexagon\PC-DMIS\2026.2".
   - **Teamcenter Integrated Environment** - PC-DMIS opens the **Teamcenter Save As** dialog box. Select the required options and click **OK**. PC-DMIS certifies and saves the measurement routine and generates the .PRG.protect file in the Teamcenter database.

     For information about the options in the **Teamcenter Save As** dialog box, see the "Step 4 - Save a Measurement Routine" topic in the PC-DMIS DCI documentation.

   Suppose the measurement routine file name is ABC.PRG, then the .protect file name is ABC.PRG.protect.
4. Navigate to the folder that contains the .protect file, double-click the file, or right-click and select **Open** to open the file in the Protect Viewer application.

Adding Watermarks to Certified Measurement Routine Reports

You can add a watermark to generated reports for certified measurement routines. The UseWaterMarkText Settings Editor entry controls whether the watermark appears, and WaterMarkText Settings Editor entry specifies the watermark text.

Changing the Language

The **File | Language** menu displays the languages that are supported by the product. A check mark indicates the current language. To switch to a new language, choose **File | Language** and select the desired language. PC-DMIS automatically saves your work, closes, and then restarts. Once restarted, PC-DMIS runs in the newly selected language.

Language Help Content

PC-DMIS automatically installs all the user interface resources for the supported languages during the main installation process. However, to show non-English offline help file content, you need to install that separately. Online help content is available over the Internet.

The offline language help files can be installed as needed by navigating to the language pack folder and running the appropriate language executable file.

To install a language help file:

1. Navigate to the language folder of the main PC-DMIS installer download. (<Product_Installer_Folder>\lang) .
2. Run the desired language pack executable to install the language help files.

This installs a **helpcenter** folder in the language directory under the PC-DMIS installation.

Connecting to Teamcenter

You can use the **File | Teamcenter** menu option to enable or disable a connection to the Teamcenter software. Teamcenter is a PLM (Product Lifecycle Management) database package developed by Siemens.

For the **File | Teamcenter** menu to appear, you must enable the Teamcenter option in your PC-DMIS license.

You can connect to Teamcenter by using either Service-Oriented Architecture (SOA) or the Shop Floor Connect (SFC) add-on. For information on SOA and SFC, see the "Teamcenter Connection Methods (SOA and SFC Connections)" topic in the PC-DMIS DCI documentation.

The **Teamcenter** menu includes these options:

- **On** - Select this option to establish a connection with Teamcenter. Once you select it, PC-DMIS displays the **Teamcenter Login** dialog box, where you can type your Teamcenter username and password. If the connection is successful, PC-DMIS displays a "Successfully connected to Teamcenter" message.

  A successful connection changes the standard **Save As** and **Open** dialog boxes to work with Teamcenter. If the connection fails, PC-DMIS defaults to **Off** and uses the standard **Save As** and **Open** dialog boxes.

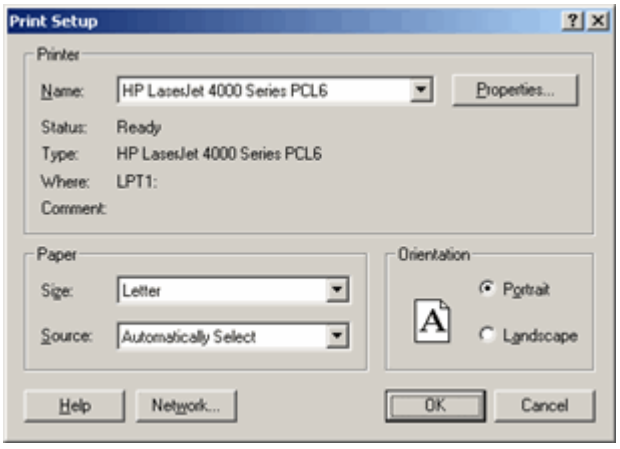
- **Off** - Select this option to disconnect PC-DMIS from Teamcenter. This reverts PC-DMIS to its standard **Save As** and **Open** dialog boxes.

- **Setup** - Select this option to open the **Teamcenter Setup** dialog box. For information on the **Teamcenter Setup** dialog box, see the "Teamcenter Setup Options" topic in the PC-DMIS DCI documentation.

For information on working with Teamcenter, see the "Using Teamcenter" topic in the PC-DMIS DCI documentation.

Setting Printer Options

The **Printer Setup** button and the **File Printing | Printer Setup** menu item access the standard Windows **Print Setup** dialog box. This dialog box lets you switch between available printers, choose paper size, page orientation, and access other printer properties. PC-DMIS then uses these options when printing the Edit window, the Inspection Report, or the Graphic Display window.



A Sample Print Setup Dialog Box

For information on this dialog box and the options available, consult the help file that came with your Windows operating system. For any advanced printer-specific properties, you may need to consult documentation specific to your printer.

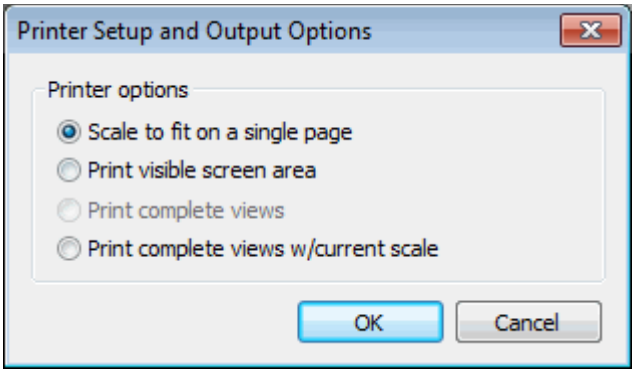
Printing the Graphic Display Window

PC-DMIS lets you send the current contents of the Graphic Display window to your printer. Select **File | Printing | Graphic Window Print** to display a Windows **Print Setup** dialog box. Click **OK** to send the report to the printer shown in the dialog box.

Before printing, you can set the output options, and preview your print job.

Setting Output and Printer Options for the Graphic Display Window

Select **File | Printing | Graphic Window Print Setup** to display the **Printer Setup and Output Options** dialog box.



Printer Setup and Output Options Dialog Box

Use this dialog box to set up your printer and various display options. The options available in the **Printer options** area enable you to select the type of graphical view to print. These include:

Scale to fit on a single page - This option scales any graphical image to fit on a single page.

Print visible screen area - This option prints only the currently visible screen area. If you're zoomed in on a feature, it only prints what's on the screen and not the entire part.

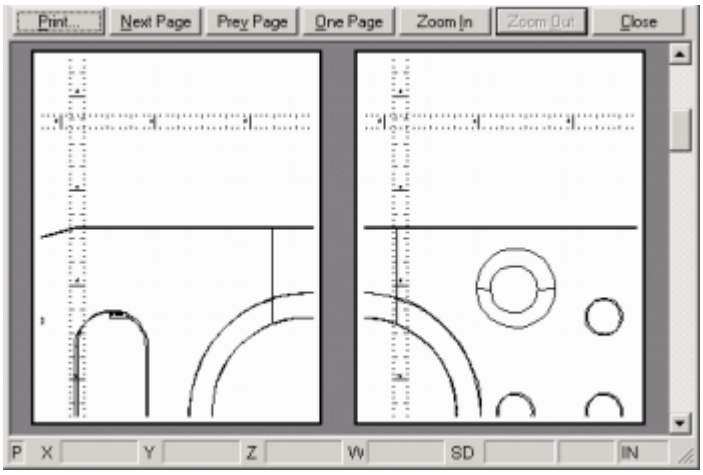
Print complete views - This option prints each view you have defined using the **Layout** area of the **View Setup** dialog box on its own page. For example, if you're showing the Z+ view and the Y- view of a part in your Graphic Display window, PC-DMIS prints two separate pages: one with the Z+ view and one with the Y- view.

Print complete views w/current scale - This option acts like the **Print complete views** option, except that it prints at the current scale. If you have zoomed in on the image, PC-DMIS still prints the entire view, but it splits the image among various pages.

To save any changes, click the **OK** button.

Previewing the Print Job

Select **File | Printing | Graphic Window Print Preview** to preview what is printed from the Graphic Display window. A Print Preview window appears.



Print Preview window showing the contents of the Graphic Display window.

The buttons at the top of the window perform these functions:

The **Print** button opens up a standard **Print Setup** dialog box so you can send the print job to the printer.

The **Next Page** and **Prev Page** buttons cycle through a print job that has multiple pages.

The **One Page** or **Two Pages** toggle button determines how many pages are displayed at one time in the Print Preview window.

The **Zoom In** and **Zoom Out** buttons allow you to take a quick closer look at a page in the Print Preview window. It doesn't affect the display of what's sent to the printer.

The **Close** button closes the Print Preview window.

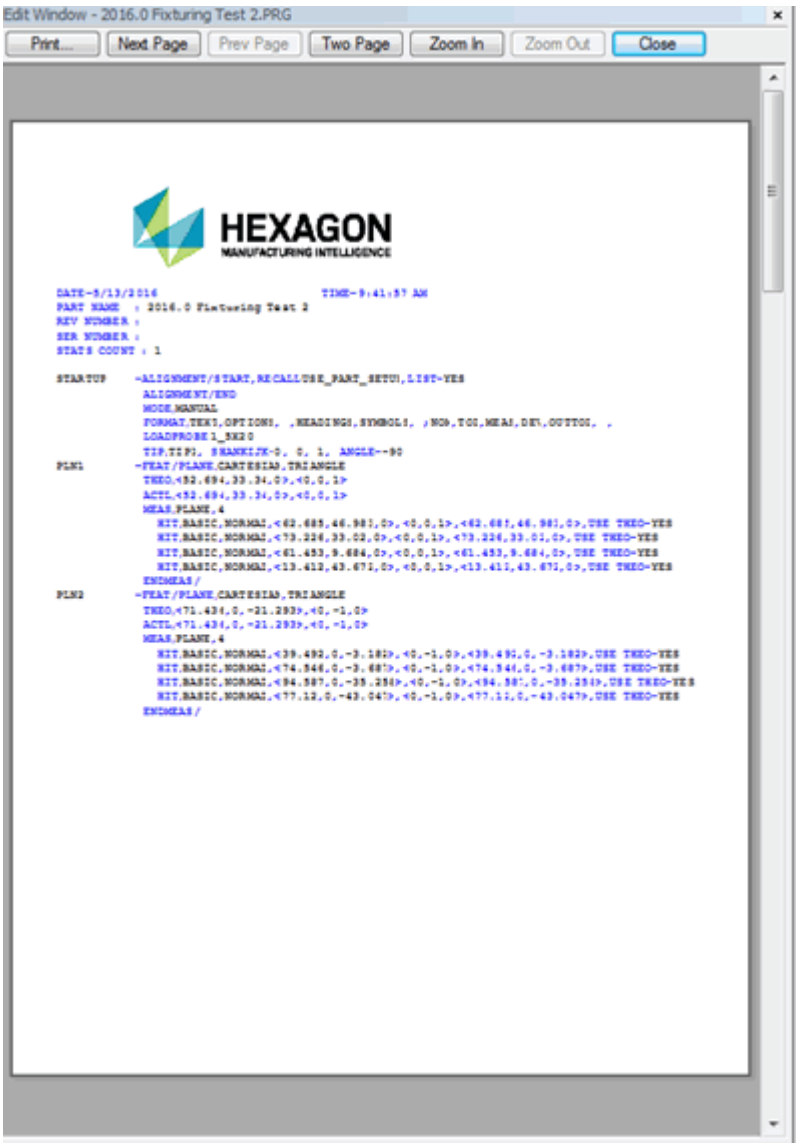
Printing the Edit Window

You can send the contents of the Edit window to your printer in these ways:

- Select **File | Printing | Edit Window Print**.
- Click the **Print** icon from the **Edit Window** toolbar.
- Press F4.

Previewing the Print Job

To preview what the printout will look like, select **File | Printing | Edit Window Print Preview**. A Print Preview window appears:



Print Preview window showing the contents of the Edit window.

The buttons at the top of the window perform these functions:

- The **Print** button opens a standard **Print Setup** dialog box that enables you to send the print job to the printer.
- The **Next Page** and **Prev Page** buttons cycle through a print job that has multiple pages.
- The **One Page** or **Two Pages** toggle button determines how many pages appear at one time in the Print Preview window.
- The **Zoom In** and **Zoom Out** buttons allow you to take a quick closer look at a page in the Print Preview window. It doesn't affect the display of what's sent to the printer.
- The **Close** button closes the Print Preview window.

When you print the Edit window's contents, you should use a TrueType font (such as Courier New). Otherwise, characters or lines may not line up as expected. To change the fonts used in the Edit window and in your reports, see "Customizing User Interface Fonts" in the "Navigating the User Interface" chapter.

Printout in Summary Mode

When you view the Edit window in Summary Mode, you get the printout exactly as you see in the Edit window. If a group is collapsed in the Edit window, the printout shows the group as collapsed.

Setting Output Options

You can set the output options for Edit window print tasks from the same **Output Configuration** dialog box that you use for the report's output. For information on the options in the **Output Configuration** dialog box, see "Setting Output and Printer Options for the Report Window".

To change printers, set page orientation, or modify other printer options, select **File | Printing | Printer Setup**. For more information, see "Setting Printer Options".

Printing the Report Window

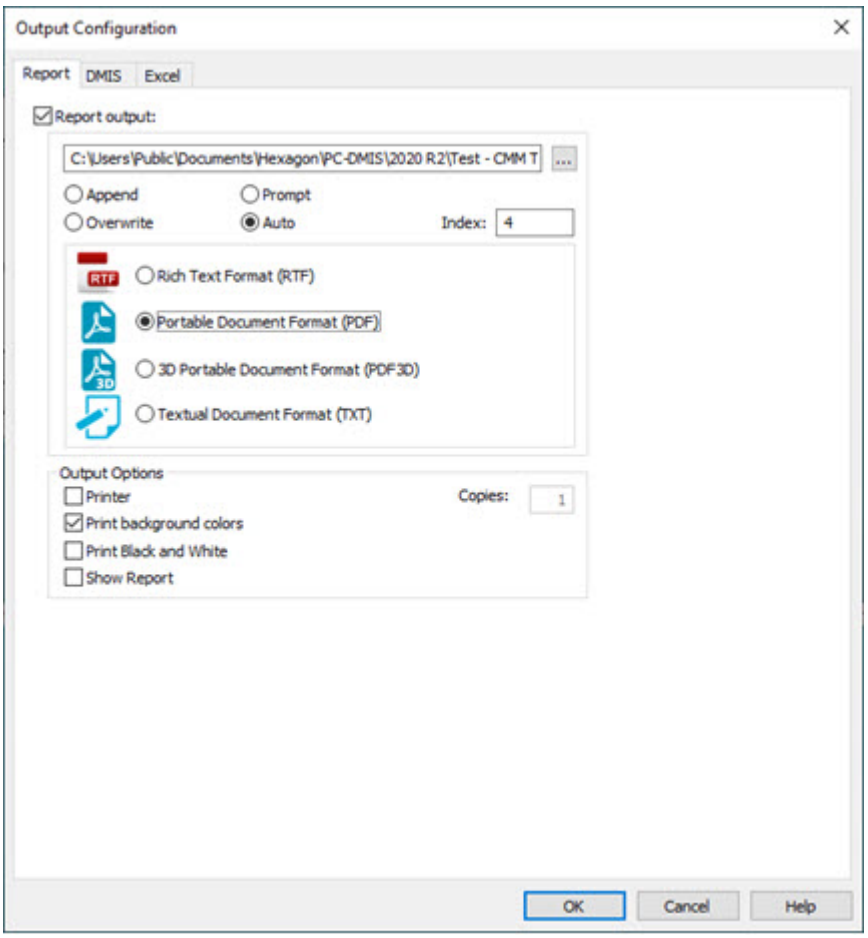
PC-DMIS lets you send the inspection report in the Report window to a variety of output devices or to a file. PC-DMIS supports a wide variety of printers and plotters. Please contact your PC-DMIS sales representative if you have any questions concerning hardware compatibility.

To print inspection reports, select the **File | Printing | Report Window Print Setup** menu option. Before printing, you can set the output options for your print job by choosing **File | Printing | Report Window Print Setup**.

When printing a text-only report, you should use a True Type font (such as Courier New). Otherwise, characters or lines may not line up as expected. If you need to change the font used in a text-only report, modify the **Font** property on the TextReportObject used in your report template.

Setting Output and Printer Options for the Report Window

Selecting the **File | Printing | Report Window Print Setup** menu option displays the **Output Configuration** dialog box.



Output Configuration Dialog Box

You can use this dialog box to tell PC-DMIS where to send the inspection report. The dialog box contains the following tabs that you can use to send the report to a file or printer, output it as a DMIS file or a Microsoft Excel file, or send to any combination of these:

Report - Sends the report output to a file (.rtf, .pdf, or .txt) or printer. For information about this tab, see "Output to an RTF, PDF, TXT or the Default Printer". For information about using this tab to output to a file using expressions, see "Output to a File Using Expressions".

DMIS - Sends the report output to a DMIS output file (.dmo). For information about this tab, see "Output to a DMIS File".

Excel - Sends the report output to an Excel file (.xls, xlsx, or .csv). For information about this tab, see "Output to an Excel File".

For information on viewing a saved inspection report, see the "Viewing an Inspection Report" topic in the "Using Other Windows, Editors, and Tools" chapter.

Setting Printing Options for Marked Sets

PC-DMIS also displays an instance of the **Output Configuration** dialog box whenever a marked set is created. This defines printing options *specific only to the newly created marked set*. Alternately, if you would rather use already defined print settings used by your

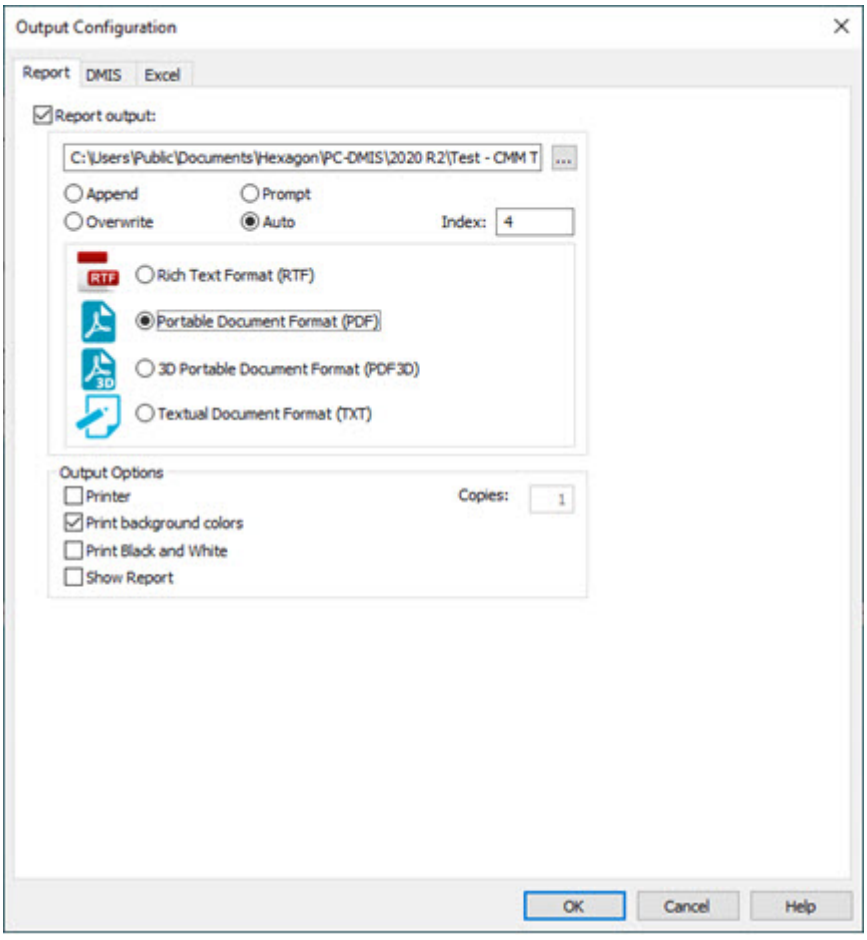
measurement routine at large, you can select the **Use Global Print Settings** check box. For information on using marked sets, see "Creating and Executing Marked Sets" in the "Editing a Measurement Routine" chapter.

To define printing options for existing marked sets, select the set in the Marked Sets window, and then select **File | Printing | Report Window Print Setup**.

Setting Image Ratio

The `MaxImageRatio` entry sets the maximum image ratio between the screen resolution and the printer resolution when printing the CAD drawing from the Report window. See the `MaxImageRatio` entry in the **Reporting** section of the PC-DMIS Settings Editor.

Output to an RTF, PDF, or TXT File or to the Default Printer



Output Configuration Dialog Box - Report Tab

In the **Output Configuration** dialog box (**File | Printing | Report Window Print Setup**), the **Report** tab defines whether to:

- Send the report's output to a printer
- Save the report as a separate file (.rtf, .pdf, or .txt)
- Do both of the above

The dialog box also provides options for the generated report.

Report Output

Report output - If selected, PC-DMIS sends the output to a file. It can be saved in either a text file (.txt), Rich Text Format (.rtf) or Portable Document Format (.pdf) file. This .pdf file can be a standard PDF file or a 3D PDF file.

The file name is initially generated and formatted to have the same name as the measurement routine plus an appended numerical index before the extension. Although PC-DMIS generates this initial file name, the name does not need to follow the default format and can be changed.

You can also type a variable name in the **Report output** box. As long as there's not a pathway already in the box, PC-DMIS interprets that variable's value for the filename.

Append - PC-DMIS adds the current data from the inspection report to the selected file. Note that the complete path *must* be specified; otherwise, PC-DMIS assumes the same directory as the measurement routine. Also, if the file does not exist, it is created when you generate the report.

Append Limitations:

For an RTF file, due to a limitation with how the RTF driver works with the PC-DMIS template approach to reporting, when *appending* to an RTF file, PC-DMIS only sends the contents of the **TextReporting** object, regardless of any other objects that may exist in your current report template.

Additionally, if you choose to append to an RTF file, it must be to a file that you have *always* used with the **Append** option. You cannot append to an RTF file that you first used with **Overwrite** or **Auto**.

For a 3D PDF file, PC-DMIS does not support the append functionality with that file format.

Overwrite - PC-DMIS overwrites the selected file with the current inspection report data. Note that the complete path must be specified; otherwise, PC-DMIS will assume the same directory as the measurement routine. Also, if the file does not exist, it is created when you generate the report.

Prompt - PC-DMIS displays the **Save As** dialog box through which you can choose the destination file for the report.

Auto - PC-DMIS generates the report file name automatically using the number in the **Index** box. The name of the generated file name will have the same name as the measurement routine appended by the numerical index and extension. Also, the generated file will be located in the same directory as the measurement routine. If a

file exists with the same name as the generated file name, the **Auto** option increments the index value until a unique file name is found.

After the report prints, PC-DMIS internally updates the value in **Index** to the next number. It also changes the file name in the **Print Options** dialog box so that it shows the newly-incremented file name.

Rich Text Format (RTF) - PC-DMIS generates the report using the Microsoft Rich Text Format (.rtf file) for interchanging documents. Images in RTF reports have a light border around them. By default, PC-DMIS generates RTF reports using an Amyuni RTF converter. This essentially orders information inside several text boxes inside the RTF file. This is needed for accurate positioning of report template elements inside the RTF report. To properly view the report, you should use Microsoft Word and set the document's view to **Print Layout**. If you find editing the report in this format difficult to work with, you may want to generate an old-style RTF report.

Generating an Old-Style RTF Report

PC-DMIS provides a way to generate an old-style RTF report, such as used in version 3.7 and earlier. To do this, follow this procedure:

1. Close down PC-DMIS.
2. Launch the PC-DMIS Settings Editor, and click **Connect**.
3. Inside the Settings Editor, expand the **USER_Printing** section, and find the `DoNotUseAmyuniRTF` entry.
4. Set the **Current Value** to **1**, click **Save Setting**, and click **OK**.

This setting only works if the RTF output is done using the **File | Printing | Report Window Print Setup** menu option. If you have inserted a PRINT/REPORT command (**Insert | Report Command | Print Command**) and you want that command to generate your RTF output, set the value to **2**.

5. Restart PC-DMIS.
6. Ensure that the Report window is set to print the report in a text-only format. To do this, right-click on the white-space at the end of the Report window and select the **Use Text Mode Dimension Reporting** check box. For more information, see "Changing the Report Window's Contents" in the "Reporting Measurement Results" chapter.

Changing Margin and Page Sizes for Old-Style RTF Reports

If the `DoNotUseAmyUniRTF` entry is set to 1 or 2, you can change the default margin and page sizes used for the RTF output by modifying these entries in the **Printing** section of the PC-DMIS Settings Editor:

`PcdmisRTFLeftMargin`

`PcdmisRTFRightMargin`

`PcdmisRTFTopMargin`

`PcdmisRTFBottomMargin`

`PcdmisRTFPaperHeight`

`PcdmisRTFPaperWidth`

Intended Uses of RTF Reports

As the RTF file type implies, an RTF report is primarily intended as a file format for interchange of text-based reports as shown below:

```
PART NAME  : Test Program
REV NUMBER : B5
SER NUMBER : 13579
STATS COUNT : 1

Active alignment changed to STARTUP

            START ANG=0,END ANG=0CYL1=CYLINDER MEASURED FROM 9 HITS
DIM LOC1= LOCATION OF CYLINDER CYL1  UNITS=MM
AX    NOMINAL       +TOL       -TOL       MEAS        DEV     OUTTOL
X      154.500      0.025      0.025    154.495     -0.005      0.000 ---#-----
Y       19.500      0.025      0.025     19.503      0.003      0.000 -----#---
Z      -35.000      0.025      0.025    -35.000      0.000      0.000 ----#----
```

Sample text-based report that uses the default .rtp report template.

RTF files containing graphical elements, such as those based off of the CADonly.rtp, TextOnly.rtp, and other similar templates, will take an extended amount of time to create, the files may become quite large, and the graphics may not look as good as with other file formats. For these reasons, the PDF format is recommended over the RTF format for graphical reports.

Portable Document Format (PDF) - PC-DMIS generates the report using the Adobe Portable Document Format (.pdf file) for electronic file sharing. Dimensions in PDF format do not have the default blue background color or display the dimension symbol.

If not already installed on your computer, you need to install the free Adobe® Acrobat® Reader™ in order to view .pdf files. You can download it from Adobe's website at: http://www.adobe.com/products/acrobat/readstep.html

3D Portable Document Format (PDF3D) - This also generates a .pdf file, but the main difference is that it also includes an embedded CAD object that you can manipulate within the .pdf file. For information and known limitations, see "Working with PDF 3D Files" in the "Reporting Measurement Results" chapter.

Textual Document (TXT) - PC-DMIS generates the report in a plain text format. You can view this type of report with any text editor. During report generation, PC-DMIS uses the default.rtp template, regardless of the actual template selected in the Report window. This textual report ignores any objects, such as images. For information on the default.rtp template, see "Viewing a Legacy Text Report" in the "Reporting Measurement Results" chapter. If you save your measurement routine to a version that doesn't support the .txt output, PC-DMIS uses the PDF output instead.

Use Global Print Settings - This check box becomes available if you first select a marked set from the Marked Sets window and then access this dialog box. The **Use Global Print Settings** check box determines whether or not PC-DMIS uses global output file parameters for the marked set. If you select this check box, PC-DMIS overrides the very specific default printing options used for marked sets with the measurement routine's global settings that give you greater control over the output. Select this check box to give you more control over the print-to-file naming schemes used with marked sets.

You can use the **Hyper Reports Inline** check box with the `REPORT/LEGACY` command. For more information, see "Embedding Reports or Report Templates in a Measurement Routine" in the "Reporting Measurement Results" chapter.

Output Options

Printer - Determines whether to send the report to the default printer.

Print background colors - Determines whether the report prints background colors.

By default, PC-DMIS marks this check box and prints the background colors. If you clear this check box, PC-DMIS does not print any background colors. If you want to clear or display the background color from a report already in the Report window, make sure you click the **Redraw the Report** button from the **Reporting** toolbar.

To change the default state of this check box, click the **Edit** menu, point to **Preferences**, and then click **Setup**. Then on the **General** tab, in the list of check boxes, mark or clear the **Print Background Colors** check box.

pc-dmis	PART NAME : V41Test			March 15, 2006		12:22	
	REV NUMBER :			SER NUMBER :		STATS COUNT : 1.0000	
0.0000	MM	LOC1 - CIR1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	154.5000	0.010	0.010	154.5000	0.000	0.000	
Y	80.5000	0.010	0.010	80.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC2 - CIR2					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	93.5000	0.010	0.010	93.5000	0.000	0.000	
Y	80.5000	0.010	0.010	80.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC3 - CIR3					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	93.5000	0.010	0.010	93.5000	0.000	0.000	
Y	19.5000	0.010	0.010	19.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC4 - CIR4					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	154.5000	0.010	0.010	154.5000	0.000	0.000	
Y	19.5000	0.010	0.010	19.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	

Sample report with background colors.

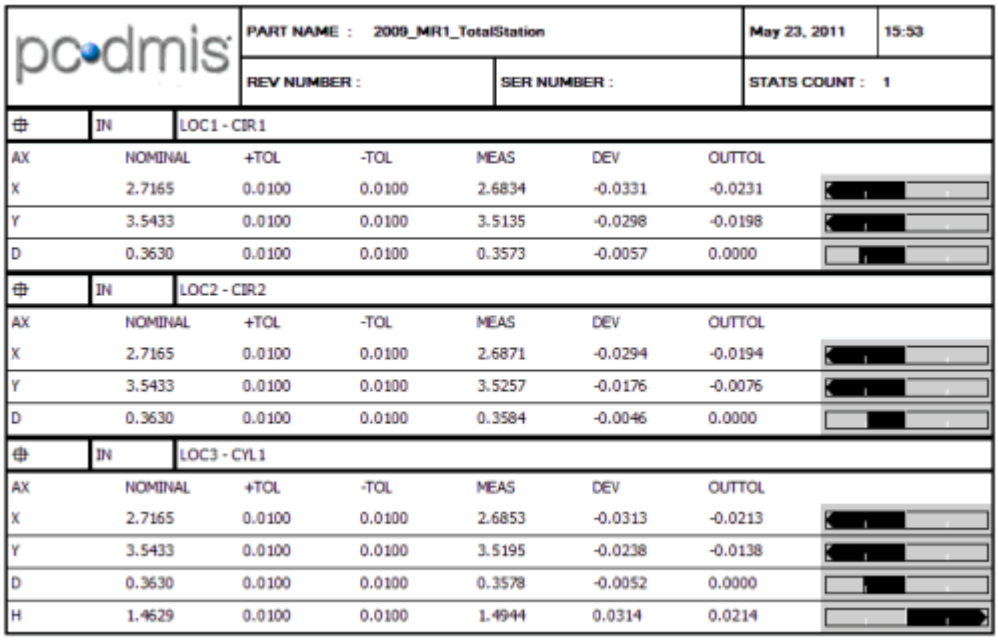
pc-dmis	PART NAME : V41Test			March 15, 2006		12:27	
	REV NUMBER :			SER NUMBER :		STATS COUNT : 1.0000	
0.0000	MM	LOC1 - CIR1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	154.5000	0.010	0.010	154.5000	0.000	0.000	
Y	80.5000	0.010	0.010	80.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC2 - CIR2					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	93.5000	0.010	0.010	93.5000	0.000	0.000	
Y	80.5000	0.010	0.010	80.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC3 - CIR3					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	93.5000	0.010	0.010	93.5000	0.000	0.000	
Y	19.5000	0.010	0.010	19.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	
0.0000	MM	LOC4 - CIR4					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	154.5000	0.010	0.010	154.5000	0.000	0.000	
Y	19.5000	0.010	0.010	19.5000	0.000	0.000	
PD	15.0000	0.010	0.010	15.0000	0.000	0.000	

Sample report without background colors.

In certain cases, you may continue to see some background colors in your report even if you've turned off the display of background colors here. This is due to a special background color setting that exists inside certain report template objects and is not controlled by this check box. By default, the **TextReportObject** and the **Report** object that exist in PC-DMIS report templates have a faint light-green background color in their color scheme. For the **TextReportObject**, this exists to help show the limits of the object inside the template. If this background color poses a problem, access the **Color** property for these objects inside the report template, and change these objects' background colors to white. For more information on how to do this, see "Changing the Report's Text Colors" in the "Reporting Measurement Results" chapter.

Print Black and White - Determines whether text, lines, and the tolerance zone graphs print in black and white or use the defined colors.

This does not make every report a completely black and white report. Some elements are not affected by this check box (for example, the Graphic Display window, images, and elements in some OCXs are not affected).



pc-dmis

PART NAME : 2009_MR1_TotalStation		May 23, 2011	15:53
REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

⊕	IN	LOC1 - CIR1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	2.7165	0.0100	0.0100	2.6834	-0.0331	-0.0231
Y	3.5433	0.0100	0.0100	3.5135	-0.0298	-0.0198
D	0.3630	0.0100	0.0100	0.3573	-0.0057	0.0000

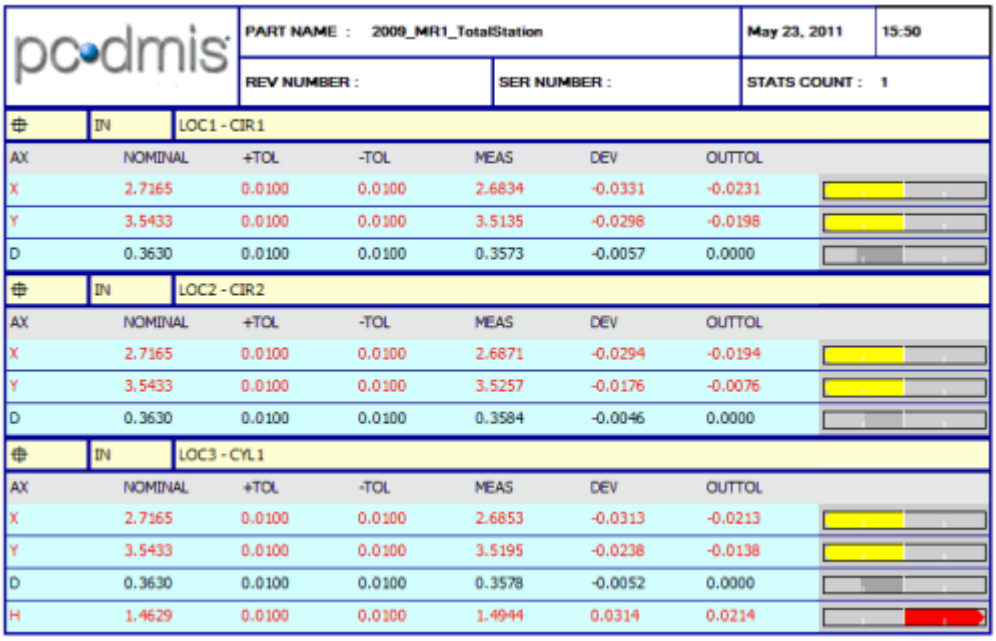
⊕	IN	LOC2 - CIR2

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	2.7165	0.0100	0.0100	2.6871	-0.0294	-0.0194
Y	3.5433	0.0100	0.0100	3.5257	-0.0176	-0.0076
D	0.3630	0.0100	0.0100	0.3584	-0.0046	0.0000

⊕	IN	LOC3 - CYL1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	2.7165	0.0100	0.0100	2.6853	-0.0313	-0.0213
Y	3.5433	0.0100	0.0100	3.5195	-0.0238	-0.0138
D	0.3630	0.0100	0.0100	0.3578	-0.0052	0.0000
H	1.4629	0.0100	0.0100	1.4944	0.0314	0.0214

Sample report in black and white.



pc-dmis	PART NAME : 2009_MR1_TotalStation		May 23, 2011	15:50
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

	IN	LOC1 - CIR1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	2.7165	0.0100	0.0100	2.6834	-0.0331	-0.0231	
Y	3.5433	0.0100	0.0100	3.5135	-0.0298	-0.0198	
D	0.3630	0.0100	0.0100	0.3573	-0.0057	0.0000	

	IN	LOC2 - CIR2					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	2.7165	0.0100	0.0100	2.6871	-0.0294	-0.0194	
Y	3.5433	0.0100	0.0100	3.5257	-0.0176	-0.0076	
D	0.3630	0.0100	0.0100	0.3584	-0.0046	0.0000	

	IN	LOC3 - CYL1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	2.7165	0.0100	0.0100	2.6853	-0.0313	-0.0213	
Y	3.5433	0.0100	0.0100	3.5195	-0.0238	-0.0138	
D	0.3630	0.0100	0.0100	0.3578	-0.0052	0.0000	
H	1.4629	0.0100	0.0100	1.4944	0.0314	0.0214	

Sample report in color.

Show Report - Determines whether to open the report that PC-DMIS sent to a .pdf or an .rtf file. If you select this option, the report opens in whatever application you have associated with the .rtf or .pdf file types after you generate the report.

Copies - Determines the number of copies to send to your printer. This option is only enabled if you mark the **Printer** option.

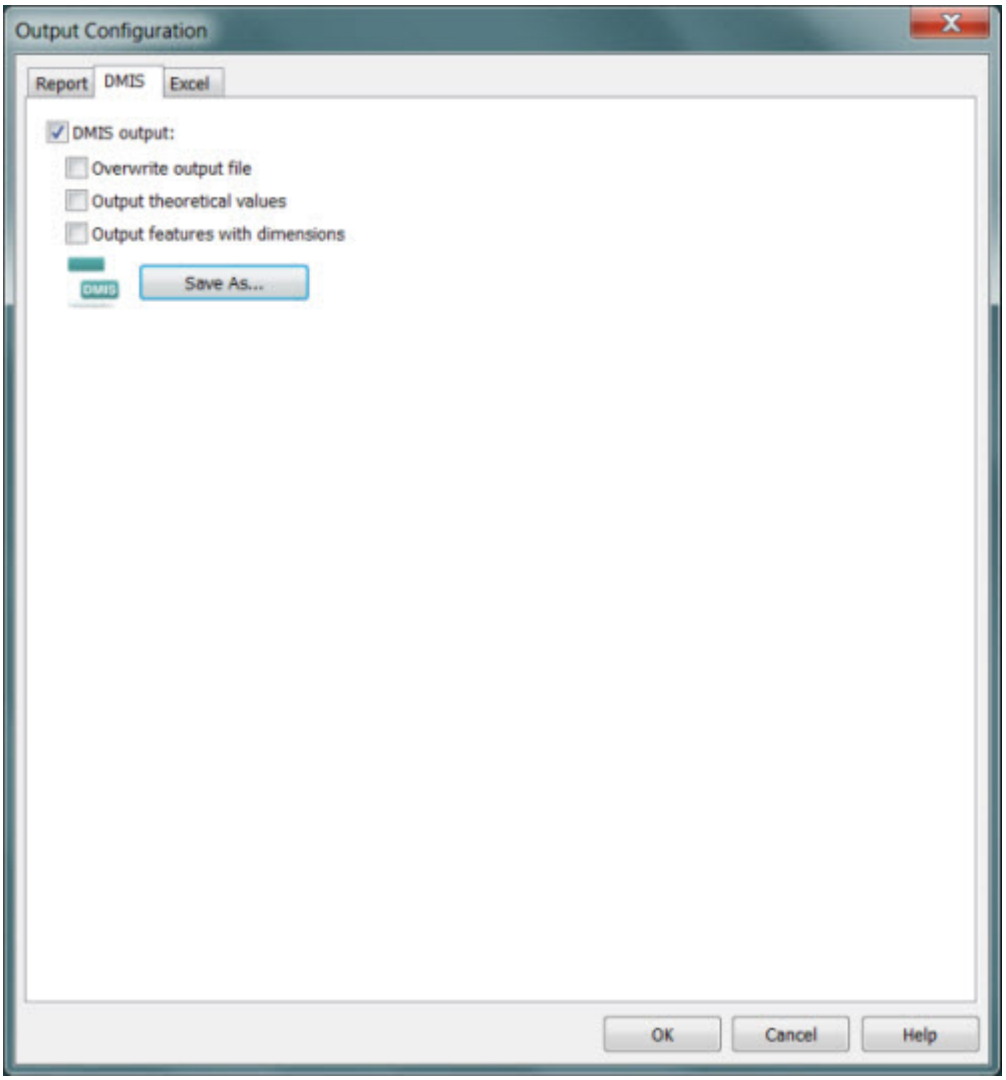
Controlling File Size by Changing the Print Resolution

If your report contains images of your part model, PC-DMIS, by default, prints these images at your printer's highest resolution capacity. This means reports showing part models and sent to a file may have a larger file size than expected.

You can control the resolution, and in turn control the report's file size, by changing the value of the `MaxPrintResolution` entry in the **USER_Printing** section of the PC-DMIS Settings Editor. For information on how to change entries, see "Modifying Setting Entries".

The `MaxPrintResolution` entry defines the dots per inch (dpi) that your printing device generates when printing CAD images. You can change this value to define the proper balance between file size and image quality to suit your needs. The default value is 1,000,000 dpi, thereby effectively using any printer's maximum print resolution capacity. If you define a value less than your printer's maximum print resolution, you will see a decrease in file size. For example, if your printer can print up to 500 dpi but you change the entry to a smaller value, like 70 dpi, you will notice a decrease in the size of the file.

Output to a DMIS File



Output Configuration Dialog Box - DMIS Tab

In the **Output Configuration** dialog box (**File | Printing | Report Window Print Setup**), on the **DMIS** tab, if you mark **DMIS output** check box and execute the measurement routine, PC-DMIS saves the inspection report's information as a DMIS output file with the same base file name as your measurement routine, with a .dmo file extension. PC-DMIS saves the file in the same directory where your measurement routine resides.

The **Overwrite output file**, **Output theoretical values**, and **Output features with dimensions** check boxes and **Save As** button become available. By default, PC-DMIS automatically generates the output file whenever you execute the measurement routine. It increments the number on the file name each time.

Overwrite output file

- If you select this check box, PC-DMIS overwrites the old output file with the new one. If you clear this check box, PC-DMIS automatically increments the file name specified in the **Save As** dialog box without overwriting any previous file.

For example, if you previously selected "test.dmo" as your output file, and **Overwrite output file** was not selected, the next time you execute the measurement routine, PC-DMIS saves a new file as "test1.dmo". New executions of the measurement routine increment the number in the saved file name in this manner.

- If you place **Overwrite output file** in its third state (a solid blue color), PC-DMIS opens the specified file in append mode. This follows the DMIS convention where you can enable, disable, and enable again the output to the same file. In PC-DMIS, this functions correctly only if you initially open the file with the **Overwrite output file** check box selected.

Output theoretical values

- If you select this check box, PC-DMIS outputs all theoretical values along with the measured values to the output DMIS file. If you clear this check box, the report does not include any theoretical values.

- If you place the **Output theoretical values** check box in its third state (a solid blue color), only those theoretical values that the original DMIS routine explicitly output are included in the report. This third state is useful when a DMIS import process creates the measurement routine, and you need to maintain the same DMIS output format.

Output features with dimensions

- If you select this check box, PC-DMIS keeps the measured features and associated tolerances together in the output file. PC-DMIS writes the measurement results immediately before their associated tolerance results for each dimension associated to the feature itself.

  If a feature is not associated with any tolerance, PC-DMIS does not generate any output.

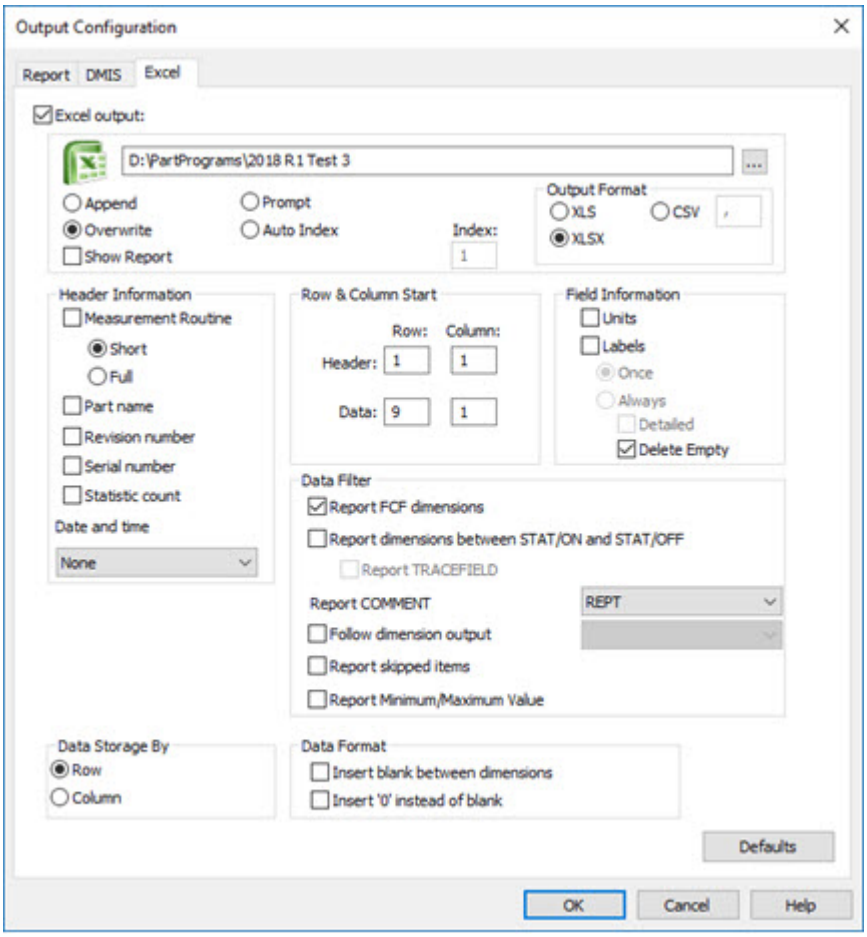
- If you clear this check box, the measurement results are written exactly when the feature is measured and not later when PC-DMIS executes the associated dimensions.

Save As

This button opens a **Save As** dialog box. This saves the inspection report to a file and directory of your choice in the DMIS output format (.dmo). PC-DMIS does not save the file once you click **Save** from the **Save As** dialog box. Instead, it creates the output file with the specified name the next time you execute the measurement routine.

The numeric portion of the file name should not exceed more than 10 digits. Otherwise, you run the risk of overwriting older output files.

Output to an Excel File



Output Configuration Dialog Box - Excel Tab

In the **Output Configuration** dialog box (**File | Printing | Report Window Print Setup**), the **Excel** tab outputs the report to an Excel file.

PC-DMIS follows the mode of the Report window:

- If you select the **View Report Mode** on the Report window's toolbar, the report output is generated from beginning to end.
- If you select the **View Last Execute Report Mode** on the Report window's toolbar, the report output is generated by following the execution list.

  For more details, see "Reporting Toolbar".

To output a report in an Excel format, do the following:

1. Click the **Excel** tab on the **Output Configuration** dialog box (**File | Printing | Report Window Print Setup**).
2. Select the **Excel output** check box to enable the tab's options.
3. Make your selections in the dialog box as needed and then click **OK**.

PC-DMIS saves the inspection report's information in the Excel file format that is selected in the **Output Format** area. The default is the same base file name as your measurement routine with a .xlsx file extension. The default location is in the same directory as your measurement routine. The available Excel file format options are:

- The latest Excel .xlsx format
- The older Excel .xls format
- The comma-separated value (.csv) format

Configuring the Excel Options

When the **Excel output** check box is selected, the following options become available:

Excel output - Displays the file name and the path where the report is saved. The file name defaults to the name of the measurement routine. You can enter a new file name and pathway, or click the "**...**" browse button to locate and select an existing file name and pathway.

You can also type a variable name in the **Excel output** box. As long as there's not a pathway already in the box, PC-DMIS interprets that variable's value for the filename.

Append - If the file name already exists in the targeted location, the current report information automatically appends to the end of the existing file.

Overwrite - If the file name already exists in the targeted location, the existing file is automatically overwritten with the current report content.

Prompt - If the file name already exists in the targeted location, you are prompted if you want to overwrite the existing file with the current report content or update the file name.

Auto Index - Used in conjunction with the **Index** box, this value updates the file name by appending an incremented numerical value by the index value. For example, if **XLS** is selected in the **Output Format** area, the initial file name is "FileName001.xls". The file names from then on are "FileName002.xls", "FileName003.xls", etc.

Index - A non-editable field that represents the increment value for the **Auto** option used when automatically naming the output file (see the **Auto** option above).

Show Report - The Excel report automatically opens for review once you generate it.

Output Format - Defines the output type. It contains these options:

XLS - Sends the output to the legacy Excel formatted file type.

XLSX - Sends the output to the latest Excel formatted file type.

CSV - Sends the output to a plain-text comma-separated value (CSV) file. You can choose a different delimiting character in the box next to **CSV**. This box is enabled when the CSV option is selected; it determines the separator, or delimiting character, to use when you export to a CSV file. The default is the comma character.

Header Information - Determines how the header data for the measurement routine is written to the output file.

Measurement Routine - Includes measurement routine names in the output file.

Short - The file name is a shortened format and shows only the file name (not the full path).

Full - The file name includes the full path.

Part name - Includes the part name in the output file.

Revision number - Includes the revision number in the output file.

Serial number - Includes the serial number in the output file.

Statistic count - Includes the number of transactions or runs in the output file.

Date and time - Specifies if the date and time appear in the output file and the format. It has these options:

None - Does not include the date and time in the output file.

System - Formats the date and time according to your computer's current format. This is the default.

24 Hour - Formats the date and time according to a 24-hour format.

Row & Column Start - Determines how the dimensional data appears in the output.

Header Row and **Column** - Define the starting location where you want the header information to appear. For example, values of 2 and 2 mean that it begins in the second row down and second column to the right.

Data Row and **Column** - Define the starting location where you want the dimensional data to appear. For example, values of 9 and 2 mean that it begins in the ninth row and second column.

Field Information - Determines how the information is formatted when you save it.

Units - Determines whether the measurement units used by each dimension are included in the export. For example, angle measurements show DEG for degrees, and size measurements show MM for millimeters or IN for inches.

Labels - Determines whether labels appear next to data fields in the output. If marked, the following options become enabled:

Once - Field labels appear only once after the heading information and before all the dimensions are listed.

Always - Field labels appear before each dimension.

Detailed - If you mark **Always**, this check box becomes available. Select this check box if you want all field labels to appear in the report.

The labels that appear if this check box is selected are: DIMENSION, DESCRIPTION, FEATURE, AXIS, SEGMENT, NOMINALS, MEAS, +TOL, -TOL, BONUS, DEV, OUTTOL, DEVANG, Datum Shift Effect, Unused Zone, Shift X, Shift Y, Shift Z, Rotation X, Rotation Y, and Rotation Z.

If the check box is not selected, only these labels appear: DIMENSION, DESCRIPTION, AXIS, NOMINALS, MEAS, +TOL, -TOL, DEV, and OUTTOL.

Delete Empty - Deletes any empty field from the report.

Data Storage By - Determines how the report is presented: either by row or by column.

Row - Dimensions are laid out horizontally.

Column - Dimensions are laid out vertically.

Data Filter - Determines how the data filters in the output report.

Report FCF dimensions - PC-DMIS passes the Feature Control Frame commands to the Excel file. See the "Using Geometric Tolerances" chapter.

Report dimensions between STATS/ON and STATS/OFF - If you select this check box, you *must* include dimension (or Feature Control Frame dimension) commands between the `STATS/ON` and `STATS/OFF` commands in your measurement routine; otherwise, the data does not export to the Excel file. If you clear this check box, all dimension commands export into the Excel file. See the "Tracking Statistical Data" chapter. The following option becomes enabled:

Report TRACEFIELD - PC-DMIS includes data from `TRACEFIELD` commands in your Excel report if the `TRACEFIELD` commands are within the `STATS/ON` and `STATS/OFF` commands. For more information, see the "Using Tracefields" chapter.

Report COMMENT - Determines whether PC-DMIS includes comments in your Excel report. It has these options:

None - PC-DMIS does not send comments to the Excel report.

REPT - PC-DMIS only sends report comments (`COMMENT/REPT` commands) to the Excel report.

Display on report - PC-DMIS sends all report comments (`COMMENT/REPT` commands), as well as any comment that has its **Display on report** check box marked to the Excel report.

For information on comments, see "Inserting Programmer Comments" in the "Inserting Report Commands" chapter.

Follow dimension output - PC-DMIS filters the data so that the Excel report only includes dimensions that have their outputs set to match whatever you select from this list:

- REPORT
- STATS
- BOTH

For example, if you choose **REPORT**, and you have a Location dimension set to OUTPUT=STATS, it is not included in the Excel report.

For more information on dimension outputs, see these topics:

- For legacy dimensions, see "Output To" in the "Using Legacy Dimensions" chapter.
- For geometric tolerances, see the **Output** area discussed in the "Reporting tab" topic of the "Using Geometric Tolerances" chapter.

Report skipped items - PC-DMIS processes items that the operator skips during execution and includes them in the report. If you clear this check box, PC-DMIS ignores the skipped items. Note that commands are skipped if the operator manually skips them or if an On Error command tells PC-DMIS to skip them. For more information on **Skip**, see "Using the Execution Dialog Box". For information about On Error branching, see "Branching on an Error".

Report Minimum/Maximum Value - Displays the maximum and minimum deviation values from the points that make up the dimension. While you can report the MaxMin for any dimension, it only makes sense for Line Profile and Surface Profile dimensions. For an example of this, see the "Parameter Settings: Dimension tab" topic in the "Setting Your Preferences" chapter.

Data Format - Determines how the output report shows the data:

Insert blank between dimensions - Inserts a blank row or column after each dimension.

Insert '0' instead of blank - Inserts a '0' (zero) for any blank data.

Defaults - Creates a "defaults" file (ExcelDefaults.p2xfile) that stores this dialog box's settings. When you click this button, PC-DMIS automatically updates the contents of the "defaults" file without showing a message.

PC-DMIS uses this defaults file in these cases:

- For new measurement routines
- For the current measurement routine if PC-DMIS cannot locate the routine-specific settings file. (Note that PC-DMIS creates a routine-specific .p2x file in the same directory as the measurement routine whenever you click **OK**.)

PC-DMIS saves the ExcelDefaults.p2xfile here: C:\ProgramData\Hexagon\PC-DMIS\2026.2\. You can view this file in any text editor.

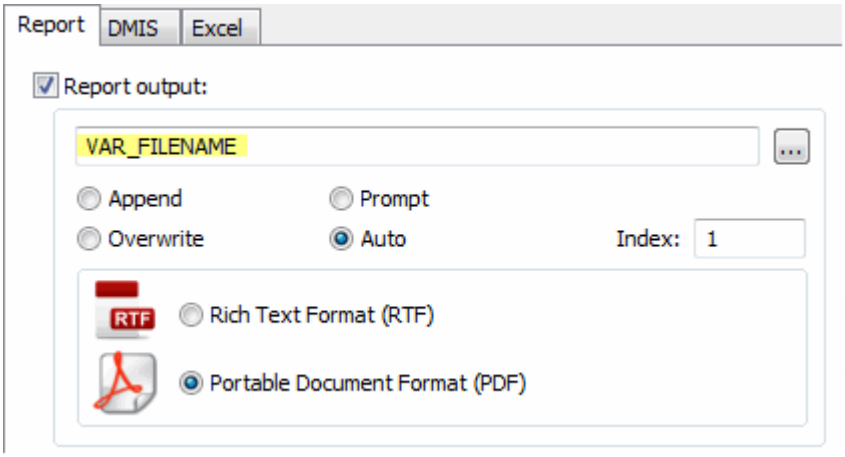
Output to a File Using Expressions

In the **Output Configuration** dialog box (**File | Printing | Report Window Print Setup**), from the **Report** tab, you can use expressions in the **File** box. With this functionality, you can use your measurement routine code along with the **Output Configuration** dialog box to dynamically change the destination directory for the output file.

For example, suppose two users, John and Amy, want to send the same measurement routine's inspection report to an existing subdirectory based on the user name. Instead of opening the **Output Configuration** dialog box for each user and changing the destination directory and the name of the report, you can have the user type their name into a Comment and then use assignment and flow control commands to store the different destination directories and report names in a variable, like this:

```
C1          =COMMENT/INPUT,YES,Type your name:
             IF/C1.INPUT=="John"
                 ASSIGN/VAR_FILENAME="C:\\inspectionreports\
\John\\John.rtf"
                 COMMENT/OPER,NO,VAR_FILENAME
             END_IF/
             ELSE_IF/C1.INPUT=="Amy"
                 ASSIGN/VAR_FILENAME="C:\\inpectionreports\
\Amy\\Amy.rtf"
                 COMMENT/OPER,NO,VAR_FILENAME
             END_ELSEIF/
             ELSE/
                 ASSIGN/VAR_FILENAME="C:\\inspectionreports\
\"+C1.INPUT+".rtf"
                 COMMENT/OPER,NO,VAR_FILENAME
             END_ELSE/
```

Then inside the **File** box, on the **Report** tab of the **Output Configuration** dialog box, you can type the `VAR_FILENAME` variable as shown here:



Example showing VAR_FILENAME in File box.

Thereafter, whenever you execute the measurement routine, if John is the user, PC-DMIS saves the report *John.rtf* to his directory; if Amy is the user, PC-DMIS saves *Amy.rtf* to her directory; if another user types his or her name, it goes to the default *C:\Inspectionreports* directory.

- For information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.
- For information on expressions and variables, see the "Using Expressions and Variables" chapter.
- For information on flow control statements, see the "Branching by Using Flow Control" chapter.

Closing or Quitting a Measurement Routine

- To save and close the active measurement routine, select **File | Close**. If you have never saved the measurement routine, you are prompted to name and save your file.

  You can only save measurement routine (.prg) files to valid versions of PC-DMIS.

  For PC-DMIS 2026.2, the furthest you can save back to is PC-DMIS version 2021.1.

- To quit the current measurement routine, select **File | Quit**. If you have unsaved data, a message appears and asks if you want to quit the measurement routine without saving. To save the measurement routine, choose **NO** in the message box, and choose **File | Save** or **File | Save As** (for more information, follow the instructions under "Saving Measurement Routines").

If you close down all open sub-windows in the software (the Report window, the Edit window, and the Graphic Display window) by clicking on each window's X, PC-DMIS immediately saves the measurement routine and then closes it. If you use the **View** menu to hide the windows, the measurement routine remains open.

Exiting the Software

To quit the current measurement routine and exit PC-DMIS, choose the **File | Exit** command. When you choose **File | Exit**, PC-DMIS automatically saves the current measurement routine before exiting.

Using Advanced File Options

Using Advanced File Options: Introduction

PC-DMIS provides you with many options to manipulate measurement routines and manage your essential files. As with most Windows-based programs, you can perform basic file management using standard Windows dialog boxes; these include creating, opening, copying, renaming, and deleting files. Additionally, through PC-DMIS, you can perform more advanced operations, such as importing and exporting CAD data or executing completed measurement routines.

This chapter specifically deals with advanced file options. For more basic options, see the "Using Basic File Options" chapter.

The options discussed in this chapter include:

- Importing CAD Data or Feature Data
- Exporting CAD Data or Feature Data
- Using Direct CAD Interfaces
- Setting Import Options
- Executing Measurement Routines

These options and commands are described in detail throughout this chapter.

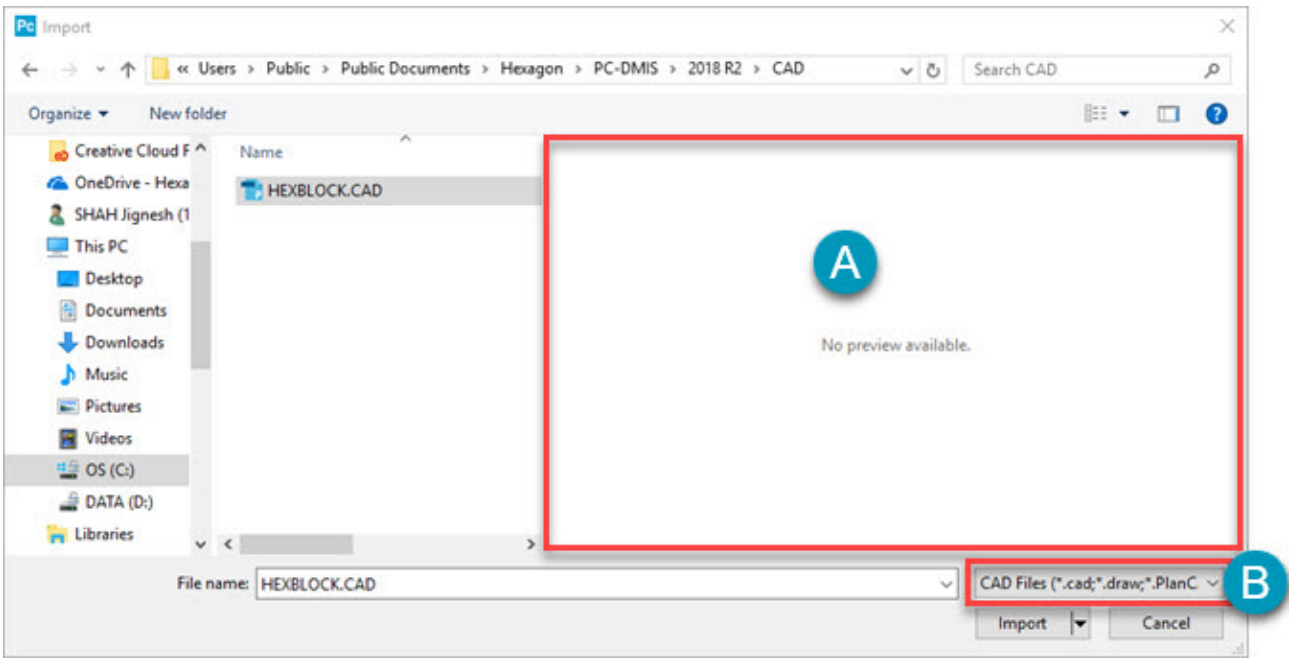
Importing CAD Data or Feature Data

When you select **File | Import**, PC-DMIS displays a submenu of data types that you can import into the current measurement routine. Data types are either:

- CAD data that the software can import into the CAD file in the Graphic Display window or
- Feature data that the software can import into the measurement routine

Once you select a menu item, the software displays an **Import** dialog box so that you can choose the file to import.

The dialog box is a standard Windows **Import** dialog box with the following additions:



Example of the Import dialog box for CAD.

A. **Preview** - This area of the dialog box shows the CAD image for .prg files. This area remains blank for all the other types of files.

B. This file type list displays the various file types that you can import. You can import any of these file types:

- **CAD Data** - ACIS, CAD, CAD by Reference, CATIA, Creo, DES, DXF, EDGECAM, IGES, Inventor, JT, Parasolid, QIF, Solid Edge, SolidWorks, STEP, STL, NX, VDAFS, Wavefront, or XYZ
- **Feature Data** - ASCII, Chorus DMIS, Datalog, DMIS, Pointcloud, MeasureMax, Inspection Plan, CSV, or XYZ. PC-DMIS imports the data into the measurement routine in the PC-DMIS format.

> With DMIS data, the data is imported into the measurement routine in PC-DMIS format, eliminating the need for a separate file.

PC-DMIS imports CAD data as CAD entities into the Graphic Display window.

- PC-DMIS keeps track of the CAD data as a separate file. This file has the same file name as the measurement routine and has a ".cad" extension.

> If you have a measurement routine with a file name of test.prg, once you add CAD data to this measurement routine, PC-DMIS creates a CAD file named test.cad and stores it in the same directory.

- You can use PC-DMIS to import existing .cad files for multiple measurement routines. For information, see "Importing a CAD by Reference".

PC-DMIS imports feature data as feature commands in the Edit window.

Importing CAD Data

PC-DMIS can import CAD data into the Graphic Display window.

These translators are supported:

- ACIS
- CAD
- CAD by Reference
- CATIA DCT (V4, V5, or V6)
- Creo DCT
- DES
- DXF/DWG
- EBM
- EDGECAM
- IGES
- Inventor
- JT
- MetIL
- NX DCT
- Parasolid DCT
- QIF
- Solid Edge DCT
- SolidWorks DCT
- STEP
- STL
- From Teamcenter

- VDAFS
- VISI
- Wavefront Object
- XYZ

If you want to use the CATIA, Parasolid, Creo, SolidWorks, NX, QIF, or VISI translators, your PC-DMIS license must have these applicable DCT license options enabled: **CATIA V4 DCT, CATIA V5 DCT, CATIA V6 DCT, NX DCT, PRO-E DCT**, **Parasolid DCT**, **QIF-DCT**, or **VISI DCT**.

General Import Procedure

The general procedure below imports the CAD data through an **Import** menu. However, you may find it easier to drag and drop the files directly into the Graphic Display window to start the import process. Drag and drop import functions if you run PC-DMIS as a standard user. If you run PC-DMIS as an administrator, this is not allowed, due to Windows UAC security restrictions.

To import CAD data into the Graphic Display window of your inspection plan, follow this general procedure. You can find specifics for your CAD type in the topics below:

1. Select the **File | Import | <CAD File Type>** menu item to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder.
3. From the **Import** dialog box, navigate to the folder that contains the file to import. Then select the file.
4. Click **Import** to import the file. If you already have a CAD model in the Graphic Display window, PC-DMIS asks if you want to replace the existing CAD data or merge the CAD data together.
    - To *add* the new imported data without deleting the existing data, click the **Yes** option.
    - To *replace* the existing data with the new imported data, click **No**.
    - To return to the **Import** dialog box, click **Cancel**.

Some imported file types automatically replace old existing CAD data with the imported data. In those cases, you won't be presented with the **Merge Import Data** dialog box.

If a Model View is active, and you merge an imported CAD file, PC-DMIS adds all the visible geometry of the imported CAD model into the active Model View.

For details on CAD Model View, see "Working with CAD Model Views" in this documentation.

5. PC-DMIS starts to import the CAD data with any accompanying GD&T dimension information.

6. If PC-DMIS detects a reference to the GD&T standard to which your CAD model was designed, PC-DMIS compares the CAD model's GD&T standard to the measurement routine's GD&T standard (see "Creating New Measurement Routines" and "Setup Options: Geometric Tolerances" tab). If the CAD model's GD&T standard differs from that of the measurement routine, PC-DMIS displays this message:

   Your measurement routine GDT standard (ASME Y14.5 - 2018) does not match the imported CAD standard (ISO 1101). Click OK to set the measurement routine to ISO 1101 or Cancel to keep the current routine standard.

   - If you click **OK**, PC-DMIS changes the measurement routine's GD&T standard to match that of the CAD model.
   - If you click **Cancel**, PC-DMIS leaves the measurement routine's GD&T standard unchanged. You are responsible for managing any GD&T discrepancies.

7. If your model contains embedded GD&T callouts, PC-DMIS asks if you want to make the first CAD model view that contains GD&T to be active and to show it in the Graphic Display window.

   - If you click **Yes**, PC-DMIS activates the first CAD model view that contains GD&T and turns on the **Show GD&T** option in the **Operation | Graphic Display Window | CAD Items** menu.
   - If you click **No**, PC-DMIS turns off the **Show GD&T** menu option in the **Operation | Graphic Display Window | CAD Items** menu.

For additional information on CAD GD&T callouts, see "Working with CAD GD&T Callouts".

If you re-import the original CAD model file, PC-DMIS removes any modifications to the GD&T CAD callouts in the Graphic Display window.

In some supported CAD model formats, the **Cancel** and **Stop** buttons are available:

During the import process, you can cancel or stop the import.

- **Cancel** - This button cancels the import altogether. If you already had a CAD model loaded before the import, PC-DMIS retains the previous CAD model.
- **Stop** - This button stops the import, but keeps any imported CAD elements up to that point.

If you import both a part model and a machine model into the Graphic Display window, and you don't have any outstanding manual machine offsets or rotations, PC-DMIS automatically positions the part model to the machine model's table.

If you have manual x,y,z, offsets or rotations from the **Part/Machine** tab of the **Setup Options** dialog box, you need to use that tab's **Auto Position** button to position the part model to the machine table. For more information on the **Auto Position** button, see the "Part Setup area" topic in the "Setting Your Preferences: Introduction" chapter.

Importing Multiple CAD Files

PC-DMIS allows you to import multiple CAD files at the same time.

If you have a wireframe and a solid model of the same part, you can import both into your measurement routine.

You can choose to import a .cad, .draw, or .PlanCAD file.

You can press Ctrl and then click on different files to select multiple files at the same time.

If a Model View is active, and you merge an imported CAD file, PC-DMIS adds all the visible geometry of the imported CAD model into the active Model View.

For details on CAD Model View, see "Working with CAD Model Views" in this documentation.

Importing an ACIS File

ACIS files come from the 3D ACIS Modeler, which is a CAD software package developed by Spatial Corporation (formerly Spatial Technology). You can import ACIS files up to v2016 1.0.

You can choose to import a .sat, .asat, or .sab file.

Importing a CAD File

To import a CAD file, perform the following steps:

You can choose to import a .cad, .draw, or .PlanCAD file.

Your file to import should have the same measurement units as the measurement routine that takes the import. If the files don't have the same measurement units, you can still import the file, but the measurement routine ends up with incorrect information. For example, if you have a circle with an X value of 4 inches in the file to import, the software imports it as 4 millimeters if the measurement routine that takes the import expects millimeters.

You can reference one CAD file from more than one measurement routine. For more information on how to reference the CAD file, see the "Importing a CAD by Reference" topic.

Importing a CAD by Reference

You can have multiple measurement routines reference a single CAD model. If you use large CAD models, to save disk space, you can reference just one .cad file for multiple measurement routines.

You can choose to import a .cad, .draw, or .PlanCAD file.

- If you import a .PlanCAD file, the software imports both the plan features and the embedded CAD model.

- If PC-DMIS detects changes in the original .PlanCAD file that you're referencing, it asks if you want to use the Change Manager to handle those changes. For information on the Change Manager, see "Updating Inspection Plans with the Change Manager".

The software only reloads the CAD model if the original .plancad file is modified.

> You cannot have circular references to .cad files. These result in error messages. For example, an error occurs if you create a measurement routine, import an IGES file, save the measurement routine, and then later attempt to import its own .cad file.

Once imported, you can use the **View | CAD Info** menu option to view information about a CAD element and display the file path for the referenced CAD file. For information, see "View | CAD Info" in the "Editing the CAD Display" chapter.

> Be aware that modifications made to the CAD model, either the original CAD model or a referenced CAD model, are made to all other measurement routines based on that CAD model. For example, if you make changes to the original CAD model, any models referenced from the original are also changed. Conversely, if you make changes to a referenced model, then the original model also has the same changes made.

Importing CATIA Files

PC-DMIS allows you to translate various CATIA file types and import the file type into a PC-DMIS measurement routine. This type of import differs from interfacing directly with the CATIA CAD file. For information on directly interfacing with CAD models, see the "Direct CAD Interface" documentation.

You can choose to import **CATIA 4 Files**, **CATIA 5 Files**, **and CATIA 6 Files**.

- If you select **CATIA 4 Files**, you can import files with .mod, .exp, .iso, and .cat filename extensions.
- If you select **CATIA 5 Files**, you can import files with .CATPart and .CATProduct filename extensions.
- If you select **CATIA 6 Files**, you can import files with .3dxml filename extensions.

CATIA v5 Assemblies

CATIA v5 assemblies can contain embedded CATIA v4 model files. When you use the CATIA v5 DCI or DCT to import a CATIA v5 assembly, PC-DMIS uses the CATIA v4 DCT to translate any embedded CATIA v4 model files. As such, your PC-DMIS license must be programmed for the CATIA v4 DCT. Otherwise, the embedded CATIA v4 model files are

not translated. For information on assemblies, see "Working with Assemblies of Parts" in the "Editing the CAD Display" chapter.

CATIA v5 Model Views

PC-DMIS supports CATIA v5 CAD model views. For more information, see "Working with CAD Model Views".

Importing a Creo File

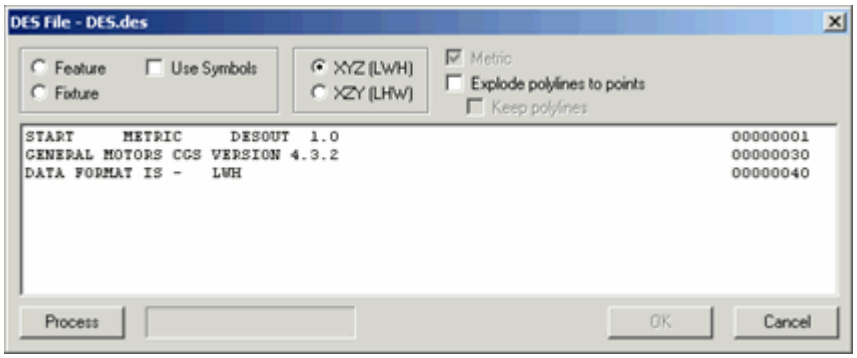
PC-DMIS allows you to translate a Creo (previously Pro/ENGINEER) file and import it into a PC-DMIS measurement routine. This type of import differs from interfacing directly with the Creo CAD file as described in the "Installing and Using Creo Direct CAD Interface" topic in the "Direct CAD Interfaces and Translators" documentation.

You can choose to import a .prt, .asm file, .xar, or .xpr file.

Importing a DES File

You can choose to import a .des file.

When you click **Import**, PC-DMIS displays the **DES File** dialog box.



DES File Dialog Box

Feature or **Fixture** - This option specifies what type of data you want to import. If you select **Feature**, you can select the **Use Symbols** check box to use feature labels.

XYZ (LWH) or **XZY (LHW)** - This option specifies the orientation for the imported data.

Metric - Use this check box if the needed units are not specified in the DES file.

Explode polylines to points - This check box converts imported lines into points. Additionally, you can select the **Keep polylines** check box to keep the imported polylines.

Process - This button imports the DES file.

OK - This button accepts the processed file.

Once you import the DES file you can create features similar to other CAD formats. For more information, see "DES Input" in the "Working in Offline Mode" appendix.

Importing a DXF or DWG File

You can choose to import a .dxf or .dwg file.

PC-DMIS stores the units of DXF files in the measurement property inside the DXF file. The measurement property is indicated by the text "$MEASUREMENT" in the DXF file.

Fixing a Missing Measurement Property

Some DXF files may omit the measurement property. In these cases, PC-DMIS assumes that the DXF file is in inch units. If the units of the DXF file should be millimeter, and you import the DXF file into PC-DMIS, PC-DMIS scales the CAD model too large by a factor of 25.4.

To fix this problem, you can use the **CAD Transform** dialog box to scale the model by a factor of 1/25.4:

1. Choose **Operation | Graphic Display Window | Transform** to access the **CAD Transform** dialog box.
2. In the **Scale** box, type **1/25.4**.
3. Click **OK** to scale the model down by that factor.

For more information on the **CAD Transform** dialog box, see "Transforming a CAD Model" in the "Editing the CAD Display" chapter.

Fixing an Incorrect Measurement Property

Some DXF files may have an incorrect measurement property. For example, the measurement property may indicate inch units but the intended units are millimeters. If you import the DXF file into PC-DMIS, PC-DMIS scales the CAD model too large by a factor of 25.4.

To fix this problem, you can edit the DXF file in a text editor:

1. Open the DXF file in a text editor, such as Notepad.
2. Search for the text "$MEASUREMENT" in the file.

   For example, the text could look something like this:

   ```
   $MEASUREMENT

   70

   0

   0
   ```

This measurement property defines the drawing units. The first number that follows group code 70 defines the drawing units for the CAD model. It can be 0 or 1 and signifies the following:

0 = inch

1 = millimeters

3. Modify the drawing units to the intended units: 0 for inch or 1 for millimeter.

   So, in the above example, to change the CAD model units to millimeter, change the drawing units property from 0 to 1, like this:

```
$MEASUREMENT
70
1
0
```

4. In your text editor, save your changes to the DXF file.
5. In PC-DMIS, import the modified DXF file.

Importing an EBM File

The EBM file type is a text file that defines features and dimensions. When you import an EBM file, PC-DMIS creates features and dimensions. PC-DMIS creates the features and dimensions based on their parameters in the EBM file. For those parameters not explicitly defined in the EBM file, PC-DMIS uses previously defined default parameters.

To show the plus and minus dimension tolerance values as they appear in the EBM file, you must select the **Minus Tols Show Negative** check box. You can find this in the **Output options** area on the **Dimensions** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**).

When you import this file type, PC-DMIS performs these error checks:

- Corrupted file
- File format and syntax errors
- Unsupported input feature types

If the software finds an error, it displays a message with a brief description of each error.

EBM File Import

File Name: D:\EBM_Files\EBM_TestFile_01.ebm

Line 13: Ignored the command REFERENTIELLE (NOM="CAISSE")

LINE 41: Ignored the command PT_GEOM_SUR_ (NOM="9802PR31G_Z121"

LINE 123: Wrong or missing fields in the command CERCLE (NOM="9802PR_XY9")

LINE 137: Wrong or missing fields in the command OBLONG (NOM="9802PR_X6")

Imported 107 features

Importing an EDGECAM File

For a general procedure, see "Importing CAD Data".

You can import geometry information from an EDGECAM file (.ppf) into the Graphic Display window.

You can choose to import a .ppf file.

Importing an IGES File

You can choose to import .igs or .iges files.

If you want to manipulate a 2D CAD drawing three-dimensionally, thereby creating desired 3D levels, you can use CAD-level functionality. If you do this, your original data should be defined in a single plane parallel to the Z (= 0) plane. For complete information on CAD levels, see the "Working with CAD Levels" topic in the "Editing the CAD Display" chapter.

For information about the different input file formats, see the "Working in Offline Mode" section.

Importing an Inventor File

You can choose to import a .ipt file (CAD file) or a .iam (CAD assembly). They are the Autodesk Inventor model filename extensions.

Importing a JT File

The JT file format is a relatively light-weight file format capable of containing large assemblies of parts with faceted data, lighting, textures, curves and surfaces; product and manufacturing information (GD&T callouts); and other information. It is useful for visualization and collaboration.

You can choose to import a .jt file.

Importing a MetIL File

MetIL files come from the Metrology Mentor, which is a part of the Nexus platform. MetIL files are in a .json format that contains measurement instructions. For each Metrology Mentor measurement plan, there are two files:

<planname>.metil.json and <planname>.analysis.json, where <planname> is the name of Metrology Mentor plan.

MetIL Converter

PC-DMIS MetIL Converter converts MetIL instructions into PC-DMIS commands. You need to add alignment to the measurement routine.

To convert MetIL instructions into PC-DMIS commands:

1. Create a Metrology Mentor measurement plan.
2. From the measurement plan, generate a program to create the MetIL and Analysis files.
3. Download the MetIL and Analysis files.
4. Open PC-DMIS.
5. Create a measurement routine in inch or mm units.

   In the **New Measurement Routine** dialog box, you must select the **ASME Y14.5 - 2009/2018** option from the **GD&T Standard** list.

6. Import the same CAD file that you used to create the MetIL file.
7. From the PC-DMIS menu, select **File | Import | MetIL**.
8. Navigate to the folder where you stored the MetIL files.
9. Select a <planname>.metil.json file.

PC-DMIS starts the conversion and display the commands in the Edit window.

Importing an NX File

PC-DMIS allows you to translate an NX (previously Unigraphics) file and import it into a PC-DMIS measurement routine. This type of import differs from interfacing directly with the NX CAD file described in the "Direct CAD Interfaces" documentation.

You can choose to import a .prt file.

If you have an NX model with global tolerances, PC-DMIS applies those tolerances to non-basic dimensions without tolerances during the model import.

Importing a Parasolid File

You can choose to import a .x_t, .x_b, .xmt_txt, or .xmt_bin file.

Importing a QIF File

You need a license to use this option.

PC-DMIS allows you to translate Quality Information Framework (QIF) file types and import them into a PC-DMIS measurement routine. The QIF file type conforms to the ISO standard for the CAD neutral file format and 3D CAD files that contain Product Manufacturing Information (PMI).

You can import .qif file types the same way you import any file type into PC-DMIS from the **File | Import** menu option.

PC-DMIS only supports QIF version 3.0.0.

Importing a Solid Edge File

PC-DMIS allows you to translate Solid Edge file types and import them into a PC-DMIS measurement routine. Solid Edge is a CAD modeler that is similar to SolidWorks, NX, Creo, and CATIAv5.

You can choose to import a .par, .asm, .psm, or .pwd file.

Importing a SolidWorks File

PC-DMIS allows you to translate SolidWorks file types and import them into a PC-DMIS measurement routine.

You can choose to import a .sldprt or .sldasm file.

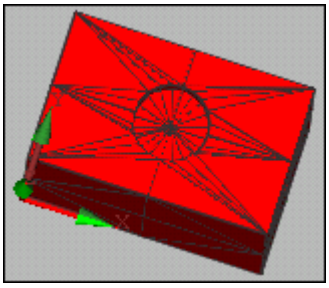
Importing a STEP File

This is a 3D model file formatted in STEP (Standard for the Exchange of Product Data).

You can choose to import a .stp, .step, or .stpZ file.

Importing an STL File

The stereolithography (STL) format, an ASCII or binary file used in manufacturing, provides a list of the triangular facets that describes a computer generated solid model. This format is the standard input for most rapid prototyping machines.



Example of an imported STL file.

You can choose to import a .stl or .stla file.

Importing from Teamcenter

Teamcenter is a PLM (Product Lifecycle Management) database package developed by Siemens. You can import CAD models stored in the Teamcenter database.

For information on how to import CAD models, see "Step 3 - Load Supported Models" in the "Using Teamcenter" chapter of the PC-DMIS DCI documentation.

Importing a VDAFS File

You can choose to import a .vda file.

A Note on Importing Point Data from PSET (Point Set) Objects

If the .VDA file you are importing contains point data within PSET Objects, PC-DMIS may create a .cad file with fewer CAD entities than what actually appears on the screen.

To get a number of CAD entities equivalent to the number of displayed points on the Graphic Display window, follow this workaround:

1. Close PC-DMIS.
2. Start the PC-DMIS Settings Editor.
3. Expand the **ImportSettings** section.

4. Set the `ExplodePolylines` entry to TRUE.
5. Set the `KeepPolylines` entry to FALSE.
6. Save your changes to the JSON file, close PC-DMIS Settings Editor, and restart PC-DMIS.
7. Import the .vda file again.

For information on how to modify entries, see the "Modifying Setting Entries" section.

Importing a VISI File

You can import geometry information from a VISI file (.wkf) file into the Graphic Display window. VISI modeling is a surface and solid modeling system that is based around the Parasolid geometric modeling kernel.

You can choose to import a .wkf file.

Importing a Wavefront Object File

For a general procedure, see "Importing CAD Data".

You can import geometry information from a Wavefront object (.obj) file into the Graphic Display window. The .obj file is an open 3D model format that many 3D graphics applications use.

You can choose to import a .obj file.

Importing an XYZ File as CAD Data

An XYZ file is a simple text file with a .xyz extension. It can contain feature information in the form of xyz coordinates with or without a vector. You can use any standard text editor to view the values in an .xyz file.

You can choose to import a .xyz file.

When you click **Import**, PC-DMIS interprets the data in the .xyz file and imports it into the Graphic Display window as separate CAD entities.

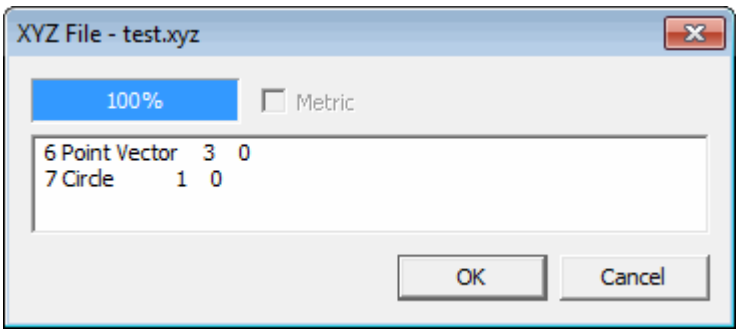
The software creates each feature based on how many comma-separated numbers are on each line. PC-DMIS reads the rows of the .xyz data file and converts the points to the following CAD entity depending on the amount of numbers per row:

Numbers per Row	Description	CAD Entity Generated
3	First number = X value Second number = Y value Third number = Z value	Point with no vector
4	First number = X value Second number = Y value Third number = Z value Fourth number = diameter	CAD circle with a workplane based vector
5	First number = X value Second number = Y value Third number = Z value Fourth number = nothing Fifth number = nothing	Point with no vector
6	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value	Point with a vector

7	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value Seventh number = diameter	Circle with vector
8+	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value Seventh number = nothing Eighth number = nothing	Point with a vector

If your XYZ file contained a line with 25,280,750,25, PC-DMIS would create a circle with the center at x=25, y=280, and z=750 with a diameter of 25.

After you select the file to import, PC-DMIS displays the features constructed from the XYZ file in the **XYZ File** dialog box.



XYZ File Dialog Box

Be aware that the order in which the features are created in the .xyz file and the order in which they are generated inside PC-DMIS may not be the same.

After you click **OK**, if you imported the file into the Graphic Display window, PC-DMIS displays the created CAD elements there. If you imported the file into the measurement routine, the features appear in the Edit window.

Notes on XYZ Files

PC-DMIS can read in an .xyz text file that contains XYZ (and possibly IJK) data. The file should contain the nominal (theoretical) inspection points that need to be measured.

The file must follow these rules:

1. Columns 1 - 6 in the first line of the file must contain the characters "XYZIJK" (without quotes). PC-DMIS uses this sequence of characters as a keyword to distinguish this file type from DES and IGES files. *It must be in capital letters*.

   In the first line of the file:

   Col. 1 = X

   Col. 2 = Y

   Col. 3 = Z

   Col. 4 = I

   Col. 5 = J

   Col. 6 = K

2. If the file contains metric data, columns 11 - 16 of the first line must contain the word METRIC. That is:

   Col. 11 = M

   Col. 12 = E

Col. 13 = T

Col. 14 = R

Col. 15 = I

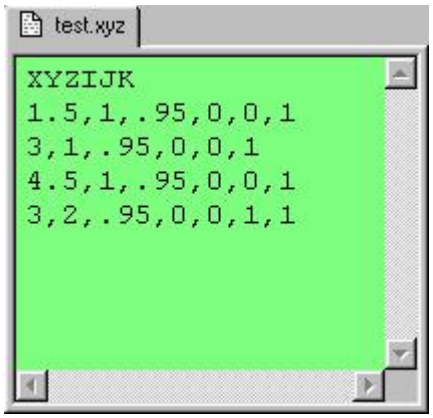
Col. 16 = C

The default, if the METRIC keyword is not present, is INCH.

The rest of the first line should be blank.

3. Each line thereafter *must* contain three to eight floating point numbers *separated by a comma*.

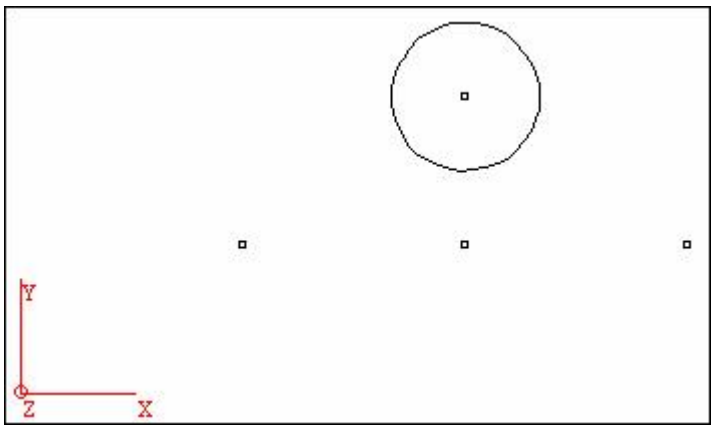
   For example, suppose your XYZIJK file looked like this sample:



Example of a XYZIJK file in a text editor.

   - Line 1 contains the XYZIJK keyword, and since the keyword METRIC isn't on columns 11 through 16, this defaults to INCH.
   - Line 2 produces a point with a vector.
   - Line 3 produces a point with a vector.
   - Line 4 produces a point with a vector.
   - Line 5 produces a circle with a vector and a diameter.

   Once imported, it produces this result in the Graphic Display window:



Example showing the Graphic Display window with data imported from the XYZIJK file.

Importing Feature Data

You can import feature data stored in another file into your measurement routine. This adds additional feature commands into the measurement routine:

It does not affect any CAD model entities displayed in the Graphic Display window.

Importing an ASCII File

ASCII files are text files that contain measurement routine information output from virtually any third party CMM software. To import these file types into PC-DMIS, select **File | Import | ASCII** to automatically create a measurement routine.

> For this menu item to appear, your PC-DMIS license must be programmed to support the **IP Measure** module.

PC-DMIS's Generic Parser (GP) is designed to parse the ASCII output from other CMM systems. The ASCII output (file) may contain file header, comments, feature commands, file footer, etc. As long as the info is written line by line in the ASCII file, GP can parse it according to the user-defined rules provided through the Rule Set Wizard. After parsing it, GP translates all the filtered-through info (like feature commands) and imports it into the active measurement routine.

> Nine PC-DMIS feature types are supported by GP. The supported features are: POINT, EDGEPOINT, LINE, PLANE, CIRCLE, CYLINDER, SPHERE, SQSLOT (Square Slot) and RDSLOT (Round Slot). See "Step 3: Feature Type Mapping and Field Assignment" topic.

To import an ASCII file as an inspection plan, follow these steps:

1. Select **File | Import | ASCII File** to open the **Import** dialog box.

2. From the file type list, select the file type that filters the current folder. If you select **ASCII Files**, you can choose to import any supported file.
3. From the **Import** dialog box, navigate to the folder that contains the file to import. Then select the file.
4. Click **Import** to import the file into the Graphic Display window. PC-DMIS displays the **Importing** dialog box. You can use this dialog box to create new rule sets, add existing rule sets, and start the import process. For more information, see "Using the Importing Dialog Box".
5. From the list of rule sets in the **Importing** dialog box, check to see if the needed rule set for your ASCII file is already visible. This rule set defines how to import the chosen ASCII file.
    - If you have an existing rule set but it isn't already added into the **Importing** dialog box, click **Add** to add it into the dialog box.
    - If you don't have a rule set defined, click **Create** to create a new rule set. These topics describe how to create a rule set.

        Rule Set Wizard - Step 1: Excluding Data

        Rule Set Wizard - Step 2: Defining Delimiters

        Rule Set Wizard - Step 3: Feature Type Mapping and Field Assignment

        Rule Set Wizard - Step 4: Handling Labels, Conflicts, and Point Offsets

        Creating a Rule Set - Previewing the ASCII File
6. Select the rule set to use for importing the ASCII file.
7. Once you have your rule set defined and selected, choose how you want to import the ASCII file:
    - **Directly** - This imports the file without showing the Rule Set Wizard first.
    - **By Wizard** - This opens the selected rule set in the rule set wizard so you can preview and modify any rules before importing the file.
8. Click **Import to Measurement Routine** to parse and import the ASCII file. Follow any on-screen instructions.

Using the Importing Dialog Box

The **Importing** dialog box allows you to create, edit, add, or delete rule sets that you can use to evaluate your ASCII file.

The selected rule set determines which information PC-DMIS imports into your measurement routine. Use the following options to select and apply rules for parsing your ASCII file.

Create - This option opens Step 1 of the Rule Set Wizard. See "Rule Set Wizard - Step 1: Excluding Data". PC-DMIS displays the values of the selected Rule Set, but if you do not select a Rule Set, PC-DMIS does not display any values.

Edit - This option allows you to edit the rule set selected from the **Rule Set List**.

Add - This option opens an **Open** dialog box that allow you to navigate to and select a preexisting rule set. PC-DMIS copies the rule set file into the designated rule file directory and then displays a new entry in the list box

Delete - This option removes the selected rule set from the **Rule Set List**.

Set as default - This option sets the highlighted rule set in the **Rule Set List** as the default. PC-DMIS then uses this rule set until you select another rule set later. If you do not select a new rule set, PC-DMIS sets the last used rule set as the default and highlights it in the list box. The next time that you import an ASCII file, PC-DMIS highlights the default rule set.

Import to Measurement Routine - PC-DMIS imports the opened ASCII file into the current measurement routine:

- **Directly** - Select this option if you want to import the ASCII file using the selected rule set *without* using the Rule Set Wizard.
- **By wizard** - Select this option if you want to open Step 1 of the Rule Set Wizard. When you click finish on Step 3, PC-DMIS imports the file using any newly specified rules.

Cancel - Select this option to abort the ASCII Import process

PC-DMIS stores all rule set files in the same directory designated by the GP. The directory is named "AscImportRules", where the rule set files are named "*.ascimportrule".

Rule Set Wizard - Step 1: Excluding Data

This step of creating a rule set helps you exclude unwanted lines of data from the import.

The **Rules for parsing lines** provided in this step determines which lines of your ASCII file will included for importing into your measurement routine. This appears when you click **Create...** from the **Importing** dialog box. Use the following options to exclude data from import:

Skip file header - When this option is selected, the specified **Number of lines** from the top of the ASCII file will be skipped. Therefore, they will not be evaluated or imported into your measurement routine.

Skip file footer - When this option is selected, the specified **Number of lines** from the bottom of the ASCII file will be skipped. Therefore, they will not be evaluated or imported into your measurement routine.

For lines containing - Lines containing the text specified to the right will be evaluated on the following criteria:

Qualifying search text is CASE specific

- **Skip the entire line** - Entire line is excluded from being evaluated if qualifying text is present.
- **Truncate the strings after** - Anything after the qualifying text will be excluded from being evaluated.
- **Truncate the strings before** - Anything before the qualifying text will be excluded from being evaluated.
- **Truncate the strings until** - Anything after the first qualifying text and prior to the text specified to the right of this option will be excluded from being evaluated.

Use the **Add>>** and **<<Remove** buttons to update the criteria to the table on the right side of this dialog box. Multiple criteria can be used to evaluate your ASCII file.

Preview and Refresh - See the "Rule Set Wizard - Previewing the ASCII File" topic.

Next - Continue with Step 2.

Cancel - Returns to the **Importing** dialog box.

Rule Set Wizard - Step 2: Defining Delimiters

This step of creating a rule set helps you determine how you want to parse the data in the ASCII file, by column or by a delimiting character. You can also define the measurement units to use.

The **Delimiter** information provided in this step determines the columns for each of the parsed lines. Use the following options to parse your data.

By column - Use this option for ASCII files that use fixed lengths for fields. This allows you to separate each field based on the length of the field. This may require a little trial and error to define the correct lengths for each field.

- **Add** - Specify the **Beginning** and **Ending** characters of the field and click **Add**. The next column number will be assigned with the **Beginning** and **Ending** values. Continue this process of adding columns until all the needed data is included in the appropriate columns.
- **Remove** - Select the column you need to remove and click **Remove**. The column is removed from the list.
- **Change** - Select the column you need to change, alter the **Beginning** and **Ending** values for the column and click **Change**. The column is updated with the new information.

By character - Use this option for ASCII files where fields are separated by tabs, spaces, commas or other characters. Select the separating character, tab or space.

- **Specific character** - Select this option and specify the one or more **Specific characters** that separate the fields in your ASCII file. Type the desired character and click **Add** to add it to the **Character List**. Select a character from the **Character List** and click **Remove** to remove a character.
- **Tab**- Select this option and click **Add** if the fields of your ASCII file are separated by tabs. **Tab** is added to the character list.
- **Space(s)** - Select this option and click **Add** if the fields of your ASCII file are separated by one or more spaces. **Space** is added to the character list.

The **Units** area specifies the measurement units to use for the imported values:

- **MM** - Millimeter
- **CM** - Centimeter
- **DM** - Decimeter
- **Inch** - Inch

Preview and Refresh - See the "Rule Set Wizard - Previewing the ASCII File" topic.

Back - Returns to Step 1.

Next - Continue with Step 3.

Cancel - Returns to the **Importing** dialog box.

Rule Set Wizard - Step 3: Feature Type Mapping and Field Assignment

This step of creating a rule set helps you to map feature types to the features in your ASCII file. Additionally, you can assign the field names to the delimited columns.

Feature Type - Select the check box(es) of the feature type(s) that are to be imported. All of the available feature types are provided in the **Feature Type** list.

To map the feature type name:

1. If a feature type exists in your ASCII file, highlight the line in the **Feature type** area.
2. Click in the list box under the heading "In ASCII file" on the same highlighted line.
3. Type the feature type name exactly as it appears in the ASCII file in the edit box.

If there are multiple feature types in the ASCII file, but a feature type name is not defined in the file, the parser doesn't have a way to differentiate which feature is a point, circle, and so forth. However, if there is only one feature type in the ASCII file, you don't need to define the feature type name. You only need to mark the check box to the left of the feature type.

Feature Fields - The available fields for the selected **Feature Type** are displayed for assignment. Each field represents a valid PC-DMIS value for the selected feature type.

To assign feature fields to the columns of the ASCII file:

1. Select a **Feature Type**.
2. Select the needed field name(s) from the **Field List**. To select consecutive fields, click the first item, press and hold down SHIFT, and then click the last item. To select nonconsecutive fields, press and hold down CTRL, and then click each item.
3. Click **Add** to add the field name to the list box on the right.

You may also ADD or REMOVE fields by double-clicking the item in the **Field List** or the list box on the right.

4. Use the **UP** and **DOWN** arrows to ensure that the field values appear in the order that the corresponding values appear in the ASCII file.
5. Select any unneeded field(s) from the list on the right and click **Remove** to remove that field name(s). To select consecutive fields, click the first item, press and hold down SHIFT, and then click the last item. To select nonconsecutive fields, press and hold down CTRL, and then click each item.

PC-DMIS may not accept certain fields in the ASCII file that may be proprietary to a third-party CMM system provider's software. For such a field, the user can add the field called "Skip" to inform the GP to skip the column that contains this field.

Preview and Refresh - See the "Rule Set Wizard - Previewing the ASCII File" topic.

Back - Returns to Step 2.

Next - Continue with Step 4.

Cancel - Returns to the **Importing** dialog box.

Rule Set Wizard - Step 4: Handling Labels, Conflicts, and Point Offsets

This step of creating a rule set helps you define how your features are handled as they're imported. You can define the feature labels, what to do if imported feature names conflict with existing feature names, and whether or not points are offset by a defined amount in XYZ.

Feature Label - If you want to define the base name for the features to import, mark **Use base name** and then type the name of the feature. For example, if you're importing point features, you might want to type "PNT" or "Point".

- **Use base name** - Mark this to define the base name for the imported features. If this is cleared, PC-DMIS uses the feature type as the base name.
- **Base name** - Type the base name the process uses for the imported features if **Use base name** is marked.

Creation Confliction - Use this area to determine how to handle conflicts with existing features of the same name.

- **Update existing nominals** - This updates existing features with the imported data. During import, the process checks the current measurement routine's existing feature names and attempts to update matching features with the information contained in the ASCII file. If you have more than one feature in the measurement routine with the same ID, only the top-most feature in the measurement routine is updated. When the import process updates the nominal values of an existing feature, a dialog box appears showing all the features that had nominal values updated.
- **Create new feature with a -1 extension** - This adds an underscore and a number to the feature ID to make it unique. So, if PNT1 was a duplicate, this would be changed to PNT1_1. If there was another PNT1, it would be renamed to PNT1_2 and so on.

Point Offset - This area lets you offset imported points by a specified location.

- **Offset the points** - If marked, points are offset by the distance in the offset boxes below. If cleared, no offset takes place.
- **X offset** - This defines the offset distance in the X direction.
- **Y offset** - This defines the offset distance in the Y direction.

- **Z offset** - This defines the offset distance in the Z direction.

Back - Returns to Step 3.

Cancel - Returns to the Importing dialog box.

Finish - This button functions according to how the Rule Set Wizard was started.

- If you started by clicking **Create**, the Rule Set Wizard is closed and a Create new rule set dialog box appears. Type the name of the new rule set in the Rule set name box and click OK. A message box informs you that the rule file is saved.
- If you started by clicking **Edit**, the Rule Set Wizard is closed, and any changes are applied to the edited rule set file.
- If you started by clicking **Import to Measurement Routine** (By wizard), The rule set is applied, and the parsed ASCII file is imported into PC-DMIS as an Inspection Plan. If any changes have been made to the rule set, then you will also be prompted to save your changes.

Rule Set Wizard - Previewing the ASCII File

As you make changes in each step of the Rule Set Wizard, you can preview the affect that the rules has on your ASCII file by using these items at the bottom of the wizard.

Preview - Marking this check box displays the results of the evaluated file in a preview drop-down pane.

Refresh - Updates the view of your file based on the changes that you made to the evaluation criteria.

No Rule Set Applied

Below is an example preview of an ASCII Inspection Plan file where no rules have been applied. Any ASCII file that contains features can be used to import.

The file used as an example is an Inspection Plan from SolidWorks with a .ip extension. The Generic Parser can work with any ASCII file with any extension.

No.	FeatType	Col1
1	Unknown	PCDIP/{Inspection Plan},1.0;
2	Unknown	UNITS/MM;
3	Unknown	FEAT/POINT,F1,14.75512,0.0,-27.61744,0.0,-1.0,0.0;
4	Unknown	FEAT/EDGEPOINT,F2,46.729183,0.0,0.0,0.0,0.0,1.0,0.0...
5	Unknown	FEAT/LINE,F3,92.487095,0.0,-27.319991,114.180011,0...
6	Unknown	FEAT/PLANE,F4,9.506888,11.536392,-21.0,0.0,0.0,1.0;
7	Unknown	FEAT/CIRCLE,F5,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
8	Unknown	FEAT/CIRCLE,F6,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
9	Unknown	FEAT/CIRCLE,F7,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
10	Unknown	FEAT/CIRCLE,F8,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
11	Unknown	FEAT/CYLINDER,F9,0.0,0.0,0.0,0.0,0.0,1.0,0.0,0.0,INN...
12	Unknown	FEAT/RDSLOT,F12,188.135396,76.5,-2.060396,0.0,-1.0...
13	Unknown	FEAT/SQSLOT,F13,63.529348,24.950494,-3.481529,0.7...
14	Unknown	FEAT/SPHERE,F14,0.0,0.0,0.0,0.0,0.0,1.0,0.0,OUTER;
15	Unknown	ENDIP;

ASCII File with NO Rules applied

Application of Step 1 of Rule Set Wizard

Below is an example of the ASCII file after the values shown in Step 1 were defined and **Refresh** was clicked. Lines 1, 2, 14, and 15 are excluded based on the defined criteria.

No.	FeatType	Col 1
1	Skip	
2	Skip	
3	Unknown	FEAT/POINT,F1,14.75512,0.0,-27.61744,0.0,-1.0,0.0;
4	Unknown	FEAT/EDGEPOINT,F2,46.729183,0.0,0.0,0.0,0.0,1.0,0.0...
5	Unknown	FEAT/LINE,F3,92.487095,0.0,-27.319991,114.180011,0...
6	Unknown	FEAT/PLANE,F4,9.506888,11.536392,-21.0,0.0,0.0,1.0;
7	Unknown	FEAT/CIRCLE,F5,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
8	Unknown	FEAT/CIRCLE,F6,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
9	Unknown	FEAT/CIRCLE,F7,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
10	Unknown	FEAT/CIRCLE,F8,0.0,0.0,0.0,0.0,0.0,1.0,0.0,INNER;
11	Unknown	FEAT/CYLINDER,F9,0.0,0.0,0.0,0.0,0.0,1.0,0.0,0.0,INN...
12	Unknown	FEAT/RDSLOT,F12,188.135396,76.5,-2.060396,0.0,-1.0...
13	Unknown	FEAT/SQSLOT,F13,63.529348,24.950494,-3.481529,0.7...
14	Skip	
15	Skip	

ASCII File with Step 1 Rules applied

Application of Step 2 of Rule Set Wizard

Below is an example of the ASCII file after the values shown in the **By character** area in Step 2 were defined and **Refresh** was clicked. Columns have been added based on the comma separator (Col1-Col14).

N..	FeatT...	Col 1	Col 2	Col 3	Col 4	Col 5	Col 6
1	Skip						
2	Skip						
3	Unknown	FEAT/POINT	F1	14.75512	0.0	-27.61744	0.0
4	Unknown	FEAT/EDGEPOINT	F2	46.729183	0.0	0.0	0.0
5	Unknown	FEAT/LINE	F3	92.487095	0.0	-27.319991	114.180011
6	Unknown	FEAT/PLANE	F4	9.506888	11.536392	-21.0	0.0
7	Unknown	FEAT/CIRCLE	F5	0.0	0.0	0.0	0.0
8	Unknown	FEAT/CIRCLE	F6	0.0	0.0	0.0	0.0
9	Unknown	FEAT/CIRCLE	F7	0.0	0.0	0.0	0.0
10	Unknown	FEAT/CIRCLE	F8	0.0	0.0	0.0	0.0
11	Unknown	FEAT/CYLINDER	F9	0.0	0.0	0.0	0.0
12	Unknown	FEAT/RDSLOT	F12	188.135...	76.5	-2.060396	0.0
13	Unknown	FEAT/SQSLOT	F13	63.529348	24.950494	-3.481529	0.707107
14	Skip						
15	Skip						

ASCII File with Step 2 Rules applied

Application of Step 3 of Rule Set Wizard

Below is an example of the ASCII file after the features types and field names have been assigned in Step 3 and **Refresh** was clicked. Feature types have been assigned based on the Feature Type field (Col1). Lines that have been successfully assigned to be imported as a feature type are displayed with blue text.

No.	Feature	Col 1	Col 2	Col 3	Col 4	Col 5	Col 6
1	Skip						
2	Skip						
3	POINT	FEAT/POINT	F1	14.75512	0.0	-27.61744	0.0
4	EDGEP...	FEAT/EDGEPOINT	F2	46.7291...	0.0	0.0	0.0
5	LINE	FEAT/LINE	F3	92.4870...	0.0	-27.319...	114.180...
6	PLANE	FEAT/PLANE	F4	9.506888	11.5363...	-21.0	0.0
7	CIRCLE	FEAT/CIRCLE	F5	0.0	0.0	0.0	0.0
8	CIRCLE	FEAT/CIRCLE	F6	0.0	0.0	0.0	0.0
9	CIRCLE	FEAT/CIRCLE	F7	0.0	0.0	0.0	0.0
10	CIRCLE	FEAT/CIRCLE	F8	0.0	0.0	0.0	0.0
11	CYLIND...	FEAT/CYLINDER	F9	0.0	0.0	0.0	0.0
12	RDSLOT	FEAT/RDSLOT	F12	188.135...	76.5	-2.060396	0.0
13	SQSLOT	FEAT/SQSLOT	F13	63.5293...	24.9504...	-3.481529	0.707107
14	Skip						
15	Skip						

ASCII File with Step 3 Rules applied

Importing a ChorusNT File

The migration path from ChorusNT to PC-DMIS is based on PC-DMIS's ability to import a Chorus DMIS file, converting the original DMIS file into equivalent PC-DMIS commands that make up a PC-DMIS measurement routine file, a file with a .prg filename extension.

PC-DMIS converts most DMIS commands into PC-DMIS commands to maximize the automatic conversion and minimize the manual modifications needed after the automatic import.

To begin the import process of ChorusNT extensions, perform the following steps:

1. Select **File | Import | Chorus DMIS** to open the **Import** dialog box.

2. From the file type list, select the file type that filters the current folder. If you select **Chorus DMIS Files**, you can choose to import a .dmi, .dms, or .dmis file.
3. From the **Import** dialog box, navigate to the folder that contains the file to import. Then select the file.
4. Click **Import** to import the file into the Graphic Display window. PC-DMIS displays the **Dmis Import Setup** dialog box.

You can use this dialog box to set the following required information during the import phase:

- Specify the probe file name used for each arm
- Enable the warning dialog at the end of the import phase
- Enable some options relating to specific customers and hardware configurations

Taking the basic steps in the following topics can help you correctly import and translate your DMIS file into a PC-DMIS measurement routine file.

Step 1: Configure PC-DMIS

Verify PC-DMIS's setup, and check if the standard settings in PC-DMIS correspond to ChorusNT settings. For example, verify the CMM axis conventions and probe head Orientation and Top Speed (CMM max move speed) value.

Use the PC-DMIS Settings Editor to set the following entries in the **DMIS** section:

- `DMISMaxMeasurementVelocityMMPS` - Define your maximum measurement speed.
- `DMISFedratPcntOfMaxMachineSpeed` - Set this to 0.

For information on setting entries, see the "Modifying Setting Entries" appendix.

Step 2: Create and Calibrate the Probe File Inside PC-DMIS

- Create a new measurement routine and, when prompted, define a new probe (**Insert | Hardware Definition | Probe**). Name it MYPROBE.PRB and specify in the **Probe Utility** dialog box the components according to your specific needs.
- Import the Chorus qualification program. Use as your probe, the previously created MYPROBE.PRB file. PC-DMIS imports each SNSDEF statement and adds the corresponding TIP angles. Note that Chorus identifies each tip with the DMIS label that looks something like this: a tip with roll = 0 and pitch = 0 is named S(**R**000**P**000T1). PC-DMIS names this as T1A0B0.

- If in ChorusNT you used a self-calibrated probe, import into PC-DMIS a simple DMIS routine that contains the definition of the "Chief" probe. It needs to have the right label (the same used in the SNSLCT/S(label),90,90 for example) and Pitch and Roll equal to 0.
- After you import the DMIS file, your MYPROBE.PRB probe file contains all the needed tips. In PC-DMIS these AB tip angles will show an asterisk symbol ‘*’ to indicate that they are not yet calibrated.
- Calibrate the tips in one of the following ways:
  - Execute the measurement routine.
  - Click the **Measure** button from the **Probe Utilities** dialog box.

Step 3: Create the PC-DMIS Measurement Routine File and Import the DMIS File

- Create a new PC-DMIS measurement routine.
- Import the ChorusNT measurement routine, specifying the MYRPOBE.PRB probe file created in the previous step.
- Usually Chorus measurement routine contain SNSLCT/ S(label) or SNSLCT/ SA(label) statements. For self-calibrated wrists, you can explicitly specify roll and pitch angles in the SNSLCT label.

After PC-DMIS imports the DMIS file, the final PC-DMIS measurement routine will have at the beginning the `LOADPROBE/MYPROBE.PRB` command, and for each SNSLCT statement the corresponding `TIP` command will appear with the correct A and B angles.

Step 4: End of Automatic Import Phase

PC-DMIS completes the Import phase, showing you a report in the **Warning** window.

For example, you might see something like this:

```
L42: ---: DECL/CHAR,O_D_L[11]

L56: ---: DECL/
REAL,TX,TY,TZ,MISX,MISY,MISZ,CXT,CYT,CZT,VX,VY,VZ

L57: ---: DECL/REAL,INVCXT,INVCYT,INVCZT

L112: !MAJOR :MMEDIA/ON,'C:\myfile.exe'

L150: !MAJOR :MRRPCS/D(MRR),0,1,0

L213: ERROR: !UNITS

L253: ERROR: !SNSLCT/S(S_103)
```

These messages are explained here:

- **L##** - This represents the *line number* for that statement in the original DMIS routine (## represents the number). For example L42 would indicate that on line 42 of the DMIS routine, the DMIS statement is found.
- **---:** - This symbolizes that the statement was ignored, that is it was not converted into a corresponding PC-DMIS command because there is no equivalent PC-DMIS command. For example, you will find that PC-DMIS always ignores the DECL statement because PC-DMIS doesn't need explicit variable declaration.
- **!MAJOR** - This means that the specified DMIS statement (Major DMIS Word) is not supported.
- ERROR - This means the specified command was not correctly imported.

Step 5: Manual Verification of the PC-DMIS Measurement Routine

Be sure to manually verify that the translation into PC-DMIS functions properly. Look through the measurement routine in Command Mode. Errors appear in red text.

Try executing the measurement routine in Offline Mode then try using the CMM in Online Mode at low speed to verify that the measurement routine imported without problems.

Things to Watch for and Some Helpful Tips and Suggestions

ChorusNT and PC-DMIS are different measuring systems, and some intrinsic characteristics need to be carefully considered. The following list contains some tips and suggestions:

Automatic Measuring Cycle

The DMIS Standard states that in a MODE/AUTO section the features are measured with the automatic cycle, ignoring any eventually present PTMEAS.

Since ChorusNT supplies automatic cycle for POINT, CIRCLE, SPHERE, and SLOT (CPARLN), all these cases are translated into the corresponding AUTO feature. However, due to the intrinsic differences between Chorus and PC-DMIS, you should carefully verify their execution.

In SLOT (round slot) automatic cycle, ChorusNT takes the first two points on one of the straight sides. PC-DMIS however, takes the points on the rounded ends of the slot.

STAR TIP Probes

PC-DMIS recognizes the ChorusNT SNSDEF extension that defines a star probe. Be careful that the PC-DMIS setup for the Probe Head orientation corresponds to the ChorusNT SNSMNT specification.

CW43, CW43L, and IW42 Wrist Mounting conventions

ChorusNT and PC-DMIS conventions for the roll angle are different. While PC-DMIS can correctly adjust it during the import of a Chorus DMIS measurement routine, you must be careful to safely execute the imported measurement routine the first time.

Theoretical values in FEAT statements

PC-DMIS always uses both Theo and Actual values, which should be correct (not too much off a deviation from actual values). If the deviation is great, you may have problems, especially with features used in alignments. This is because PC-DMIS defines two matrices for each alignment:

- First, the CADTOPART matrix. Based on the theoretical values, this converts feature THEO values.
- Second, the MACHINETOPART matrix. Based on actual measured values, this converts the actual measured values in features.

Since ChorusNT doesn't have the CADTOPART matrix, it always uses the actual measured values. If you have the wrong nominal values, you only become aware of them if you execute an OUTPUT statement on that feature. To verify correctness, you should add them into the original DMIS routine OUTPUT statements for your features, especially for the alignment features.

FILNAM

FILNAM in ChorusNT specifies the name of the output file when using the `DISPLY/STOR` command. PC-DMIS recognizes it but this command must appear after the DISPLY in the DMIS routine. Consult your DMIS manual for more information.

VFORM

The ChorusNT vendor format statement is V(label)=VFORM/ALL which translates into the PC-DMIS `FORMAT/TEXT` command that has the following information:

- MEASURED
- NOMINAL
- DEVIATION
- UP_TOL
- LO_TOL
- CRIT/OOT

The final PC-DMIS `FORMAT` command will look like this:

```
FORMAT/
TEXT,OPTIONS,ID,HEADINGS,SYMBOLS,SD;MEAS,NOM,DEV,TOL,OUTTOL, ,
```

DEFGRF

DEFGRF statement is a Chorus extension that lets you define dialog boxes with buttons and input fields. PC-DMIS imports this command and creates a script using the BASIC language (Cypress Enable). Due to different screen resolutions, it’s possible that the resulting dialog box may need some adjustment. You can modify the Basic Script file using PC-DMIS's Basic Script Editor. For details, see the PC-DMIS Basic documentation.

FROM and GOHOME

ChorusNT's conventions move the head center not the tip's ball center. While the PC-DMIS import can make the necessary offsets, you should use a low feed the first time you execute the imported ChorusNT routine inside PC-DMIS.

MRRPCS

In ChorusNT this command defines the reference system to be used in a routine created with ChorusNT's MIRROR Utility. PC-DMIS does not support the import of Chorus mirrored routines. You should instead use PC-DMIS's more powerful and complete mirror utility on a .prg file created by importing the "left side" Chorus routine.

GAUGE

In ChorusNT this statement defines and measures a sphere gauge, specifying the name: G(label), the diameter, and the stem direction. It also optionally specifies the center position, the incidence angle, and the number of points used to measure it.

```
GAUGE/SPHERE,G(label), diam, i, j, k, [angle, x, y, z,
npoints]
```

Because of the different parameters, PC-DMIS translates these two commands: `AUTO/SPHERE` and `CALIBRATE ACTIVE`.

F(label)= AUTO/SPHERE

The `AUTO/SPHERE` command uses the same name, diameter, and direction as the GAUGE statement. If the center position was not specified and during execution you manually select the first point on the top of the pole, then init=1 and perm=1, otherwise init=0 and perm=0.

PC-DMIS can also adjust the orientation, angle_vec, and normal_vec values accordingly.

```
CALIBRATE ACTIVE TIP WITH FEAT_ID=F(label),
QUALTOOL_ID=label,MOVED=YES
```

CALIB

In ChorusNT this statement calibrates the specified probe using the specified sphere gauge, G(label), and angle:

```
CALIB/SENS, S(probe),G(label), [angle]
```

PC-DMIS translates this statement into these three commands:

```
TIP/ T1A..B..
```

The tip command will have the same AB angles as the S(probe)) statement.

```
F(label)=AUTO/SPHERE
CALIBRATE ACTIVE TIP WITH FEAT_ID=F(label),
QUALTOOL_ID=label, MOVED=NO
```

Importing a Datalog File

Datalog programs are text files containing feature information that is stored in a proprietary format used by Chrysler.

The **File | Import | Datalog** menu item imports the information stored in the Datalog text file into your measurement routine. This functions much like importing an Inspection Plan file. Also, the **Change Management** dialog box discussed in "Updating Inspection Plans" topic of the "Using Inspection Plans in PC-DMIS" section also supports the Datalog file type.

Importing DMIS Files

PC-DMIS can import generic DMIS files and DMIS routines created using extensions from specific vendor environments.

> Extensions may include non-standard commands, or added or modified parameters of standard DMIS commands.

To import a DMIS routine file, follow these steps:

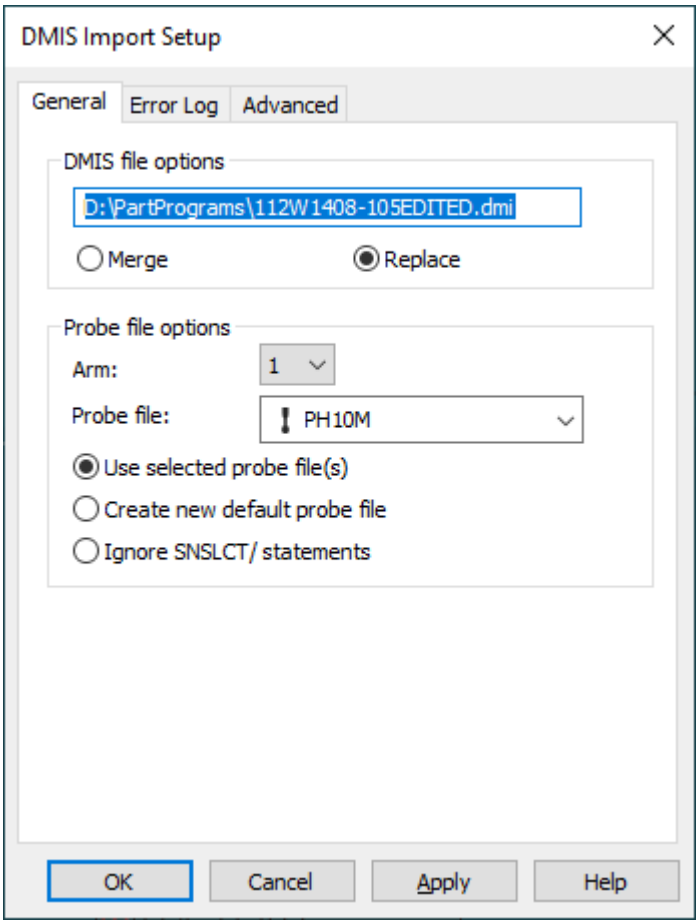
1. Select **File | Import | DMIS** to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder. If you select **Chorus DMIS Files** or **DMIS Files**, you can choose to import a .dmi, .dms, or .dmis file.

   > Chorus NT programs follow the DMIS 3.0 Standard with extensions and are better managed. In most cases, you should choose **Chorus DMIS Files** over normal **DMIS Files**. For notes on how to translate ChorusNT DMIS files into a PC-DMIS measurement routine, see "ChorusNT to PC-DMIS Translation Notes".

3. From the **Import** dialog box, navigate to the folder that contains the file to import. Then select the file.
4. Click **Import** to import the file into the Graphic Display window. A progress bar appears and indicates the progress.
5. PC-DMIS makes a first-pass translation of the input file. Once it completes this pass, the **DMIS Import Setup** dialog box appears. You can use this dialog box to select many options that are useful during the actual import phase.
6. Select the necessary options from the **DMIS Import Setup** dialog box. For information on how to use this dialog box, see the topics below.
7. Click **OK**. PC-DMIS finishes the importation process.

PC-DMIS inserts an `RMEAS/LEGACY` command into the imported measurement routine. You can modify the `DMISRmeasImport` entry to change the default value for RMEAS commands on future imported DMIS files. For information on this command, see "Setting Up Relative Measure (RMEAS)".

DMIS Import Setup - General tab



DMIS Import Setup Dialog Box - General Tab

The **General** tab of the **DMIS Import Setup** dialog box (**File | Import | DMIS**) allows you to control probe file translation for the imported DMIS measurement routine. The controls used on this tab are described here:

DMIS file options - This box specifies the DMIS file to import. It automatically contains a pathway to the file selected from the **Open** dialog box described in the DMIS import procedure above.

Merge and **Replace** - These options specify whether or not PC-DMIS replaces existing commands with the imported DMIS commands or merges the new commands with the existing commands.

- **Replace** - This option replaces the existing commands in the currently loaded measurement routine with new PC-DMIS commands during the import process.
- **Merge** - This option appends PC-DMIS commands created during the import process at the end of the currently loaded .PRG file.

Arm - This list specifies the arm to use from a multiple arm system.

Probe File - This list specifies an existing probe file.

Use selected probe file(s) - This option uses an existing probe file selected from the **Probe file** list.

1. From the **Arm** list, select the arm that will use the displayed probe file from the drop-down window.
2. Select the probe file from the list.
3. Click **Apply**. PC-DMIS adds any tips (sensors) referenced in the DMIS measurement routines that are not already in the probe file.

Create new default probe file - This option creates a new probe file based on the SNSDEF statements in the DMIS measurement routine file. PC-DMIS generates a new probe file and gives it the same name as the DMIS measurement routine filename and a ".prb" extension. The created probe configuration defaults to a PH9 wrist with a TP2 connection with a TIP2BY20MM tip. However, if no SNSDEF statements are detected in the DMIS measurement routine, a probe configuration will not be created.

Ignore SNSLCT/ statements - This option ignores SNSLCT statements in the DMIS measurement routine during the import process.

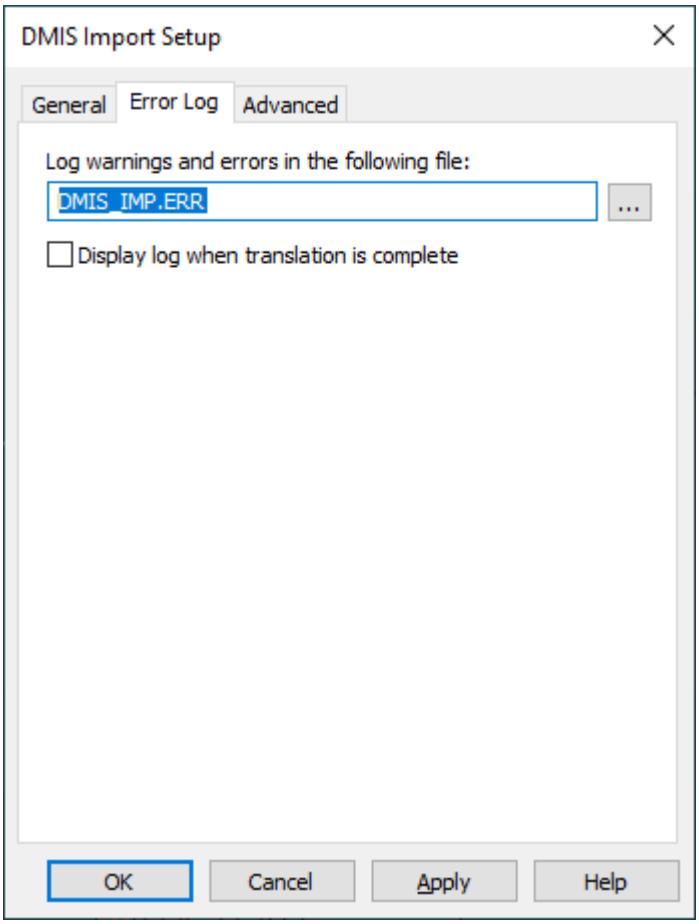
If the default probe configuration does not match the probe configuration of your machine, you should construct a probe file with the correct probe configuration using PC-DMIS *prior to* importing a DMIS measurement routine file. Then, select the created probe file from within this tab and click **Apply** and then **OK**. This allows the translator to select the most appropriate tip or tips during translation of the SNSDEF statements.

Related Topics:

DMIS Import Setup – Error Log tab

DMIS Import Setup – Advanced tab

DMIS Import Setup - Error Log tab



DMIS Import Setup Dialog Box - Error Log Tab

The **Error Log** tab of the **DMIS Import Setup** dialog box (**File | Import | DMIS**) causes PC-DMIS to store any warnings or errors at the end of the import process in a text file of your choice. To select a text file, click the Browse button.

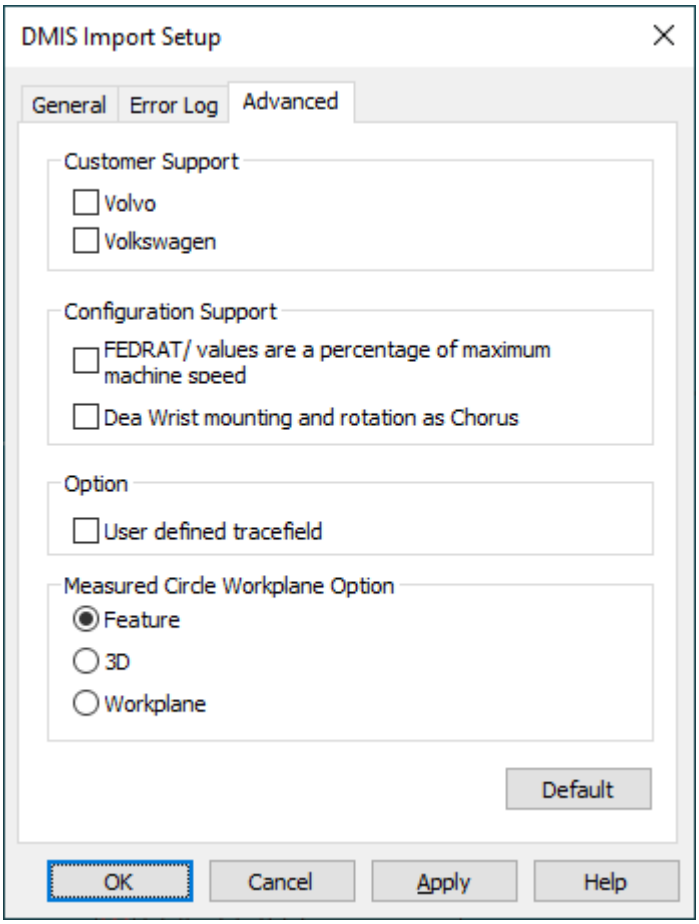
If you want PC-DMIS to automatically display this error log at the end of the import process, click the **Display log when translation is complete** check box. The text file opens in a **DMIS Import Results** dialog box.

Related Topics:

DMIS Import Setup – General tab

DMIS Import Setup – Advanced tab

DMIS Import Setup – Advanced tab



DMIS Import Setup Dialog Box – Advanced Tab

The **Advanced** tab of the **DMIS Import Setup** dialog box (**File | Import | DMIS**) allows you to import specific customer requests and configurations. These requests may include non-standard commands or added / modified parameters of standard DMIS commands. When you import Chorus DMIS files, PC-DMIS disables unnecessary options and enables needed ones.

For information on the controls on this tab, see the descriptions here:

Customer Support - This area contains specific check boxes for DMIS vendors. If your DMIS files are created using software from these companies, select the appropriate check box to enable extension support specific to those vendors. These include the following:

- Volvo
- Volkswagen

Configuration Support - This area contains check boxes that allow you to select different DMIS configurations. These include the following:

FEDRAT/ values are a percentage of maximum machine speed

DMIS commands don't automatically specify whether imported FEDRAT/ commands (commands that control the machine's speed) refer to maximum machine speed or maximum touch speed.

- *If you select this check box*, imported FEDRAT/ statements are a percentage of the *maximum machine speed*.
- *If you deselect this check box*, imported FEDRAT/ statements are a percentage of the *maximum touch speed*.

DEA Wrist mounting and rotation as Chorus

If you're using a DEA CMM equipped with an IW42, CW43, or CW43L wrist and you select this check box, PC-DMIS follows Chorus conventions and adjusts the B rotation angle during the import of the SNSLCT statement.

This option is automatically selected when you import Chorus DMIS files.

Option - This area contains a **User Defined Tracefield** check box. This check box determines whether certain imported DMIS commands are transferred into user-defined tracefields.

According to the DMIS standard, you can query the operator for the following information during measurement routine execution with these DMIS commands:

Info	DMIS Command
Part Name	PN(label) = 'text'
Part Serial Number	PS(label) = 'text'
Manufacturing Device	MD(label) = 'text'
DME Name	DI(label) = 'text'
Operator Name	OP(label) = 'text'

Suppose the DMIS file you are to import has these DMIS commands:

```
PN(label1) = PARTID/' part number'

PS(label2) = PARTSN/'2345'

MD(label3) = MFGDEV/'BRAVO1'

DI(label4) = DMEID/'PC-DMIS'

OP(label5) = OPERID/'My Name'
```

If you select the **User Defined Tracefield** check box, PC-DMIS supports the same functionality required by the DMIS standard and it imports DMIS commands into TRACEFIELD commands, like this:

```
TRACEFIELD/DISPLAY,LIMIT=15 ; PN(LABEL1) : part
number

TRACEFIELD/DISPLAY,LIMIT=15 ; PS(LABEL2) : 2345

TRACEFIELD/DISPLAY,LIMIT=15 ; MD(LABEL3) : BRAVO1

TRACEFIELD/DISPLAY,LIMIT=15 ; DI(LABEL4) : PC-DMIS

TRACEFIELD/DISPLAY,LIMIT=15 ; OP(LABEL5) : My Name
```

Then when you execute our measurement routine, PC-DMIS provides a small dialog box for each TRACEFIELD command, so that you can type this information.

If you do not select the **User Defined Tracefield** check box, by default, PC-DMIS inserts these DMIS commands into normal comment report commands, like this:

```
COMMENT/REPT,"PN(LABEL1)=' part number'"

COMMENT/REPT,"PS(LABEL2)='2345'"

COMMENT/REPT,"MD(LABEL3)='BRAVO1'"

COMMENT/REPT,"DI(LABEL4)='PC-DMIS'"

COMMENT/REPT,"OP(LABEL5)='My Name'"
```

Measured Circle Workplane Option - A measured circle is a 2D feature that is relative to the normal vector. This area contains options that define how PC-DMIS handles the normal vector for measured circles in DMIS files during a DMIS file import.

Feature - A plane feature defines the normal vector. The plane feature needs to be immediately above the circle feature in the DMIS file.

If your DMIS file doesn't have a plane feature, your circle feature will still import, but there will be an empty feature reference in the circle feature. In this case, after import, you can define a plane feature in the Edit window and then select it in the measured circle feature.

3D - PC-DMIS uses the feature's nominal vector to define the normal vector. This is the traditional method to define the normal vector.

Workplane - PC-DMIS uses the current workplane as the normal vector.

Default - This button sets your selection as the default to use when importing DMIS files.

Related Topics:

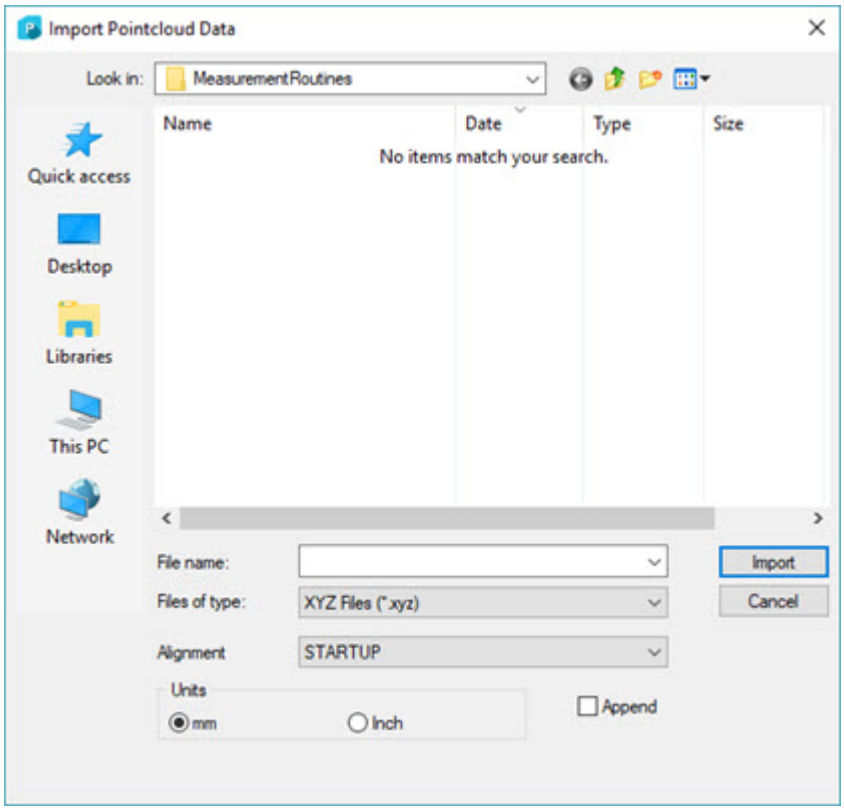
DMIS Import Setup – General tab

DMIS Import Setup – Error Log tab

Importing Pointcloud files

To import a Pointcloud file, perform the following steps:

1. Select **File | Import | Pointcloud** to open the **Import Pointcloud Data** dialog box.



Import Pointcloud Data Dialog Box

2. Choose the appropriate Pointcloud type from the **Files of Type** list at the bottom of the dialog box. PC-DMIS lists files ending with extensions according to what you selected.

- If you select the **XYZ Files** option, PC-DMIS lists files with a .xyz extension.
- If you select the **PSL Files** option, PC-DMIS lists files with a .psl (Polyworks) extension.
- If you select the **STL Files** option, PC-DMIS lists files with a .stl extension.
- If you select the **NSD Files** option, PC-DMIS lists files with a .nsd (3DReshaper) extension.

3. Mark the **Append** check box if you want to add the new COP data onto existing COP data.
4. Navigate to the appropriate directory and select the desired pointcloud file from the list.
5. Click **Import** to load the Pointcloud file into your measurement routine.

Importing from Planner (.plancad, .planxml, .xml)

You can import the Inspection Plan from Planner into PC-DMIS by importing a .planxml or .xml file exported from Planner. The .planxml or .xml file contains XML elements with the inspection plan information.

To import a .plancad, .planxml, or .xml file:

1. Select **File | Import | Inspection Plan** to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder. If you select **All Plan Files**, you can import a .plancad, .planxml, or .xml file.
   - If you import a .PlanCAD file, the software imports both the plan features and the embedded CAD model.
   - If PC-DMIS detects changes in the original .PlanCAD file that you're referencing, it asks if you want to use the Change Manager to handle those changes. For information on the Change Manager, see "Updating Inspection Plans with the Change Manager".

   Your file to import should have the same measurement units as the measurement routine that takes the import. If the files don't have the same measurement units, you can still import the file, but the measurement routine ends up with incorrect information. For example, if you have a circle with an X value of 4 inches in the file to import, the software imports it as 4 millimeters if the measurement routine that takes the import expects millimeters.

3. Click **Import** to import the inspection plan and create a measurement routine from it. For .plancad and .planxml files, once you click **Import**, the software opens the **Sensor Mapping** dialog box so that you can map features to sensors. For more information, see the "Mapping Features to Sensors" topic.

PC-DMIS uses entries to determine the settings for the imported features. You can use the Measurement Strategy Editor or the **Auto Feature** dialog box to modify these settings.

Importing a Legacy Planner Inspection Plan (.ip)

To import a Planner Inspection Plan as an .ip file, perform the following tasks first:

- Define Parameters and Rules
- Perform an optimization step on the imported plan
- Ensure that the probe won't collide with the part by inserting proper move commands

Once you have completed these tasks, you can then execute your imported inspection plan as a newly-created PC-DMIS measurement routine.

If you have problems using the PC-DMIS Planner commands inside PC-DMIS, ensure that there is an *InsPlan.dll* file located in the directory where you installed PC-DMIS.

Inspection Plan Default File (.ipd)

You must have an Inspection Plan Default file or .ipd to import a .ip file into PC-DMIS. The "Legacy Planner Parameters" topic provides more information on this file.

Once you've performed the above preparatory steps, you can import the legacy inspection plan and any rules by importing an inspection plan's .cad file that has the inspection plan embedded in it or by importing a previously-exported inspection plan.

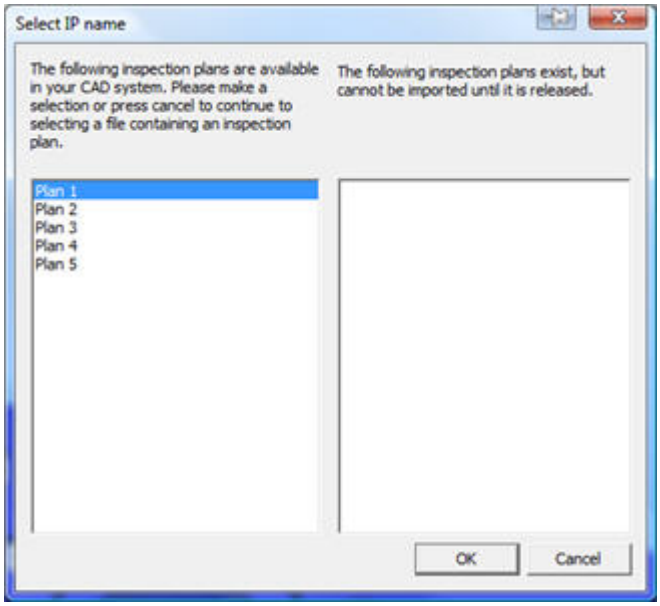
To import an embedded Inspection Plan from a CAD file:

1. Select **File | Import | CAD**. The **Open** dialog box appears.

When importing a CAD file using the **CAD by Reference** menu option, PC-DMIS updates the file accordingly if any changes were made to the IP CAD file by the legacy PC-DMIS Planner application.

2. Browse to and select the needed IP CAD file, and click **Import**. PC-DMIS imports the IP CAD file.

3. Select **File | Import | Inspection Plan**. The **Select IP name** dialog box appears.



Select IP name Dialog Box

4. Select the needed Inspection Plan, and click **OK** or click **Cancel** to browse to and select an exported Inspection Plan via the **Open** dialog box.

5. Follow any on-screen instructions.

    - Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Legacy PC-DMIS Planner Parameters and Rules" topic for more information.

    - If the insertion point is not at the end of the measurement routine, you are prompted to move the cursor to the end of the measurement routine before continuing.

    - If the CAD file comes from a CATIA v5 model, any comments associated with a feature in the eTool data of the CATIA model are imported as Feature Locator Text for the feature. Note that eTool is a proprietary format used by Chrysler. For information on Feature Locator Text, see the "Providing and Using Feature Locator Instructions" topic in the PC-DMIS CMM documentation.

    - Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See "Optimizing the Path" for more information.

    - Specify the options for "Inserting Clearance Moves Automatically" and click **OK** to complete this process or click **Cancel** to skip this step.

To import an Inspection Plan from an exported .ip text file:

1. Select **File | Import | Inspection Plan**. An **Import** dialog box appears.

2. Browse to the directory containing your file with an .ip filename extension.

3. Select the inspection plan, and click **Import**.

4. Follow any on-screen instructions.

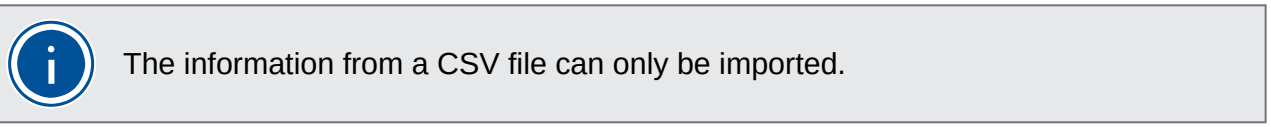
    - Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Legacy PC-DMIS Planner Parameters and Rules" topic for more information.
    - If the insertion point is not at the end of the measurement routine, you are prompted to move the cursor to the end of the measurement routine before continuing.
    - Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See "Optimizing the Path" for more information.
    - Specify the options for "Inserting Clearance Moves Automatically" and click **OK** to complete this process or click **Cancel** to skip this step.

Once you've imported your inspection plan, the imported commands are preceded by a start comment that provides the following basic information for the imported inspection plan:

- Date and time
- The .ip file name with path
- The .ipd file name and path

An end comment indicates the end of the imported inspection plan.

Importing a CSV File



Select **File | Import | CSV** to import information from a CSV file.

Each row in the CSV file represents a feature and a feature command that PC-DMIS creates during the import. If tolerance values are specified, a location dimension command is also added.

The first row of the CSV file contains the format designators. This designator row indicates the data for each column. There are predefined column headers. Columns that do not include a format designator or that have an unrecognized column header are ignored. Columns can be in any order. If the "Type" column is missing or if there is no feature type specified for a row of data, PC-DMIS treats this row of information as a "Point" feature.

If a row has only an asterisk in the first column with no other information in the row, this signals that the next row is a format designator row. This lets you switch formats inside a single CSV file.

File Format...

A CSV file is a **C**omma **S**eparated **V**alues file created by many software packages, including Microsoft Excel. Here is an example of an Excel table used to create a CSV file:

TYPE	NAME	X	Y	Z	X2	y2	z2	D	A	L
Point	PNT1	67.98	89.88	13.99						
Line	LIN1	51.75	0	-13.1	62.25	0	-13.1			
Plane	PLN1	100.25	0	-25.6						
Circle	CIR1	124	50	0				60.5		
Ellipse	ELL1	179.9	10	0						14.125
SlotR	SLTR1	191.05	72.998	0				7.02		33.045
SlotS	SLTS1	69.7	23.35	0				11.041		12.018
SlotN	SLTN1	76.5	6	0				6		12
Polygon	POL1	26.35	49.5	15				22		
Cylinder	CYL1	124	50	0				60.5		
Cone	CON1	69	90	14					30	14
Sphere	SPH1	188.5	45.65	0				12.75		

Example of a spreadsheet table used to create a CSV file (continued below).

D	A	L	W	i	j	k	I2	J2	K2	+ TOL	- TOL
				0	0	1				0.05	0.05
				1	0	0				0.05	0.05
				0.707	0.707	0				0.05	0.05
60.5				0	0.707	0.707				0.05	0.05
		14.125	7.23	0.707	0	0.707	1	0	0	0.05	0.05
7.02		33.045	12.8	0.707	0	0.707	1	0	0	0.05	0.05
11.041		12.018	4.2	0.707	0	0.707	1	0	0	0.05	0.05
6		12	3.5	0.707	0	0.707	1	0	0	0.05	0.05
22				0.707	0	0.707	1	0	0	0.05	0.05
60.5				0.707	0	0.707				0.05	0.05
	30	14		0.707	0	0.707				0.05	0.05
12.75				0	0	1				0.05	0.05

Example of a spreadsheet table used to create a CSV file (continued).

Download the above CSV example file here: CSV_Example_File.csv

Another example of a spreadsheet table used to create a CSV file is shown below:

TYPE	NAME	X	Y	Z	+TOL	-TOL
Point	X48a001	68	90	14	0.05	0.05
Point	X48a002	74	85	12	0.05	
Point	X48a006	76	84	11		
Point	X48a011	79	83.5	10.75	0.035	0.035
Point	X48a021	85	83.25	10.67	0.035	0.035
Point	X48a022	88	2.375	10.5	0.035	0.035
Point	X48a029	97	82.125	10.375	0.05	0.05

Example of a spreadsheet table used to create a CSV file.

Row two defines a point feature with both +TOL and -TOL. PC-DMIS adds the point feature command and the location dimension command based on this row.

The first row (TYPE, NAME, X, Y, Z, +TOL, -TOL) is the format designator row.

Neither +TOL nor -TOL is specified in the point feature in row four. Only a feature command is added. A location dimension command is not added.

Format Designators

The very first line (or row if viewed in Excel) in a CSV, is the format designator line. It determines what type of data is expected in the subsequent lines. The following is a list of valid format designators allowed when a CSV file is imported.

TYPE – This is the feature type. The legal entries for this column are: **Point**, **Line**, **Plane**, **Circle**, **Ellipse**, **SlotR**, **SlotS**, **SlotN**, **Polygon**, **Cylinder**, **Cone** and **Sphere**.

NAME – Feature name

X : Feature’s nominal X value

Y : Feature’s nominal Y value

Z : Feature’s nominal Z value

X2 : Feature’s second nominal X value

Y2 : Feature’s second nominal Y value

Z2 : Feature’s second nominal Z value

D : Feature’s nominal diameter

A : Feature’s nominal angle in degrees

L : Feature’s nominal length

W : Feature’s nominal width

i : Feature’s nominal i vector

j : Feature’s nominal j vector

k : Feature’s nominal k vector

i2 : Feature’s second nominal i vector

j2 : Feature’s second nominal j vector

k2 : Feature’s second nominal k vector

+TOL : Feature’s plus tolerance

-TOL : Feature’s minus tolerance

Asterisk (*) Character for Multiple Format Designators

An asterisk character (*) in the very first position of a comma delimited row indicates that this is the end of the data for the current format designator. The asterisk character also indicates that the next row is a new format designator row. This lets you have features of multiple types in the CSV file with simpler format designator rows:

	A	B	C	D	E	F	G
1	TYPE	NAME	X	Y	Z	+TOL	-TOL
2	Point	PNT1	3	5	10	0.05	0.05
3	Point	PNT2	6	5	10	0.05	0.05
4	*						
5	TYPE	NAME	X	Y	Z	D	
6	Circle	CIR1	3	50	10	25	
7	Circle	CIR2	3	100	10	25	
8							

*The * in row 4 instructs PC-DMIS to use the new format designator in row 5 for the two circle features.*

File Import Sequence

The following describes the sequence of events that occur when you import a CSV file:

1. Each row of data is read into PC-DMIS from the CSV file.
2. The information is matched with the previously defined column headers to determine the meaning of each item of information.
3. Based on the feature type, PC-DMIS uses the information to create a feature command.
4. If one or both tolerance values are specified, PC-DMIS adds a location dimension command.

Importing XYZ Feature Data into the Measurement Routine

An XYZ file is a simple text file with a .xyz extension. It can contain feature information in the form of xyz coordinates with or without a vector. You can use any standard text editor to view the values in an .xyz file.

To import an XYZ file into the Edit window, follow these steps:

1. Select **File | Import | XYZ into Measurement Routine** to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder. If you select **XYZ Files**, you can choose to import a .xyz file.
3. From the **Import** dialog box, navigate to the folder that contains the file to import. Then select the file.
4. Click **Import** to import the file. PC-DMIS interprets the XYZ data and imports it into the Edit window as separate feature commands.

For information on how the software interprets the XYZ data in the file, see "Importing an XYZ File as CAD Data".

Importing a Direct CAD File

Direct CAD Interfaces (DCI) allow you to connect directly to your CAD software. As the CAD file is updated in your CAD software, the corresponding changes are reflected in PC-DMIS.

Instead of translating CAD data into the internal CAD format of PC-DMIS, the Direct CAD file types are accessed directly. That is, the CAD data of these file types remains in its native format after being imported. Routines native to the CAD type are used for all calculations done on the CAD data. Accessing the CAD data directly eliminates problems that can occur because of translation inaccuracies or limitations.

PC-DMIS supports the use of the following DCIs:

- CATIA 5
- Creo (previously Pro/Engineer)
- Solidworks
- NX (previously Unigraphics)

When you select the **NX** option, the NX file name appears in the **Layers to Import** dialog box.

For more information on how to import Direct CAD files, see the "Direct CAD Interfaces" documentation.

Tessellating a Part without a CAD License

You can tessellate the image of a Direct CAD Interface part even if you do not have its CAD license currently available. The method for accessing this functionality varies by the Direct CAD Interface. In general, however, you need to access your Direct CAD Interface program. PC-DMIS will then display the part, but some functionality will not be available. For example, you cannot create a measured vector point on the part.

Importing a MeasureMax Project

PC-DMIS can import a MeasureMax project, converting it to work in a PC-DMIS measurement routine.

To import a MeasureMax project, follow these steps:

1. Select the **File | Import | MeasureMax Project** to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder. If you select **MeasureMax Files**, you can choose to import a .vbp file.
3. Click **Import**. PC-DMIS automatically converts the project and inserts the PC-DMIS commands at the end of your measurement routine.
4. Follow any on-screen prompts to import the MeasureMax project into PC-DMIS.

Importing an Inspection Plan File

Inspection plans are created using the Planner application. Planner shares many similarities to PC-DMIS. Instead of an all-inclusive metrology application, Planner focuses on taking a part model and creating inspection plans of specified features. Inspection plans determine which features of the geometry and GD&T/dimension to use in the inspection process.

PC-DMIS 2019 R1 was the last version that included Planner.

A Planner user can then export these inspection plans from Planner as a .plancad, a .planxml, or an .xml file. These files contain both the underlying CAD model and the inspection plan data serialized together into a single file.

The **File | Import | Inspection Plan** menu item discussed here lets you then import the inspection plan file into PC-DMIS. The import process supports the old legacy PC-DMIS Planner .ip file or the newer .plancad, .planxml, or .xml file types.

- Your file to import should have the same measurement units as the measurement routine that takes the import. If the files don't have the same measurement units, you can still import the file, but the measurement routine ends up with incorrect information. For example, if you have a circle with an X value of 4 inches in the file to import, the software imports it as 4 millimeters if the measurement routine that takes the import expects millimeters.
- As part of the import, PC-DMIS runs an optimizing routine, which automatically adds measurement and motion commands in an optimal order for the imported features.

For additional information on importing inspection plans, see "Importing a Legacy Planner Inspection Plan" and "Importing from Planner" in the "Using Inspection Plans in PC-DMIS" chapter.

Exporting CAD Data or Feature Data

Similar to importing data, PC-DMIS can also export CAD data or feature data from your measurement routine into one of these file formats:

Alignment:

- PolyWorks Text
- GDS (.gds)

Part model:

- AS3 (.as3)
- CAD Mesh (.cad or .draw)
- PDF 3D (.pdf)
- STL (.stl)
- Wavefront (.obj)
- STEP (.stp or .step)
- XAML (.xamlsolid or .xamlwireframe)

Pointcloud:

- XYZ (.xyz) - Exports the pointcloud to an XYZ file type.
- IGES (.igs) - Exports the pointcloud to an IGES file type.
- PSL (.psl)

Other file types:

- DES (.des)
- DXF or DWG (.dxf or .dxg)
- Generic text (.txt)
- IGES (.igs) - Exports the PC-DMIS measurement routine to an IGES file type.
- STEP AP203 (.stp or .step)
- STEP AP214 (.stp or .step)
- VDAFS (.vda)
- XYZ (.xyz) - Exports the PC-DMIS measurement routine to an XYZ file type.
- BASIC (.bas)
- Datalog (.txt)
- DMIS (.dmi or .dms)
- Microsoft Excel (.xls)
- Inspection Plan (.ip)
- XML (.xml)

When you export your measurement routine as CAD data, PC-DMIS does not export previously imported CAD part models. Instead, PC-DMIS generates a CAD file from the feature commands in your measurement routine.

To export data from the current measurement routine:

1. Select the **File | Export** menu option and the export type (or select an appropriate submenu and then the export file type) to open an **Export** dialog box.
2. From the file type list, select the desired file type. PC-DMIS displays the available files for the indicated type.

3. In the **File Name** box, type the name you want to give the exported file, or select an existing file from the list. If you select an existing file, PC-DMIS replaces the current data in that file with the information generated by the **Export** operation.

4. Click **Export** to close the **Export** dialog box. (Depending on your data type, PC-DMIS may display another dialog box that asks you for additional input. For more information, see the topics below.) PC-DMIS generates the exported data file in the selected directory and gives it the extension from the file type list.

Exporting to a PolyWorks Text File

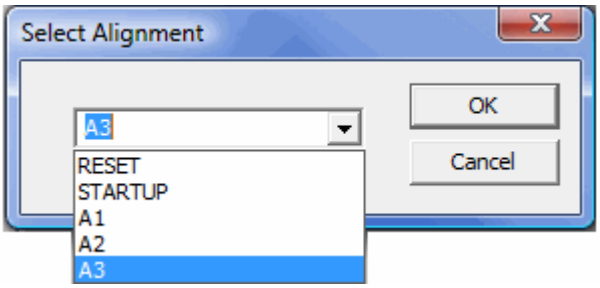
Using the **Export | Alignment | PolyWorks Text** menu item, PC-DMIS exports an alignment from the measurement routine into a PolyWorks text file. This allows you to use the exported PC-DMIS alignment in software applications that support the exported format (such as ScanWorks™) using the same coordinate reference frame as PC-DMIS.

Exporting to a Gds File

Using the **Export | Alignment | Gds** menu item, PC-DMIS can export an alignment from the measurement routine into a .gds file format. The .gds format is the Romer France file format for alignments. Romer France's portable arms can then use this small alignment file. It is also used to interface with other software packages, such as 3DReshaper.

By default, when you export to a .gds file, PC-DMIS exports the most recent alignment of the active measurement routine.

To export a specific alignment, from the **Setup Options** dialog box, first select the **Select Alignment to Export** check box. Then, when you export to a .gds file, the **Select Alignment** dialog box opens so that you can select an alignment.



Select Alignment Dialog Box

For more information on the **Select Alignment to Export** check box, see the "Select Alignment to Export" topic in the "Setting Your Preferences" chapter.

The .gds file name defaults to index.gds. PC-DMIS saves the file to the armdata folder of the G-Scan software. If you don't have the ROMSOFT software installed, a common **Export** dialog box opens so that you can select where to export the .gds file.

Exporting to an AS3 File

You can easily export your CAD model to DataView's .AS3 file format and then load this information into DataView to view your CAD model.

To export to the AS3 DataView file format:

1. Select **File | Export | Part Model | Wavefront** and then choose the directory where you want to export the file.
2. Type a file name.
3. Click **Export**.
4. PC-DMIS creates the .OBJ file at the specified location.

Exporting a CAD Mesh File

With the **File | Export | Part Model | CAD Mesh** menu option, you can export your CAD model as a lightweight mesh (as a .cad or .draw file). This export saves the model's surfaces as triangle meshes instead of the mathematical surface definitions. This essentially strips the model to its bare essentials that you can then use for visualization or collision detection. For a size comparison, after export a 2.9 MB .iges model file exported as a CAD mesh file takes up only 458 KB of space.

You may also find this menu option a useful solution to create a lightweight model for things like quick fixtures and other components.

A CAD mesh file is not an alternative for a true CAD model file. For example, with a mesh file, you lose the ability for PC-DMIS to use QuickFeatures or to do one-click Auto feature selection.

Exporting to a PDF 3D File

You can export your CAD model to a PDF 3D file format. This type of format stores your CAD model in a .pdf file. You can manipulate the CAD model in the .pdf file.

To export to a PDF 3D file:

1. Select **File | Export | Part Model | PDF 3D**, and then choose the directory where you want to export the file.
2. Type a file name.
3. Click **Export**.
4. PC-DMIS creates the .pdf file at the specified location.

For more information, see "Working with PDF 3D Files" in the "Reporting Measurement Results" chapter.

Exporting to an STL (Stereolithography) File

You can easily export your CAD model to DataView's .STL file format and then load this information into DataView to view your CAD model.

To export to DataView:

1. Select **File | Export | Part Model | STL** and then choose the directory where you want to export the file.
2. Type a file name.
3. Click **Export**.
4. PC-DMIS creates the .STL file at the specified location.

Exporting to a Wavefront Object File

You can export your CAD model to the Wavefront .obj file format and then load this information into applications that support .obj files to view your CAD model. The .obj file is an open 3D model format used by many 3D graphics applications.

To export to a Wavefront .obj file,

1. Select **File | Export | Part Model | Wavefront**, and then choose the directory where you want to export the file.
2. Type a file name.
3. Click **Export**.
4. PC-DMIS creates the .obj file at the specified location.

Exporting to a STEP File

You can export your CAD model as a STEP file with the application Protocol 214. This type of format stores your CAD model as a .stp file. You can then import the file back into PC-DMIS or use it with other CAD systems.

To export your CAD model to a STEP file, you must use the STEP AP214 option:

1. Select **File | Export | Part Model | STEP | STEP AP214** and then choose the directory where you want to export the file.
2. Type a file name.

3. Click **Export**.
4. PC-DMIS creates the .stp file at the specified location.

When you export your CAD model as a STEP file, PC-DMIS includes only the original part geometry. It does not modify or transform the internal CAD geometry based on any applied transformations.

Exporting to an XAML Model File

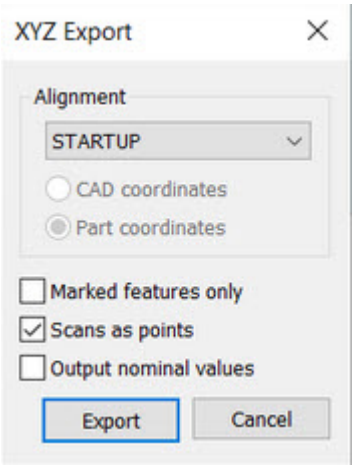
Using the **File | Export | Part Model | XAML** sub menu, you can choose to export your part model as one of these file XMAL model types:

- **Solid** - (*.xmalsolid)
- **Wireframe** - (*.xamlwireframe)

XAML is typically used as a markup language to define user interfaces. PC-DMIS, however, uses an extension that exports a subset of XAML used for specifying CAD data. This results in a 3D graphical display comparable to the PC-DMIS CAD format but weighing in at about half the size of the PC-DMIS CAD format. This light-weight graphics file is used in DataPage+.

Exporting to an XYZ File

When you click the **Export** button during the XYZ export process (**File | Export | XYZ**), PC-DMIS displays the **XYZ Export** dialog box.



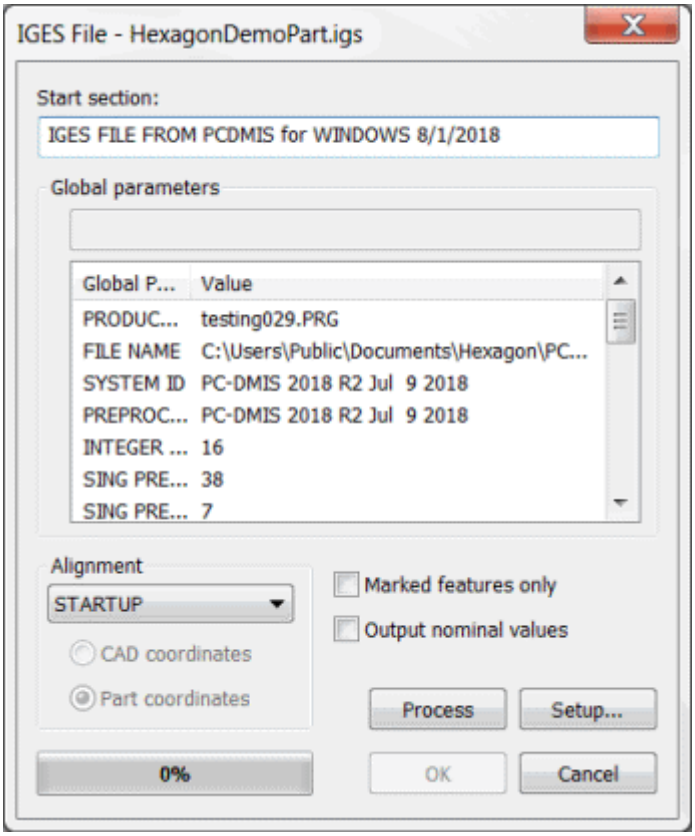
XYZ Export Dialog Box

The **XYZ Export** dialog box allows you to select options for your XYZ export. This exports your measurement routine as an .xyz file. This file type is readable inside standard text editors. For information on the .xyz file type, see the "Importing an XYZ File as CAD Data" for a discussion of this file's characteristics.

This dialog box has the exact same options as the **STEP Export** dialog box. For information, see "Exporting a STEP File".

Exporting an IGES File

When you click the **Export** button during the IGES export process (**File | Export | IGES**), PC-DMIS displays the **IGES File** dialog box.



IGES File Dialog Box

The **IGES File** dialog box displays information about your selected IGES export in the **Start section** and **Global parameters** areas. It also allows you to select additional options for your IGES export.

To export an IGES file, do the following:

1. Select an alignment in the **Alignment** list.
2. You can export an IGES CAD model with measured features.
    - If the features are not measured with an alignment, PC-DMIS exports them according to the selected option (**CAD coordinates** or **Part coordinates**).
    - If the features are measured with more than one alignment, the **CAD coordinates** and **Part coordinates** options become available. Select the desired option.

3. To alter the display of the exported image, click the **Setup** button and then complete the **IGES Write Setup** dialog box. For help, see "Altering the Display of Exported IGES CAD".

4. To export only the marked features, select the **Marked features only** check box.

5. To cause PC-DMIS to create the export file from the nominal values in your measurement routine, select the **Output nominal values** check box. If you clear this check box, PC-DMIS exports the measured feature results.

6. Click the **Process** button to output the IGES data from the selected measurement routine. PC-DMIS indicates when the file is 100% processed.

7. Click the **OK** button to export the IGES file.

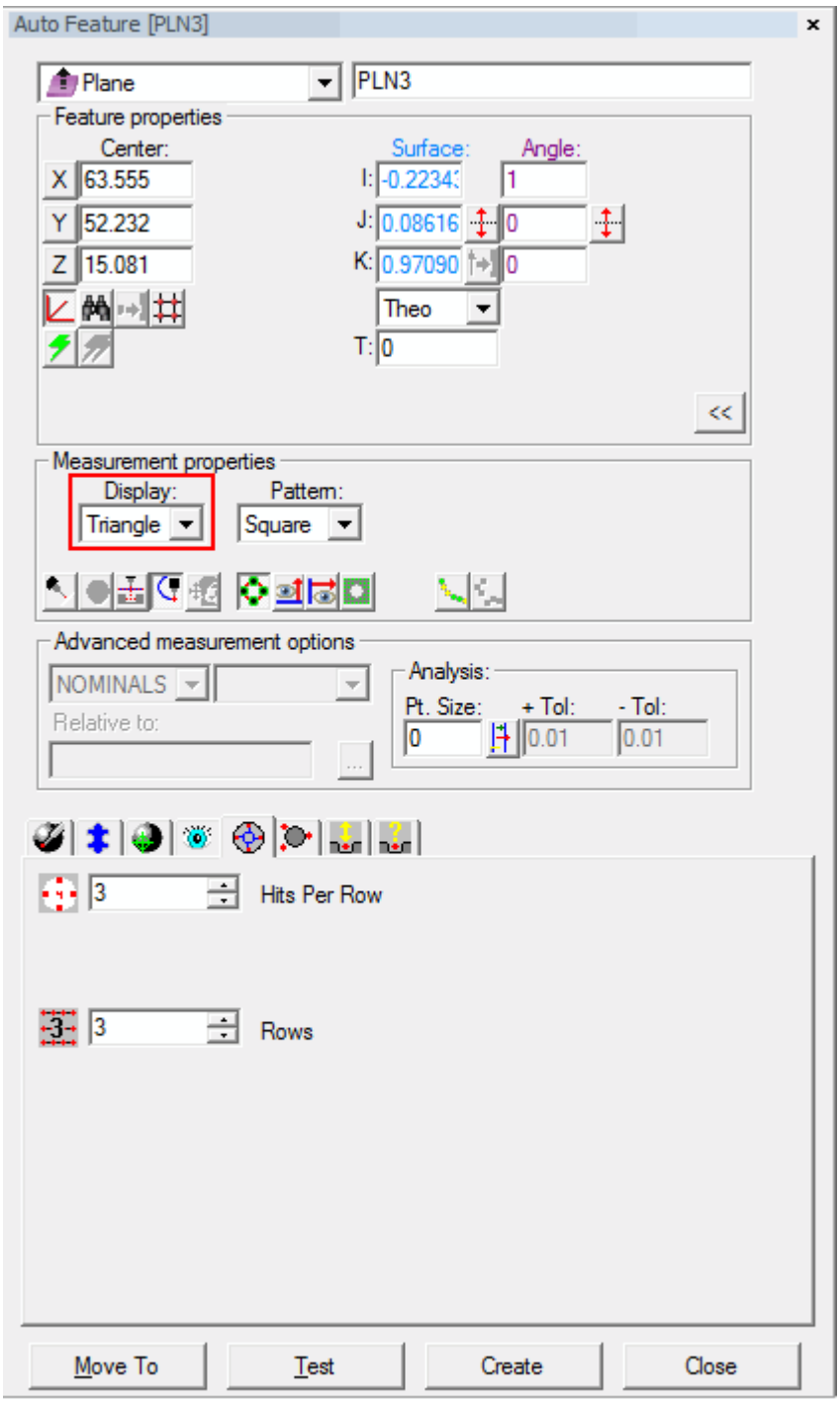
Exporting Plane Features in IGES

Plane features are exported in IGES the way they appear in the Graphic Display window. If the triangle representation of the plane is selected from the **Display** area of the dialog box, it is exported as a triangle bounded plane. If the outline is selected, the plane is exported as a convex polygon.

For an example of these display types, see "Using the Display Area" under the "Constructing a Plane Feature" topic.

The following sub-topics show the locations of the **Display** areas in each of the plane feature dialog box types:

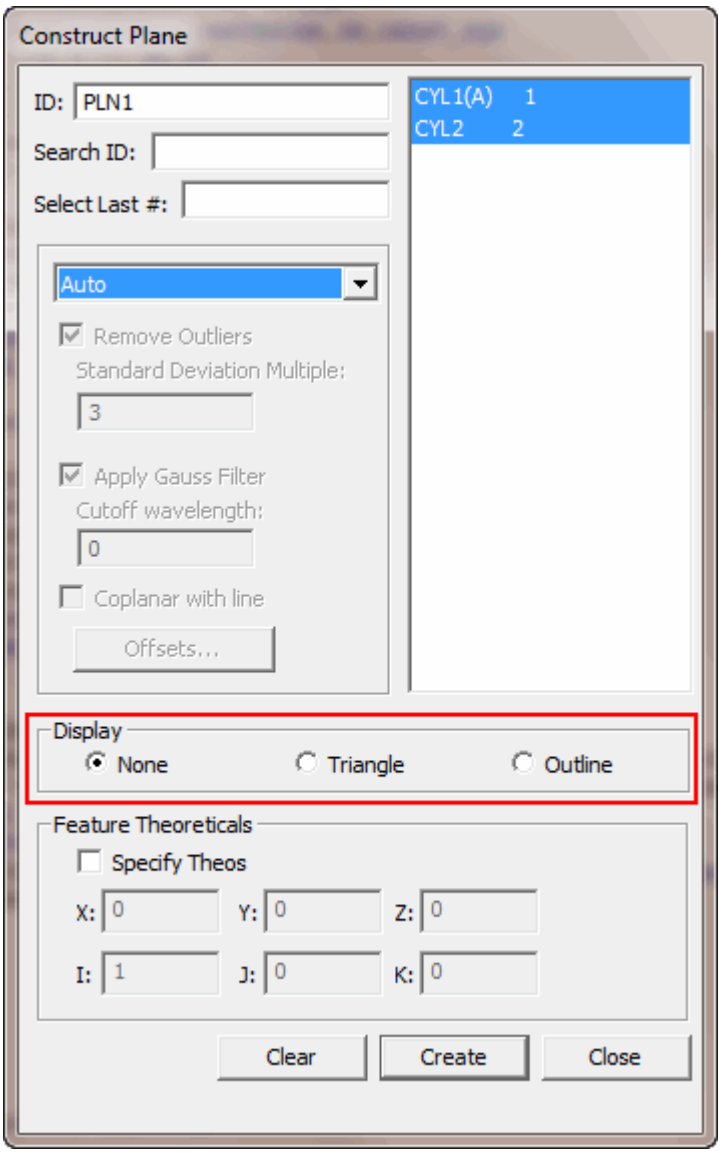
Auto Plane Feature Dialog Box



Auto Plane Feature dialog box emphasizing the Display section

For information, see the "Creating Auto Features" chapter.

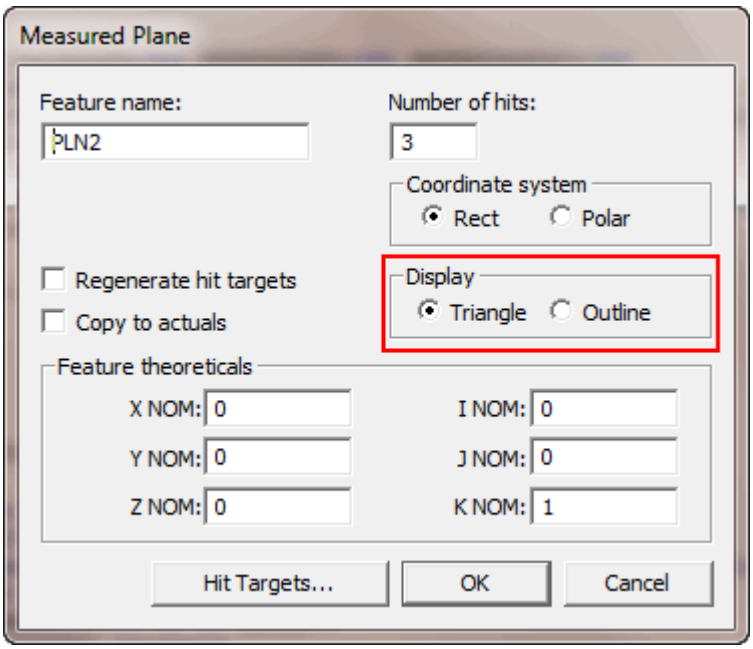
Construct Plane Feature Dialog Box



Construct Plane dialog emphasizing the Display section

For information, see the "Constructing New Features from Existing Features" chapter.

Measured Plane Feature Dialog Box



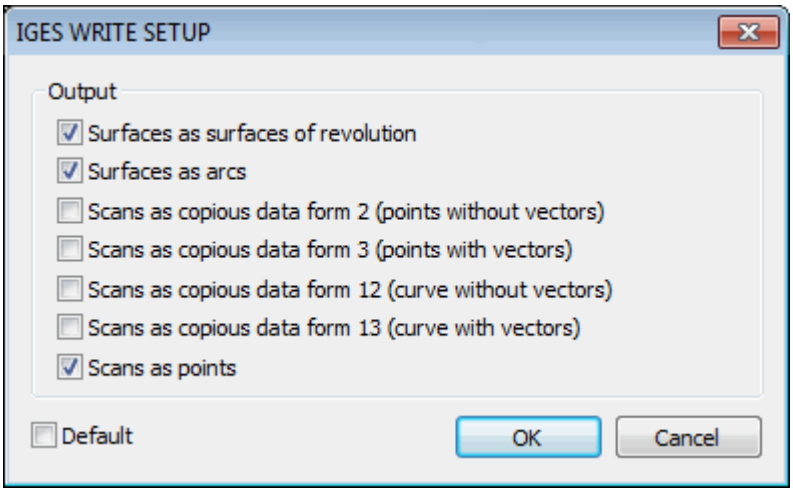
Measured Plane Feature dialog box emphasizing the Display section

For information, see the "Creating Measured Features" chapter.

The display type can also be changed in the Edit window in the plane feature's settings.

Altering the Display of Exported IGES CAD

The **IGES Write Setup** dialog box allows you to alter how the IGES/DMIS image displays in the exported file.



IGES Write Setup Dialog box

To access this dialog box and alter how the IGES/DMIS image displays, do the following:

1. Begin to export your CAD file (see "Exporting an IGES File").
2. When the **IGES File** dialog box appears, click the **Setup** button.

3. Select the desired check boxes in the **IGES Write Setup** dialog box.
4. Click the **OK** button.

PC-DMIS draws the selected CAD image with the requested features.

Understanding the Available Output Options

The **IGES Write Setup** dialog box, accessible from the **Setup** button in the **IGES File** dialog box (**File | Export | IGES**), provides these options that control how PC-DMIS exports IGES entities:

Surfaces as surfaces of revolution check box - Select this check box to export cones, cylinders, and spheres as surfaces of revolution entities.

Surfaces as arcs check box - Select this check box to export cones, cylinders, and spheres as arcs.

- PC-DMIS exports a cone as a circle at its base and a point at its vertex.
- PC-DMIS exports a cylinder as two circles: one at the top and one at the bottom.
- PC-DMIS exports a sphere as a circle about its equator.

To export cones, cylinders, and spheres as both surfaces of revolution and arcs, select both the **Surfaces as surfaces of revolution** and **Surfaces as arcs** check boxes. If you clear both check boxes, PC-DMIS doesn't export cones, cylinders, or spheres.

Scans as copious data form 2 (points without vectors) check box - Select this check box to export scans as copious data IGES entities in form 2. This format includes point data without vectors.

Scans as copious data form 3 (points with vectors) check box - Select this check box to export scans as copious data IGES entities in form 3. This format includes both point and vector data. The various forms control the format of the copious data entities.

Scans as copious data form 12 (curve without vectors) check box - Select this check box to export scans as copious data IGES entities in form 12. This format represents the data as a piecewise-linear curve without vectors. The various forms control the format of the copious data entity.

Scans as copious data form 13 (curve with vectors) check box - Select this check box to export scans as copious data IGES entities in form 13. This format represents the data as a *piecewise-linear curve with vectors*. The various forms control the format of the copious data entity.

Scans as points check box - Select this check box to export scans as multiple point entities.

To export scans as both copious data and points, select both the **Scans as copious data** check box (such as form 2, 3, 12, or 13) and the **Scans as points** check boxes. If you clear both the check boxes, PC-DMIS doesn't export scans.

Exporting a Pointcloud PSL File

You can easily export your CAD model to a Pointcloud .PSL file format and then load this information into DataView to view your CAD model.

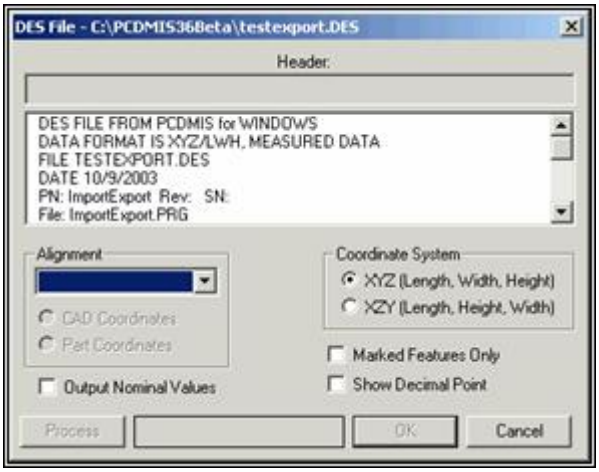
To export to DataView,

1. Select **File | Export | Part Model | PSL** and then choose the directory where you want to export the file.
2. Type a file name.
3. Click **Export**.
4. PC-DMIS creates the .PSL file at the specified location.

Exporting a DES File

You can export your measurement routine as a DES file. A DES file is a proprietary GM format used by GM's internal applications.

To export this file, select **File | Export | DES**. Choose where you want to place the file, and click **Export**. PC-DMIS displays the **DES File** dialog box.



DES File Dialog Box

The **DES File** dialog box displays information on your selected DES export in the **Header** area; it also allows you to select additional options for your DES export.

1. Select an alignment from the **Alignment** list.
2. Select a coordinate system from the **Coordinate System** area.

3. If desired, select the **Marked Features Only** check box to export only the marked features.

4. If desired, select the **Show Decimal Point** check box to show the decimal point in the exported data.

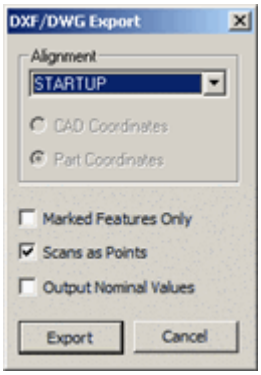
5. If desired, select the **Output Nominal Values** check box to cause PC-DMIS to create the export file from the nominal values in your measurement routine. If you don't select this check box, PC-DMIS exports the measured feature results.

6. Click on the **Process** button in the **DES File** dialog box to output the DES data from the selected measurement routine. PC-DMIS indicates when the file is 100% processed.

7. Click the **OK** button to finish exporting the DES file.

Exporting a DXF or DWG File

When you click the **Export** button during the DXF or DWG export process (**File | Export DFX** or **File | Export | DWG**), PC-DMIS displays a **DXF/DWG Export** dialog box.



DXF/DWG Export Dialog Box

The **DXF/DWG Export** dialog box allows you to select additional options for the export of these file types.

1. Select an alignment from the **Alignment** list.

2. If desired, select the **Marked Features Only** check box to only export the marked features.

3. If desired, select the **Scans as Points** check box to cause scan data to get converted to its scanned points.

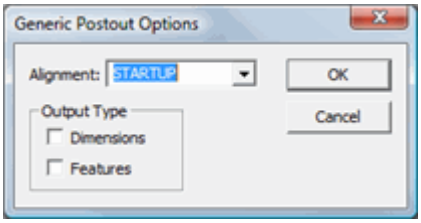
4. If desired, select the **Output Nominal Values** check box to cause PC-DMIS to create the export file from the nominal values in your measurement routine. If you don't select this check box, PC-DMIS exports the measured feature results.

5. Click the **Export** button to finish exporting the DXF or DWG file.

Exporting to a Generic Text File

Using the **File | Export | Generic** menu option, PC-DMIS can export your measurement routine's feature and dimension data to a simple text file as comma separated values. You can then import this data into other software applications, such as Microsoft Excel, as needed.

After choosing where to export the file and clicking the **Export** button, PC-DMIS displays a dialog box asking for you to choose the alignment to use when exporting the data and the type of data to export:



Generic Postout Options Dialog Box

Alignment is a drop-down list of alignment options you can choose from to apply to the data when exported.

Output Type area is where you select the type of data to export, either Dimensions, Features, or both.

An Example Exported Circle Feature:

```
CIR1, 93.486348, 19.488589, -1.269350, 0.000000, 0.000000,
1.000000, 14.997670

CIR1 HITS, 85.984616, 19.473057, -1.279984, 0.999998,
0.002070, 0.000000

CIR1 HITS, 93.479917, 11.992677, -1.287909, 0.000858,
1.000000, 0.000000

CIR1 HITS, 100.988033, 19.458504, -1.285809, -0.999992,
0.004010, 0.000000

CIR1 HITS, 93.486169, 26.984522, -1.265512, 0.000024,
-1.000000, 0.000000
```

The format for the first line of the feature is:
[Feature Label], [X], [Y], [Z], [I], [J], [K], [D]

The format for the hits lines, lines 2 - 5, is:
[Hits Label], [X], [Y], [Z], [I], [J], [K]

Only Auto feature hits data exposed in the Edit window can be exported to a generic text file.

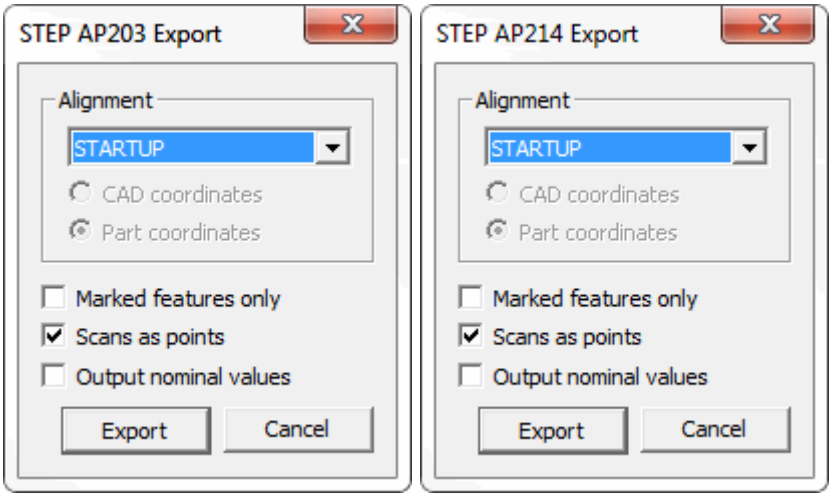
An Example Exported Location Dimension:

```
LOC1 AX:X, 93.485000, 0.010000, 0.010000, 93.486348,
0.001348, 0.000000

LOC1 AX:Y, 19.500000, 0.010000, 0.010000, 19.488589,
-0.011411, 0.001411

LOC1 AX:D, 15.000000, 0.010000, 0.010000, 14.997670,
-0.002330, 0.000000
```

The format for the lines is:
[Dim Label]AX:[Axis Indicator], [Nominal], [+ Tol], [- Tol], [Measured], [Deviation], [Out of Tol]

Exporting a STEP File

Exporting a STEP file is done by clicking **File | Export | STEP** from the main menu and, from the pull out list, choose either the **AP203** or **AP214** option. The corresponding **STEP Export** dialog box opens, as shown below.



STEP AP203 and AP214 Export Dialog Boxes

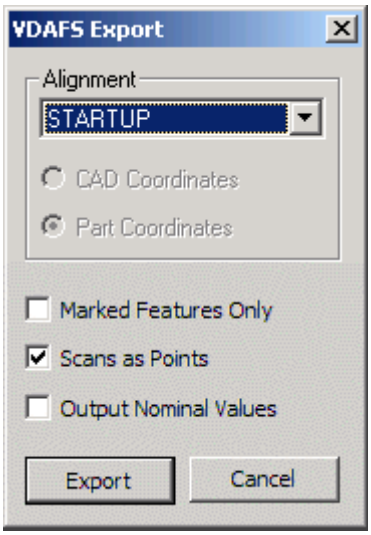
The **STEP Export** dialog box allows you to select additional options for your STEP export.

1. Select an alignment from the list in the **Alignment** area.
2. Select a coordinate system from the **Alignment** area.

3. If desired, select the **Marked Features Only** check box to only export the marked features.
4. If desired, select the **Scans as Points** check box to cause scan data to get converted to its scanned points.
5. If desired, select the **Output Nominal Values** check box to cause PC-DMIS to create the export file from the nominal values in your measurement routine. If you don't select this check box, PC-DMIS exports the measured feature results.
6. Click the **Export** button to finish exporting the STEP file.

Exporting a VDAFS File

When you click the **Export** button during the VDAFS export process (**File | Export | VDAFS**), PC-DMIS displays the **VDAFS Export** dialog box.



VDAFS Export Dialog Box

The **VDAFS Export** dialog box allows you to select options for your VDAFS export. This dialog box has the same options as the **STEP Export** dialog box. For information on these options, see "Exporting a STEP File".

Exporting to a BASIC File

PC-DMIS provides you with tools that help you to automate PC-DMIS. Exporting your measurement routine as a BASIC script (*.BAS file) allows you to then automate that measurement routine using any external application that supports the BASIC language. For more information on how to automate PC-DMIS, see these documentation topics:

- PC-DMIS BASIC Language Reference documentation.
- "Inserting BASIC Scripts" in the "Adding External Elements" documentation.

Exporting to a Datalog File

The **File | Export | Datalog** menu item exports your measurement routine's feature and dimension data to a simple Datalog text file. Datalog is a proprietary Chrysler format used by Chrysler's internal applications.

When you choose to export the file, a **Datalog Export** dialog box appears. Choose a location to store the exported text file, and click **Save**. You can view the resulting file by opening the file inside of a text editor such as Notepad.

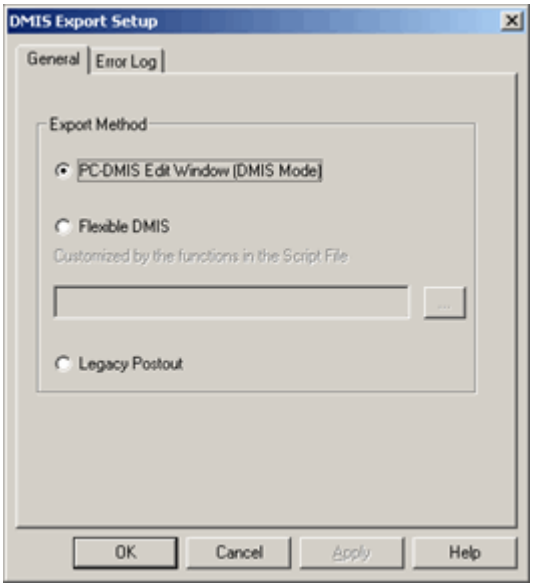
Exporting to a DMIS File

DMIS files adhere to the DMIS standards for DMIS commands. You can execute DMIS files on other applications that use the DMIS language.

To export your PC-DMIS measurement routine file as a DMIS file, follow these steps:

1. Select **File | Export | DMIS**. This action opens the **Export** dialog box.
2. In the **Export** dialog box, choose a file name and click **Export** to open the **DMIS Export Setup** dialog box. This dialog box contains the **General** tab and the **Error Log** tab.
3. After you choose your options on these tabs, click **OK** to complete the export process.

General tab



DMIS Export Setup Dialog Box - General Tab

The **General** tab lets you choose one of three methods to convert your PC-DMIS measurement routine into a DMIS file.

- **PC-DMIS Edit Window (DMIS Mode)** - This method exports a DMIS file exactly as the measurement routine looks in the Edit window when you are in DMIS Mode.

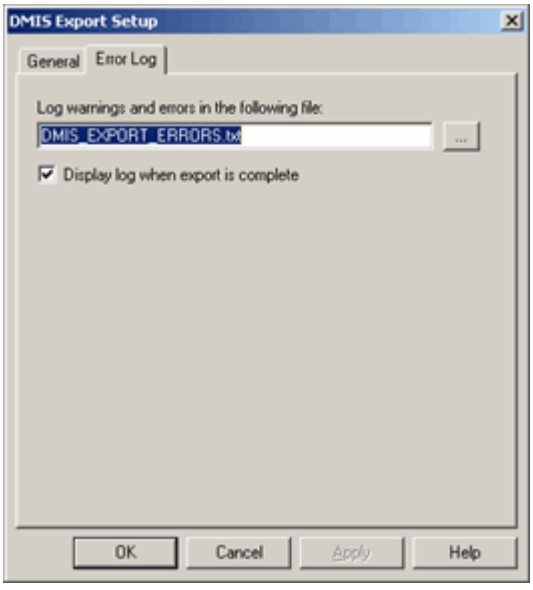
- **Flexible DMIS** - This method allows you to customize the exported file to match your specific DMIS version, measurement methods, or supported DMIS statements. When you select this option, a file selection box becomes enabled that allows you to browse to and select a specific BASIC script file (*.bas extension). You can create your own script file, or modify an existing script file to control exactly what PC-DMIS should export.

  PC-DMIS provides you with three ready-to-use .bas files that reside in your PC-DMIS installation directory. These files customize your export so that it uses a defined subset of supported DMIS statements for a specific DMIS version. The files are PCD2DMIS030.BAS, PCD2DMIS040.BAS, and PCD2DMIS050.BAS (for DMIS 3.0, 4.0, and 5.0 respectively). The commands that are not customized in the .bas file are exported as they are for the **PC-DMIS Edit Window (DMIS Mode)** option.

  If a function in the BASIC script returns an error when you use this option, then the exported file gets written as if you had selected the **PC-DMIS Edit Window (DMIS Mode)** option above.

- **Legacy Postout** - This method exports the file as it used to do in older versions of PC-DMIS.

Error Log tab



DMIS Export Setup Dialog Box - Error Log Tab

The **Error Log** tab lets you view or save errors encountered during an export phase so that you can correct them in your script file or take some other appropriate action. PC-DMIS displays errors or warnings in these instances:

- When a function in the BASIC script returns a logical error, meaning you have a problem somewhere in your script that's causing an automation logic error. In this

case, the BASIC Script function returns a text string with this header: `$$ LOGERR |` .

- When a parameter in a PC-DMIS command cannot be converted into DMIS code. In this case, the BASIC Script function returns a text string with this header: `$$ DMISERR |`

Click the **...** button to locate and select an existing text file to which PC-DMIS should send the errors.

Click the **Display log when export is complete** check box to display the text file in a text editor once PC-DMIS finishes the export process.

SuppressDMESWComandOutput Entry

A `SuppressDMESWComandOutput` entry lets you suppress `DMESW/COMAND` statements in the output file with the $$ comment marker prefix if you use the **Flexible DMIS** export method discussed above. For more information, see the "SuppressDMESWComandOutput" topic in the PC-DMIS Settings Editor documentation.

Exporting to Excel

PC-DMIS allows you to export your measurement routine's results as text data to a Microsoft Excel template. To do this, select **Insert | Report Command | Excel Form Report** to open the **Excel Form Report** dialog box.

For more information, see the "Using the Excel Form Report Command" section of the PC-DMIS Toolkit Modules documentation.

Exporting to an Inspection Plan

Inspection plans determine which features of the geometry and GD&T/dimension to use in the inspection process. Usually, inspection plans are created using Planner, an application that shares many similarities to PC-DMIS. However, you can also use the **File | Export | Inspection Plan** menu option in PC-DMIS to export your measurement routine as an inspection plan.

- **Planner XML (*.planxml, *.xml)** - PC-DMIS can import this default file type into an open measurement routine. Planner can do the same into an open inspection plan. In addition, third-party applications can potentially parse and use the data contained inside this widely-used tagged format. This is the format used by Planner.
- **Planner IP File (*.ip)** - Both PC-DMIS and the legacy PC-DMIS Planner can import this older proprietary format. This is the format used by the legacy PC-DMIS Planner.

You can then import the IP into a measurement routine (see "Importing an Inspection Plan File") or load the plan into PC-DMIS Planner for further development.

PC-DMIS 2019 R1 was the last version that included Planner.

Exporting to an XML File

You can use the **File | Export | XML** menu item to export your PC-DMIS measurement routine's measurement data into an XML (.xml) file. You can then share the file or store the data on a database server.

From the **Export Measurement Routine to XML** dialog box, type or select a filename from the list and click **Save** to complete the export process.

Using Direct CAD Interfaces

Direct CAD Interfaces (DCI) allow you to connect directly to your CAD software. As the CAD file is updated in your CAD software, PC-DMIS reflects the corresponding changes.

Instead of translating CAD data into the internal CAD format of PC-DMIS, the Direct CAD file types are accessed directly. That is, the CAD data of these file types remains in its native format after being imported. Routines native to the CAD type are used for all calculations done on the CAD data. Accessing the CAD data directly eliminates problems that can occur because of translation inaccuracies or limitations.

For detailed information, see the "Direct CAD Interface" documentation.

PC-DMIS supports the use of the following DCIs:

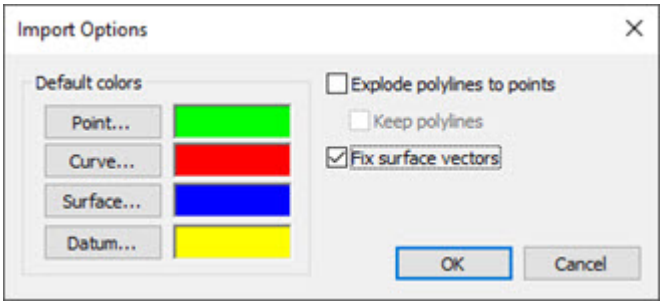
- ACIS
- CATIA 5
- Creo
- Solidworks
- NX

Your PC-DMIS license must have the following options enabled: **CATIA V5 DCI**, **NX DCI**, **PRO-E DCI**, and **Solidworks DCI**.

Setting Import Options

You can easily set import options to determine the default colors for certain imported entity types, as well as how PC-DMIS displays imported curves.

To perform these manipulations, select the **Edit | Preferences | Import Options** menu item. This menu item displays the **Import Options** dialog box.

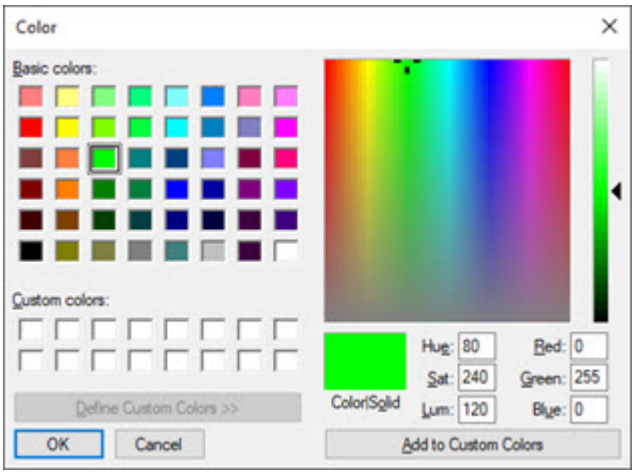


Import Options Dialog Box

This dialog box contains the **Default Colors** area as well as some check boxes.

Default Colors

This area lets you change the default colors for imported Points, Curves, Surfaces, and Datum entity types. If the entity types don't already have a defined color, it uses this default color. To change a color, simply click on a button in this area. A standard **Color** dialog box appears, allowing you to select a new color.



Color Dialog Box

When you import the next feature, PC-DMIS uses the newly-defined colors.

Check Boxes

Explode polylines to points

Usually, when you import curves entities, they appear as individual curves. In reality, however, each curve is really a polyline, a bunch of lines connected by a series of points. Selecting this check box makes imported polyline curve entities appear as a series of points, one point for each polyline vertex. Clearing this check box makes imported curves appear as normal.

Keep polylines

Selecting this check box allows the image to continue to display the original polyline along with the points when you select the **Explode polyline to points** check box. Clearing this check box only shows the series of points.

Fix Surface Vectors
If you want your surface vectors to be correct and always point away from the surface, mark the **Fix surface vectors** check box. This is helpful for pointcloud and laser scanning operations, but it does increase the import processing time. If you don't use laser scanning, you can clear this check box.

PC-DMIS uses these settings for all future import operations.

Executing Measurement Routines

With PC-DMIS, you can easily execute an entire measurement routine or perform partial executions. This topic describes how to perform both full and partial executions. It also explains how to use the **Execution** dialog box to execute your measurement routines and how to resize the dialog box.

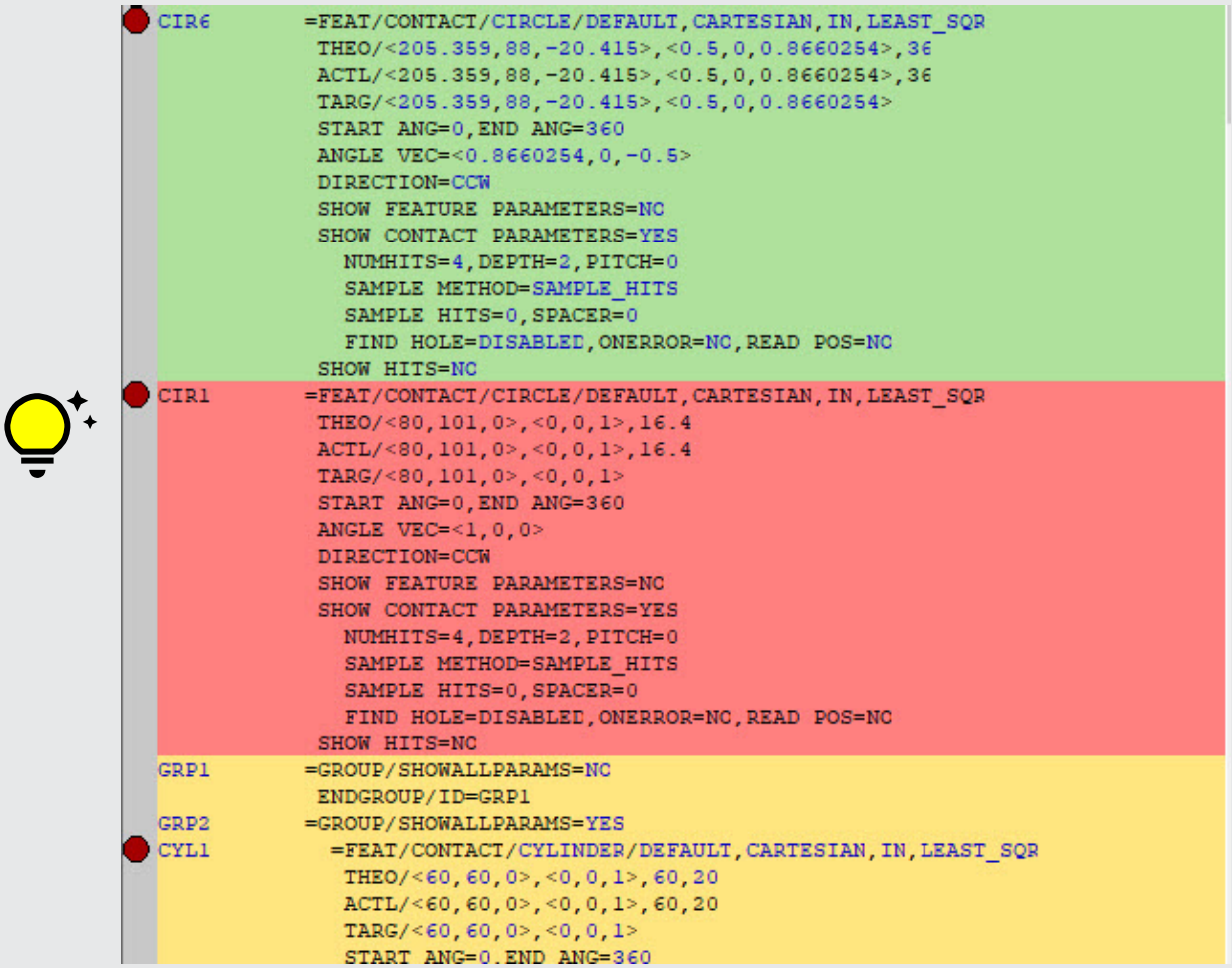
When you execute some or all of your measurement routine, select **File | Execute** or a menu item from the **File | Partial Execution** submenu to open the **Execution** dialog box. Also, if you don't have breakpoints in your measurement routine, PC-DMIS displays a different layout of the screen display called the "execution layout". This layout initially hides the Edit window and expands the Graphic Display window, to show more of the part. It also displays the Probe Readouts window. You can change what windows appear during execution by pausing the execution and showing or hiding a window. Subsequent executions use your updated execution layout.

When execution finishes, the screen layout returns to the learn time layout. Any window you show or hide during learn time becomes part of the learn time layout.

When the PC-DMIS Protect license option is enabled, operators (users who are not members of the PC-DMIS-Programmer or PC-DMIS-User group) can execute only certified measurement routines. To let operators also execute uncertified measurement routines, set the `RestrictCertifiedMRExecutionToOperators` entry in the **Option** section of the PC-DMIS Settings Editor to **FALSE**. For information on certified measurement routines, see the "Certify" topic.

If you execute a measurement routine with breakpoints, PC-DMIS continues to display the Edit window during execution:

In Command Mode, PC-DMIS highlights the current command to execute in red.



Example of colors used in Command Mode during execution with a breakpoint

Execute

To execute all marked commands of the entire measurement routine, select the **File | Execute** option.

Execute Feature

To execute only the feature or operator comment command on which the cursor rests, select the **File | Partial Execution | Execute Feature** option.

PC-DMIS displays the **Execution** dialog box.

If PC-DMIS is set to Manual Mode (mode = MANUAL), automatic machine motion does not occur. Instead, PC-DMIS prompts you to move the machine to take the necessary hits.

If PC-DMIS is set to DCC Mode (mode = DCC), it automatically moves the probe based on the parameters set in the dialog box.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

To maintain good execution speeds, PC-DMIS does not generate a report in the Report window if using the **Execute Feature** functionality. However, you can still view report-like information inside the Status window.

Execute From

To resume a previous cancelled execution, use the **File | Partial Execution | Execute From** menu option. This command executes the feature listed until the end of the measurement routine. This menu option is only available when you have canceled a prior execute command by using the Execution dialog box.

If during a measurement routine execution, you click the **Cancel** button while measuring feature CIR1, the Execute From menu becomes available so that you can continue inspecting the part beginning at CIR1.

Execute From Cursor

To execute the measurement routine from the current cursor location, select the **File | Partial Execution | Execute From Cursor** menu option. The measurement routine begins execution at the cursor's current location.

Use the Execute From Cursor option only for troubleshooting, not for full executions after you thoroughly test the measurement routines. This command retains data from the previous execution by design, and prolonged use may impact system performance.

To skip specific commands, such as the manual features in a measurement routine, you can use the Mark option. For information on how to use the Mark option, see the "Mark" topic in the PC-DMIS Core documentation.

Execute Block

To execute a block of commands, select the commands you want to execute, and then with the commands still selected, choose the **File | Partial Execution | Execute Block** menu command. PC-DMIS will execute only the selected block of commands.

Execute From Start Points

To begin execution at a specified start point, first set the start point by right-clicking inside Command Mode and selecting **Set Start Point** from the shortcut menu, and then choose the **Execute from Start Points** menu command.

> Be aware when the current tip for that location in the measurement routine does not match the current orientation of the probe head. The software does not execute the previous tip command in order to change the tip orientation.

Mini Routines

Mini routines enable you to measure a selected dimension or group of dimensions from a long measurement routine.

Because mini routines can execute partial measurements in a measurement routine, you must ensure that there will be no collisions when you execute a partial measurement routine. You are responsible for creating such a measurement routine. For details, see "Mini Routines" in the PC-DMIS Core documentation.

Dynamic Inspection

To execute Dynamic Inspection, select the **File | Partial Execution | Dynamic Inspection** menu option or click the **Dynamic Inspection** icon from the **Execute** toolbar to open the **Dynamic Inspection** dialog box. This dialog box displays the characteristics that O-QIS selected to measure the next part. Select the appropriate options and click **Execute** to execute the measurement routine.

For more information, see the "Dynamic Inspection" chapter in the PC-DMIS Toolkit Modules documentation.

Any Order Execution

You can execute manually-measured features in the measurement routine in any order. This is useful for portable arm machines.

This functionality becomes possible if the following conditions are met:

- It only applies to features measured with manual hits.
- The measurement routines cannot have any branching or loops.
- The features that you measure out of order must use the same alignment and probe as the features initially defined to measure in the current measurement routine. PC-DMIS does not search across `LOAD PROBE` or `ALIGNMENT` commands.
- Out of order execution only works for features measured after a complete alignment. The initial alignment must be measured in order.

How It Works

When you execute your measurement routine, PC-DMIS prompts you to take the first hit on the first feature. You can decide to take your first hit on a different feature instead. When you do this, PC-DMIS determines if you took that hit within a tolerance of the hit PC-DMIS expected. If your hit is more than a specified tolerance (usually 10 mm) away from the feature, then PC-DMIS searches backwards and forwards through the measurement routine for the closest feature that has an initial hit that matches the initial hit that you took. You can define the tolerance on the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**).

If you take a hit that is not in tolerance for any feature in your entire measurement routine, the arm sounds a beep alert, and PC-DMIS ignores the hit. PC-DMIS shows an error message in the **Execution** dialog box. You should take a different hit closer to a measurement routine feature to continue.

When PC-DMIS finds the appropriate feature, it passes your first hit into the feature and then moves the arrow to indicate where to take the next hit for the new feature. PC-DMIS only checks the tolerance on the first hit of the feature. After that, it assumes that you will continue taking hits until you finish measuring that feature.

Once you finish measuring a feature out of order, PC-DMIS attempts to go back to its initial order. It continues to prompt you to measure the original feature each time you finish measuring something else out of order until you either skip the original feature or measure it. PC-DMIS tracks the features you measure out of order so that they aren't remeasured when PC-DMIS resumes its regular execution route.

If you remeasure a feature, PC-DMIS recomputes everything that uses that feature up until the current execution location.

Reset Execution List

As execution progresses, PC-DMIS maintains an internal list containing all the commands that have been executed. This is called the execution list. PC-DMIS uses the execution list for generating reports in the Report window. This list is saved with the measurement routine and restored when the measurement routine is loaded.

PC-DMIS always clears out the information in this list whenever you perform a full measurement routine execution.

However, if you perform a partial execution (using execution menu items from the **File | Partial Execution** sub menu), it does not clear out the execution list. For example, if you perform a full execution, followed by a partial execution, the information from the full execution remains in your execution list and shows up in your report along with the partial execution information.

If you don't want this, the **File | Reset Execution List** menu item immediately clears out the execution list.

The `ClearExecutionListPriorToProgramExecution` entry is available to modify how the Execution List is cleared out. Details can be found in the **Reporting** section of the PC-DMIS Settings Editor.

Additionally, the `SaveExecuteList` entry in the **Reporting** section in the PC-DMIS Settings Editor controls this behavior. It defaults to 1, meaning the list is saved and restored as described. If you don't want PC-DMIS to save or restore the list, change this setting to 0.

Reporting a Dimension Using a Feature Skipped During Execution

If you skip a feature when you execute a measurement routine, PC-DMIS displays dimensions in red when the input feature was set to:

- SKIPPED_DURING_EXECUTION - This occurs when you get the PC-DMIS prompt "Read new probe position" and you press **No** to skip.
- SKIPPED_DURING_EXECUTION_BECAUSE_INPUT_SKIPPED - This occurs from a constructed feature type that has a skipped feature as one of its inputs.

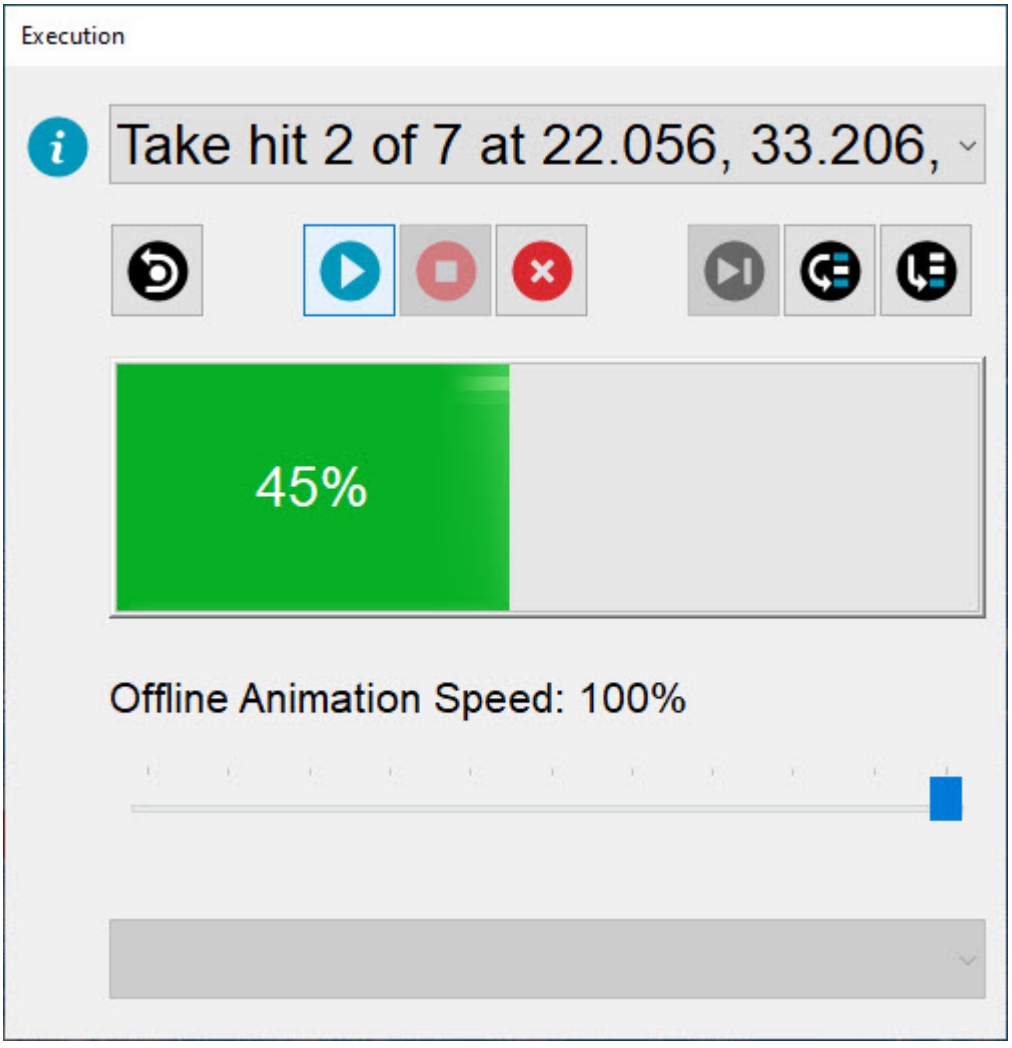
Previously, PC-DMIS was not refreshing dimensions that use a skipped feature during measurement routine reporting. When you create a new dimension that uses a skipped feature, it will have a different result than it would after you execute the measurement routine. Now, PC-DMIS marks the dimensions in red to highlight that it too is skipped because an input was skipped during execution. This "red" state is only present after execution. If you close and re-open the measurement routine, it returns to the original text colors.

Using the Execution Dialog Box

You can use the On Error command to tell PC-DMIS to skip commands that generate certain machine errors during the execution of a measurement routine. For information about On Error branching, see the "Branching on an Error" topic.

You can use the Check Calibration Limits command to check for probe calibration errors and raise alerts during execution. You can combine this command with the On Error command to control the execution. For information about the Check Calibration Limits command, see the "Creating a Check Calibration Limits Command" topic in the PC-DMIS Toolkit Modules documentation.

To begin all execution operations, open the **Execution** dialog box (**File | Execute** or **File | Partial Execution**).



Execution Dialog Box

Once the **Execution** dialog box appears, PC-DMIS attempts to execute all marked commands in the current measurement routine.

- If PC-DMIS is in Manual Mode, PC-DMIS requests that you manually move the probe to the feature and take hits with the probe.
- If PC-DMIS is in DCC Mode, PC-DMIS automatically begins the measurement process. It moves to and probes each feature and takes the needed number of hits.

> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

These items are available in the **Execution** dialog box:

Machine Commands - Displays the current action to perform.

These include any moves and hits for the current feature (the feature about to be measured). These commands are only available if the measurement routine is stopped during execution, from an error, from the operator, or if the execution takes place in Manual Mode. If this takes place in DCC Mode, you can resume the measurement routine execution by highlighting the desired hit or move from the **Machine Commands** list and clicking **Continue**.

> Suppose a clearance move command was programmed with the wrong X, Y, or Z values. During the execution of that move, you might get a motion error. To continue, you would do the following:
>
> 1. Select the **Machine Commands** list.
> 2. Select the command following the offending move.
> 3. Click **Continue**.

Erase Hit - Removes the currently-highlighted hit in the **Machine Commands** list.

Most CMM jog boxes also have a button assigned to erasing the last hit. Pressing that button on the CMM's jog box while in Execute mode, produces the same result as clicking the **Erase Hit** button on the dialog box.

New Row - Starts a new row for the manual scan being executed (or learned).

This button is only available when a manual scan is executed.

To use the **New Row** command button:

1. Press **Stop** to pause the scan.
2. Click **New Row** . This indicates a new row to scanned.

Or

1. Click **New Row** first. PC-DMIS automatically pauses the scan.
2. Move the probe to the next row.
3. Click **Continue** .
4. PC-DMIS scans the new row.

Continue - Resumes the execution of a measurement routine halted by a CMM motion error, or the **Stop** button.

You can choose the specific location within the current feature where the probe is to resume measuring the part. The default location is displayed in the **Machine Commands** list. To select another location, click the list and select the desired line. PC-DMIS continues the measurement process at that location once you click **Continue**.

The software does not know why you chose to pause an execution. Therefore, to safely resume a paused execution, check to ensure that the probe has a clear lane of travel to its next point. A message box reminds you to do this in these circumstances:

- The probe is already inside the ClearanceCube.
- The probe is outside the ClearanceCube but the first target position exists within the ClearanceCube.
- Anytime the first movement intersects with the ClearanceCube.

For information on the ClearanceCube, see "Using the ClearanceCube" in the "Editing the CAD Display" chapter.

Stop - Halts the probe at its current position and suspends execution of the measurement routine.

> This button does not stop the measurement routine execution if the current move is a PH9 orientation change. The measurement routine stops after the PH9 stops.

The topic "Manually Controlling a Feature's Measurement" discusses using **Stop** to switch to Manual Mode during the execution process.

Cancel - Stops the execution and closes the dialog box.

Scan Done - Stops collecting data in a manual scan and processes the data for finding nominals, vector compensation, and other data.

This is only available if a manual scan is executed (learned).

As you execute the scan, PC-DMIS gathers data from the CMM. Simultaneously, PC-DMIS filters (or reduces) the data according to the scan method (**Fixed Time**, **Fixed Distance**, **Fixed Time / Distance**, and other methods) you provide. PC-DMIS excludes data that does not satisfy the criteria (such as **Fixed Time** or **Distance** increments) and keeps data that does satisfy the criteria.

Step Next - Continues the measurement process by stepping through each new command that moves the CMM.

This pauses the CMM between each step, which requires you to click **Step Next** to continue to step through the routine. This puts PC-DMIS into Step Mode. When PC-DMIS is in Step Mode, you can insert hits into features and insert new features between existing features or commands. If you insert a new feature, PC-DMIS displays a dialog box that enables you to do the following:

- **Skip** to the `HIT /BASIC` command of the new feature.
- **Remeasure** the new feature.
- **Continue** to step through the measurement routine, and ignore the new feature entirely.

You can also simulate Step Mode offline.

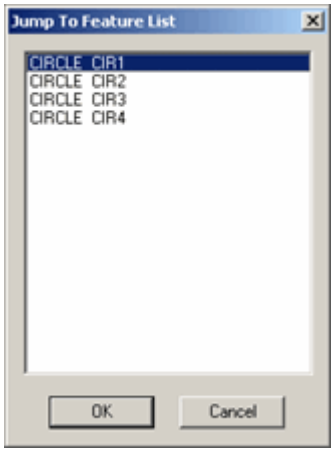
Step Next appears in the dialog box when PC-DMIS pauses execution for a breakpoint (see "Using Breakpoints" in the "Editing a Measurement Routine" chapter).

Skip - Skips the next command displayed in the **Machine Commands** list. It also skips commands that depend on the skipped command.

For details on how PC-DMIS reports dimensions that use skipped features during execution, see the "Reporting a Dimension Using a Feature Skipped During Execution" topic in this documentation.

Jump - Stops execution so that you can jump to a new feature.

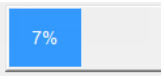
This displays the **Jump to Feature List** dialog box. From this list, you can select the next feature to execute and deviate from the automatically-generated execution path.



Jump to Feature List Dialog Box

After you select a feature from the list and click **OK**, PC-DMIS executes the selected feature. After PC-DMIS executes that feature, it returns to the pre-defined path and attempts to execute the next non-measured feature until you click **Jump** again and select a new feature.

Suppose you have seven circles labeled CIR1 through CIR7, and they are programmed to execute in that order. If after CIR1, you click **Jump** and select CIR6, PC-DMIS immediately measures CIR6. When finished, PC-DMIS returns and measure circles CIR2 through CIR5, and then it finishes by measuring CIR7.



Progress Bar - Indicates the percentage of the measurement routine that has finished executing.

The percentage is derived from the amount of executed commands and hits taken compared against the total number of marked commands and hits to take. The progress bar updates as it progresses through the measurement routine. You may

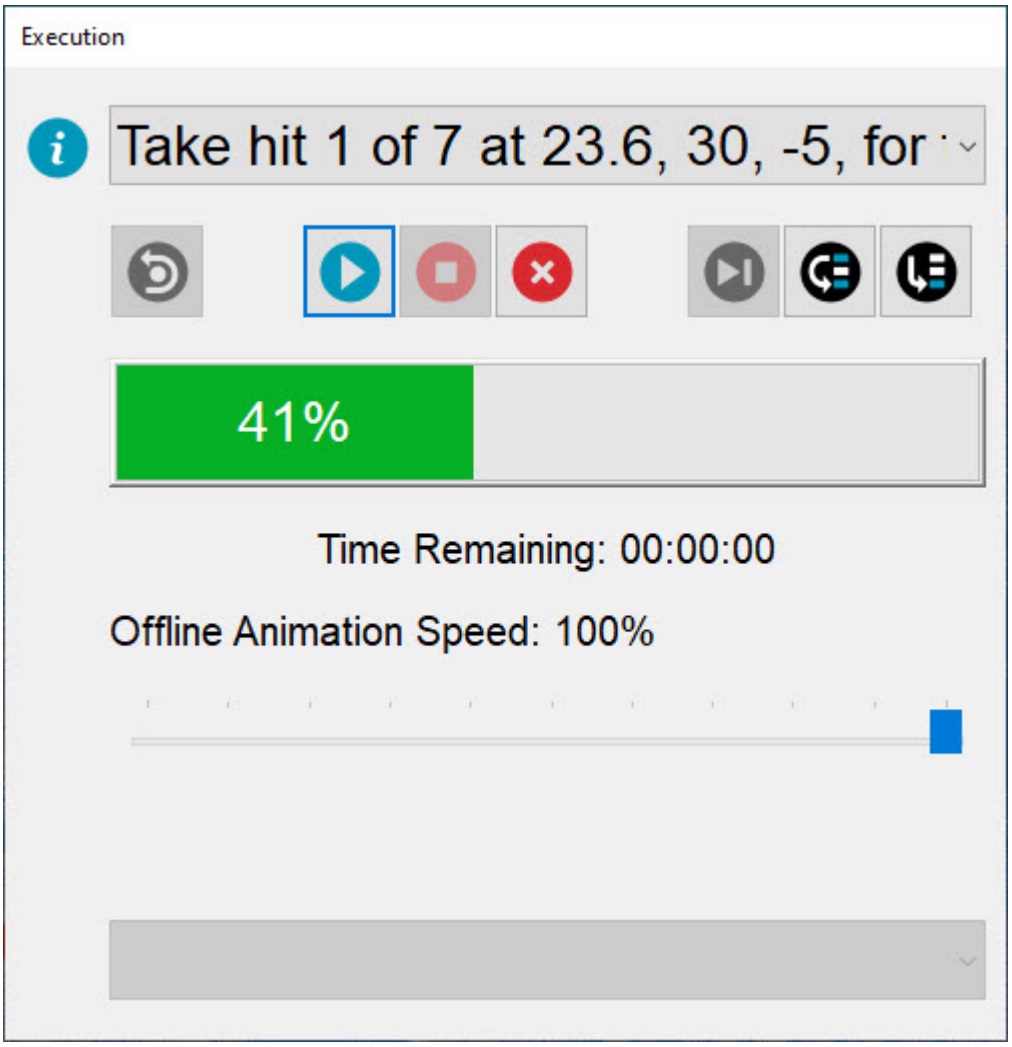
find this useful when you have a large measurement routine and want a general idea of how long it takes to complete.

> This functionality does not factor in individual scan hits or alterations to the number of executed items due to flow control statements.

Machine Errors - Displays any errors that occur during the execution of the measurement routine.

Some possible errors are an unexpected probe hit or an unexpected end of move. You can view the errors by clicking this list.

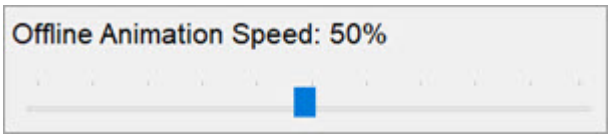
Time Remaining - Displays the execution time (in <hours>:<minutes>:<seconds>) that remains for a measurement routine or mini routine if the **Record and display execution time** check box in the **Execution** area on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**) is selected. For example:



Example of the Execution dialog box with remaining time.

If the **Add to execution time (seconds)** check box in the **Execution** area is also selected, PC-DMIS adds the number of seconds that are entered in the box to the measurement routine or mini routine execution time when the execution finishes.

Offline Animation Speed slider - The **Offline Animation Speed** slider is only available when you run your measurement routine in Offline mode. The slider allows you to adjust the speed of the execution of the measurement routine in Offline mode as a percentage of the **Max animation speed** value. The default value is 100%.



The **Offline Animation Speed** slider on the **Execution** dialog box is directly linked to the **Animation speed %** slider option on the **Animation** tab of the **Setup Options** dialog box. The value you set the slider to on the **Execution** dialog box, for example, is also set for the slider on the **Animation** tab of the **Setup Options** dialog box. When you close and then reopen the measurement routine, the slider value in both locations is set back to the default value of 100%.

For details on measurement routine animation setup options, see the "Execution area" section of the "Setup Options: Animation tab" topic in the PC-DMIS Core documentation.

When you execute a measurement routine for dual arms in Offline mode, PC-DMIS shows two **Execution** dialog boxes, each with an **Offline Animation Speed** slider. The sliders are synchronized so, if you adjust one slider, the other slider adjusts to the same percentage value.

Note the following:

- PC-DMIS records the remaining time for only the DCC portion of a measurement routine.
- PC-DMIS stops recording the remaining time when the measurement routine or mini routine pauses due to attention required by the user. For example, the execution may pause when a comment is executed, a message appears, or an error appears, and the execution stops.
- PC-DMIS does not record the remaining time if the execution does not get completed or if it is suspended (see **Stop** above).

Manually Controlling a Feature's Measurement

To manually control the measurement of a feature:

1. From the **Execution** dialog box (**File | Execute**), click **Stop** .

2. From the **Probe Mode** toolbar, click the **Manual Mode** icon . You are given control for measuring the current hit.

Once you finish measuring the current feature in MANUAL mode, PC-DMIS then automatically returns the system to DCC Mode and continues to measure the part under computer control. You can click the **DCC Mode** icon to go back to DCC Mode at any time.

PC-DMIS does *not* replace the original hits with any of the new hits. If a measurement value needs to be altered, it must be done in the Edit window.

At the end of manual measurement of some Auto features, the software may prompt for you to save the new location of the feature as the target for future executions of the measurement routine. If you click **YES** in response to this prompt, the software saves the currently measured position as the new target.

Re-sizing the Execution Dialog Box

You can re-size the **Execution** dialog box (**File | Execute** or **File | Partial Execution**):

1. Place the pointer on the outer edge of the dialog box. The arrow icon for the pointer changes to a double arrow icon.

2. Drag the pointer to the desired location and release the mouse button to resize the dialog box.

You may find this capability useful when you need a larger dialog box in order to read the data better from a distance or if you want to use larger icons.

Setting Your Preferences

Setting Your Preferences: Introduction

PC-DMIS allows you to set personal preferences that tailor the form and function of PC-DMIS to meet individual specifications. Using this chapter, you will be able to control output format, screen graphics, machine parameters, and other options.

The main topics in this chapter include:

- Selecting Setup Options
- Modifying Report and Motion Parameters
- Setting Up the Edit Window
- Setting Up the Probe Readouts Window
- Setting Up Multiple Arms
- Defining the Rotary Table
- Setting Up Probe Changer Options
- Managing Multiple Probe Changers
- Loading the Active Probe
- Examples of Working with Single and Multiple Probe Changers
- Setting Up the Machine Interface
- Compensating for Temperature
- Using Simplified Temperature Compensation
- Temperature Gage
- Editing Part Materials and Coefficients
- Specifying Search Paths
- Changing OpenGL Options
- Using the Measurement Strategy Editor
- Setting Up Sensor Mapping

- Setting Import Options
- Understanding File Locations
- Understanding Data Files
- Defining Default Environment
- PC-DMIS FUSION Settings

The following paragraphs describe the various system options and their functions.

A Note on Storing Settings for Multiple Users

Switching Between Machine Profiles

A Note on Storing Settings for Multiple Users

PC-DMIS now stores all changes made to the software's settings, parameters, or user-interface customizations for each user. This is controlled internally by using Windows user permissions. When you log onto the operating system with your specific user name, the system automatically recalls your settings. The system stores your settings inside named sub directories where you installed PC-DMIS.

Switching Between Machine Profiles

Your license must have the **IP Measure** option for the **Save Machine Profile** and **Recall Machine** menu items to appear.

These two menu items let you easily switch between machine profiles when you create measurement routines in Offline mode.

Edit | Preferences | Save Machine Profile - This menu item displays a **Save As** dialog box that asks for a file name. It records all the current settings for the virtual machine in a file with an "mpl" file name extension. PC-DMIS saves the following into the file for later recall:

- Loaded machine model
- Loaded tool changer model
- Rotary table settings
- Mount orientation for the probe
- Starting default probe component (PH10, CW43, and so on)
- Active probe file

- Any "fixed" quick fixture components
- Machine's speed settings and limits

Edit | Preferences | Recall Machine - This menu item lets you restore a machine's settings that were stored in an .mpl file. PC-DMIS displays an **Open** dialog box. Once you select the file to open, PC-DMIS restores the settings.

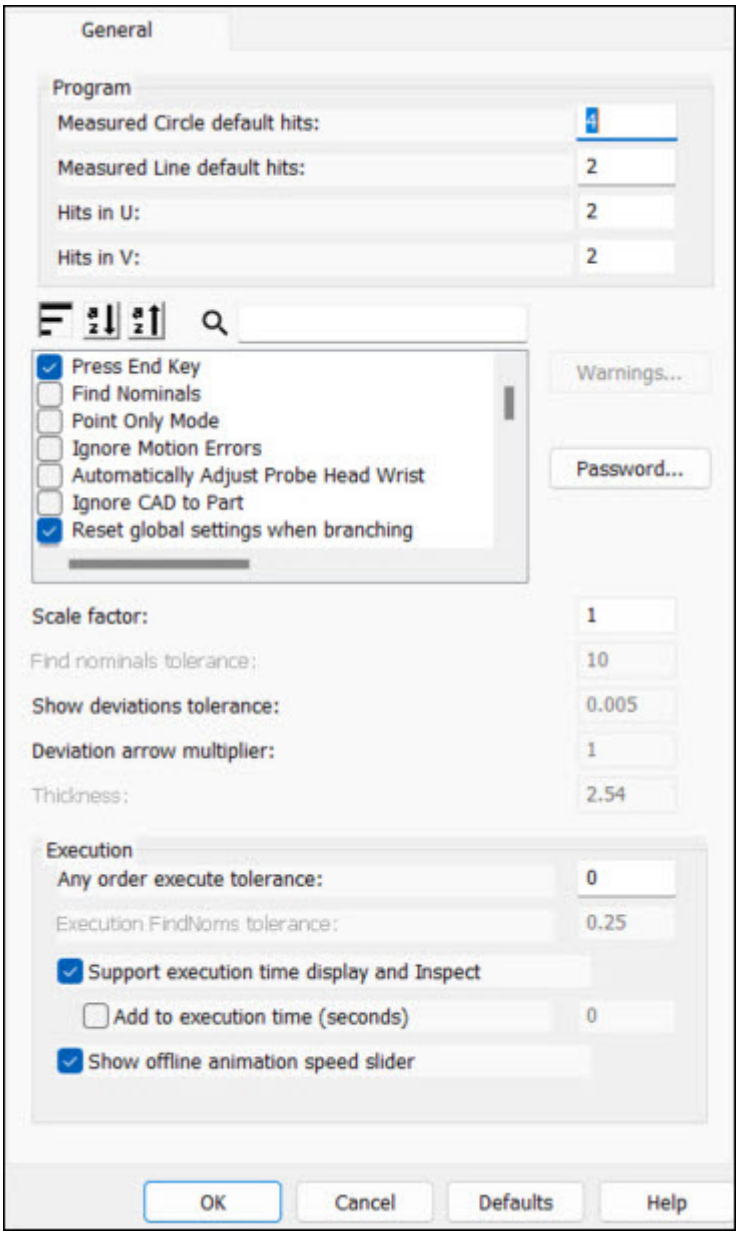
Selecting Setup Options

Selecting the **Edit | Preferences | Setup** menu option accesses the **Setup Options** dialog box. Use the tabs in this dialog box to alter the form and function of PC-DMIS. The tabs include the following:

- Setup Options: General tab
- Setup Options: Part/Machine tab
- Setup Options: Dimension tab
- Setup Options: Geometric Tolerances tab
- Setup Options: ID Setup tab
- Setup Options: Laser Sensor tab
- Setup Options: Sound Events tab
- Setup Options: Animation tab

For information on how the **OK** and **Defaults** buttons work, see the Command Buttons topic in the "Navigating the User Interface" chapter.

Setup Options: General tab



Setup Options Dialog Box - General Tab

To access the **General** tab, open the **Setup Options** dialog box (**Edit | Preferences | Setup**) and select the **General** tab. You can use this tab to change various functions that control the measurement process.

Program Area

The edit boxes in the **Program** area of the **General** tab allow you to edit the following options:

Measured Circle default hits

The **Measured Circle default hits** box allows you to set the default number of hits PC-DMIS takes while learning circles using CAD data. The minimum number of hits that you can type in is three. It only changes the number of hits for circles about to be programmed, not for circles that have already been programmed.

The entry in the PC-DMIS Settings Editor for this option reads: **AutoCirHits= [number of hits]**. For information on modifying entries, please view the "Modifying Setting Entries" chapter.

Measured Line default hits

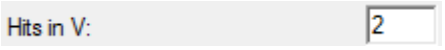
The **Measured Line default hits** box allows you to set the default number of hits PC-DMIS takes while learning lines using CAD data. The minimum number of hits that you can type in is two. It only changes the number of hits for lines about to be programmed, not for lines that have already been programmed.

The entry in the PC-DMIS Settings Editor for this option reads: **AutoLineHits= [number of hits]**. For information on how to modify entries, please view the "Modifying Setting Entries" chapter.

Hits in U

The **Hits in U** box indicates the minimum number of rows that must be taken when scanning.

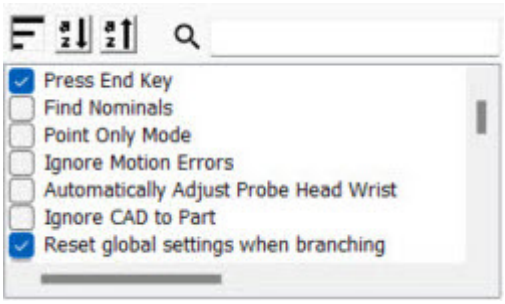
Hits in V



The **Hits in V** box indicates the minimum number of hits per row that must be taken when scanning.

The **Hits in U** and **Hits in V** scanning options are only active when you use them with the Curves and Surfaces options.

Check Boxes for the General tab



The **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**) lets you turn various options on or off. This allows you to customize the PC-DMIS Setup options to fit your specific needs.

You can use these options to sort or search the list:

Sort Default button - Click this button to display the list in the default order as it appears when you open the dialog box.

Sort A to Z button - Click this button to sort the list in alphabetical order (A - Z).

Sort Z to A button - Click this button to sort the list in reverse alphabetical order (Z - A).

Search box - Type any text in this box to filter the list and display only the options that contain the entered string.

Press End Key

The **Press End Key** check box controls whether or not PC-DMIS waits for you to press the END key before it accepts the last hit taken. Selecting this check box allows you to preview the last hit before you accept it. If you don't select this check box, the current hit that you take with the jog box is not stored in the hit buffer, but automatically becomes the last hit for the feature, even if you take the hit at the wrong spot.

Find Nominals

The **Find Nominals** check box controls how PC-DMIS treats hits. When selected, PC-DMIS will then automatically consider each probe touch, finding the closest CAD nominal for that touch. It continues to accumulate hits until you press the End key. PC-DMIS then calculates the feature type and applies the CAD nominals.

Once selected, the **Find Nominals Tolerance** box becomes available. See "Find Nominals Tolerance".

Point Only Mode

The **Point Only Mode** check box controls how PC-DMIS responds to each probe hit. When selected, PC-DMIS automatically considers each probe hit to be a single point measurement and automatically creates an Auto Vector Point. If the check box is not selected, PC-DMIS will accumulate probe hits until the END key is pressed. Only then will it determine the type of feature just measured.

When you select this check box, the **Thickness for Point only Mode Points** check box becomes available. See "Thickness for Point Only Mode Points".

Edge Point Only Mode

The **Edge Point Only Mode** check box controls how PC-DMIS responds to each set of probe hits. In this case, a set is defined as two singular probe hits. The first hit should always be taken on the surface. The second hit should always be taken on the edge. When you select this check box, PC-DMIS automatically considers each set of probe hits to be a single point measurement and automatically creates an Auto Edge Point. If you do not select the check box, PC-DMIS accumulates probe hits until you press the End key. Only then does PC-DMIS determine the type of feature you measured.

PC-DMIS uses the options as displayed in the **Hits**, **Auto**, and **Misc** areas of the **Edge Point** dialog box when you create an Edge Point (see the "Creating Auto Features" chapter).

PC-DMIS applies the CAD nominals if you marked the **Find Noms** check box option.

Ignore Motion Errors

Select **Insert | Parameter Change | Probe | Ignore Motion Errors** to access the **Ignore Motion Errors** option.

This option does not apply to all machine types. Some machines can utilize this option, while on others it will have no effect. For those machine types where it does have an effect, the exact nature of the effect may also depend on the machine type.

Selecting the **Ignore Motion Errors** check box tells PC-DMIS to try to avoid stopping from a motion error during movement when the ignore request is active. If you select this check box and click **OK**, PC-DMIS inserts an `IGNOREMOTIONERRORS/ON` command into the Edit window.

When PC-DMIS executes the `IGNOREMOTIONERRORS/ON` command, the command enables a state where PC-DMIS will try to avoid stopping due to motion errors, if possible, when it enters DCC motion. The exact manner of implementation may vary by machine type.

On traditional CMMs with contact probes, the most common implementation is through disabling the probe. However, if PC-DMIS disables the probe during a move, normally it does need to re-enable the probe to take a measurement. This may result in a repeating

sequence where PC-DMIS disables the probe for transitional moves between measurements but then re-enables the probe when it actually takes the measurement.

You can use this command to prevent problems from spurious vibration or acceleration-induced triggering, or both.

Automatically Adjust Probe Head Wrist

If you select the **Automatically Adjust Probe Head Wrist** check box, then during execution when the software reaches a TIP command, it will, based on the geometry, automatically select the tip for which the IJK of the tip shank most closely approximates the IJK on the TIP command. The IJK on the TIP command is relative to part coordinates so the tip actually selected can vary based on the part alignment.

WARNING: If you want PC-DMIS to recalculate the wrist angles in the face of a change in the workpiece alignment, you must ensure that the **Automatically Adjust Probe Head Wrist** check box is selected before you execute the measurement routine.

The TIP commands in the measurement routine will be updated with the new angles at each execution.

How it Works with Non-Mapped Wrists

For non-mapped wrists, PC-DMIS returns the closest theoretical position. If there is a calibrated tip that matches this theoretical position, PC-DMIS uses that calibrated tip. If PC-DMIS can find calibrated tips that fall within the angle tolerance defined in the **Wrist warning delta** box on the **Part/Machine** tab, it'll use those tips over non-calibrated tips that have closer angle matches. (For more information about the tab, see "Setup Options: Part/Machine tab").

If PC-DMIS cannot find what it thinks is a theoretical match, it gives the error "Illegal TIP command or TIP not qualified".

If PC-DMIS found a good theoretical match but that tip doesn't exist or isn't yet calibrated, it will wait until the machine catches up to the TIP statement, so that no machine motion is in progress. The software then asks you if it should use the closest calibrated tip.

- If you select **YES**, PC-DMIS uses the calibrated tip.
- If you select **NO**, PC-DMIS adds a tip object to match the best theoretical fit and halts, but does not cancel, the execution. A message appears on the PC-DMIS Status bar to press continue when you're done calibrating the new probe. At this point, you should access the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), perform any calibration work as needed, and then click **Continue**.

If you select the **Automatically Adjust Probe Head Wrist** check box and click the **Mark Used** button in the **Probe Utilities** dialog box, PC-DMIS may not select all the probe tips for calibration. For more information about the **Mark Used** button, see the "Mark Used" topic in the "Defining Hardware" chapter.

How it Works with Mapped Wrists

With an infinite wrist that is actually mapped (and therefore calibrated), PC-DMIS automatically returns the best matching tip position and proceeds using that position.

Other Notes

If you want PC-DMIS to continue to use the original tips during any execution of the measurement routine, then clear the **Automatically Adjust Probe Head Wrist** check box.

Ignore CAD to Part

Each time you create an alignment (saved or otherwise), PC-DMIS creates two transformation matrices:

1. **Machine to part transformation matrix** - Computed from the input features' measured values stored internally in machine coordinates.
2. **CAD to part transformation matrix** - Computed from the input features' theoretical values stored internally in CAD coordinates.

When you don't have available CAD data, theoretical data usually comes from the measured values of the learned features. It is difficult to get consistent results using theoretical values. This can happen when some of these values are edited, while others are not.

If you select the **Ignore CAD to Part** check box when you save an alignment, PC-DMIS ignores the *CAD to part* matrix and instead only saves the *machine to part* alignment. All theoretical values are then in the same coordinate system.

Generally, if you are not using CAD data, select this check box.

Effects on CAD Equals Part

If you want to execute an alignment from features measured without CAD data in DCC Mode, and you use the **CAD Equals Part** menu option (or **CAD = Part** button), be sure to select the **Ignore CAD to Part** check box prior to setting the CAD equal to the part. For information on setting the CAD equal to the part, see the "Equating CAD to Measured Part Data" topic in the "Creating and Using Alignments" chapter.

Theoretical Values for Copied Features

- *If you clear this check box and copy a feature to a new location in the Edit window*, the theoretical values of the feature are associated with the alignment at the feature's original location.
- *If you select this check box and copy a feature to a new location in the Edit window*, the theoretical values of the feature are associated with the alignment at the feature's new location.

See "Changing Alignment Nominal Values" in the "Creating and Using Alignments" chapter.

Reset global settings when branching

If you select the **Reset global settings when branching** check box, PC-DMIS resets the global values for state commands after encountering a branching statement. See the list of affected commands below. (For information on branching, see the "Branching by Using Flow Control" chapter.) PC-DMIS does not execute any state commands jumped over because of a branching statement. This allows you to skip over sections of the measurement routine without having these settings change.

For example, suppose your measurement routine has the following:

```
TIP /T1A0B0

MYLABEL=LABEL/
```

Measurements etc...

```
TIP/T1A90B90

GOTO/MYLABEL
```

If you mark the check box, when PC-DMIS reads the `GOTO` statement, it jumps to `MYLABEL`. It then searches up, using the first encountered `TIP/` command: `TIP/T1A0B0`

When returning from subroutines, any global setting modified by the subroutine is not affected by the **Reset global settings when branching** flag. See "Pass Back Settings from Subroutine" for more information.

If you clear the check box, when PC-DMIS reads the `GOTO` statement, it jumps to `MYLABEL`. PC-DMIS does not reset global settings when encountering a branching statement. Instead, it uses the last executed `TIP/` command: `TIP/T1A90B90`

PC-DMIS defaults to marking the check box.

Commands Reset After Branching:

- Start/align
- Recall/align
- Mode/
- Rmeas /
- Workplane/
- Tip/
- Loadprobe/
- Retract/
- Check/
- Touchspeed/
- Movespeed/
- Polarvectorcomp/
- AutoTrigger/
- Triggerplane/
- TriggerTolerance/
- Videosetup/
- Displayprecision/
- ManRetract/
- Scanspeed/
- Prehit/
- Clamp/
- Clearp /
- Format/
- 132column/
- Gaponly/
- Retrolinearonly/

- Probecom/
- Array_indices/
- Fly/
- Positivereporting/
- Ignoremotionerrors/

Move Feature to Reference Plane

If you select the **Move Feature to Reference Plane** check box, PC-DMIS automatically projects the measured feature to the workplane. In general, this works for any of the following feature types as long as they use the best fit functionality internally:

- Circles
- Ellipses
- Lines
- Polygons
- Slots

For a detailed explanation on reference types, see the description of "Reference Feature area" in the "Using the Quick Start Dialog Box" topic in the "Using Other Windows, Editors, and Tools" chapter.

Show Extended Sheet Metal Options

If you select the **Show Extended Sheet Metal Options** check box, PC-DMIS displays all of the available Sheet Metal options in the **Auto Features** dialog box. (See the various extended sheet metal topics in the "Creating Auto Features" chapter.)

For this option, the `ShowExtendedSheetMetal` entry in the PC-DMIS Settings Editor reads 0 or 1. For information on modifying entries, see the "Modifying Setting Entries" appendix.

Fixed Dialog Positions

If you select the **Fixed Dialog Positions** check box, PC-DMIS displays dialog boxes in their default positions. If this check box is cleared, PC-DMIS displays each dialog box where it was last positioned.

The `DialogFixedPosition` entry in the PC-DMIS Settings Editor for this option is set to 0 or 1. For information on modifying entries, see the "Modifying Setting Entries" appendix.

Lock Marked Sets

If you select the **Lock Marked Sets** check box, PC-DMIS will prevent users from accidentally deleting or otherwise modifying the current mark set configurations. PC-DMIS will only allow you to execute and activate marked sets. You must clear this check box to add or delete features from a marked set.

Automatically Scale to Fit

If you select the **Automatically Scale to Fit** check box, PC-DMIS automatically scales the screen every time you measure a feature.

For this option, the `AutoScaleToFit` entry reads 0 or 1 in the PC-DMIS Settings Editor. For information on how to modify entries, see the "Modifying Setting Entries" chapter.

Show Hit Deviations

If you select the **Show Hit Deviations** check box, PC-DMIS will draw an arrow whenever a hit is taken to show the measured value minus the nominal deviation.

Use Circular Moves on Round Features

If you select the **Use Circular Moves on Round Features** check box, and if you are "learning" a measurement routine by taking hits on the part, PC-DMIS automatically places circular move commands into the inside circular features as well as into the outside of circular features. This includes Circles, Cylinders, Cones, and Spheres. This state, however, is only relevant in Learn mode. Once the circular or move commands are present inside features, they remain there unless directly removed by the user.

The `UseCircularMoves` entry in the PC-DMIS Settings Editor for this option is set to either 0 or 1. For information on modifying entries, please view the "Modifying Setting Entries" chapter.

Thickness for Point Only Mode Points

When you select this check box, you can apply thickness to points created with Point Only Mode. Selecting this check box makes the **Thickness** box in this same dialog box available for editing. You can then type a thickness and apply that thickness to points created using Point Only Mode.

For additional information, see "Point Only Mode" and "Thickness".

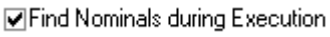
Allow fine tuning of Alignment

Whenever you change an alignment, PC-DMIS asks if it should update the commands below with the changed alignment. If you have this check box selected, and you click the **NO** button at this prompt, PC-DMIS changes the MACHINETOPART transformation if required. If deselected, the MACHINETOPART transformation does not change.

Use CAD Provided IDs for Features

The **Use CAD Provided IDs for Features** check box allows you to import feature IDs from a CAD file. When you select this check box, PC-DMIS automatically inputs the given CAD ID into the **Auto Feature** dialog box when the CAD feature is selected with a left mouse click. If you decide to keep this value, then the feature created will have the given ID.

Find Nominals During Execution



If you select the **Find Nominals during Execution** check box, new nominals are found for both Surface and Vector points during measurement routine execution. See "Execution FindNoms Tolerance" to define the tolerance values PC-DMIS will use. Also, see "Find Nominals Tolerance".

Auto Continue Execution if FindHole Fails

If you select the **Auto Continue Execution if FindHole Fails** check box, PC-DMIS allows you to automatically continue executing a part if the **Find Hole** option found on the **Auto Feature** dialog box fails to find a hole.

In the past, when the **Find Hole** option failed, PC-DMIS prompted you to place the probe in the center of the hole to continue running the measurement routine. However, with the **Auto Continue Execution if FindHole Fails** check box selected, PC-DMIS automatically prints an error message to the report and continues running the remaining measurement routine.

For more specific information on Find Hole, see the "Working with Contact Find Hole Properties" topic in the "PC-DMIS CMM" documentation.

Show Startup Dialog

The **Show Startup Dialog** check box allows you to determine whether or not PC-DMIS shows the **Open File** dialog box each time you start PC-DMIS. This dialog box shows a listing of available measurement routines to open.

When you clear the check box, PC-DMIS will disable this startup dialog.

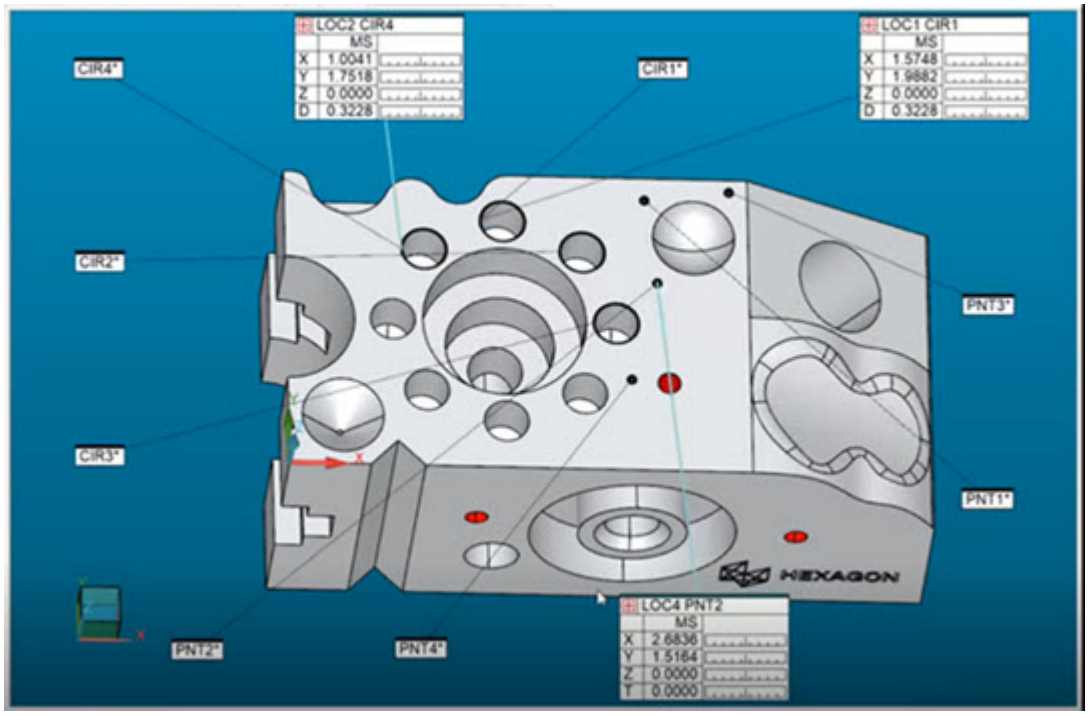
See "Opening Existing Measurement Routines" in the "Using Basic File Options" chapter for additional information on the **Open File** dialog box.

Automatic Label Positioning

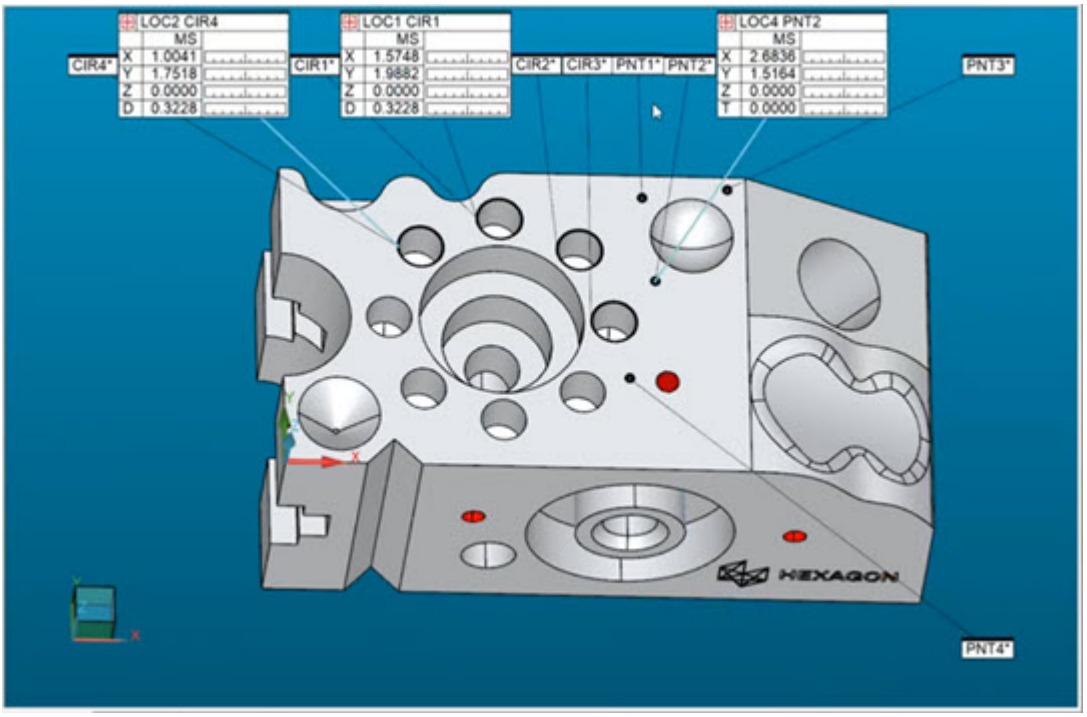
The **Automatic label positioning** check box allows PC-DMIS to automatically position the feature labels. When you select this check box, PC-DMIS repositions the feature ID labels whenever you pan, zoom, or rotate the part model.

You can also enable Automatic Label Positioning from the Graphic Display window. Right-click a feature ID label and select **Automatic Label Positioning**, and then choose one of the following options:

- **Spread-out** - When you select this option, PC-DMIS positions the labels around the Graphic Display window.



- **Near to Edge** - When you select this option, PC-DMIS positions the labels close to the nearest edge of the Graphic Display window.



Animate Probe during Program Mode

The **Animate probe during Program Mode** check box activates probe animation during Program Mode. When you select this check box, the probe will animate the taking of hits in the Graphic Display window as the hits are generated from CAD.

Show Icon in Text Boxes

The **Show icon in text boxes** check box allows you to determine if icons depicting the feature or dimension type should appear in text boxes and inside feature ID labels. The text boxes include feature ID, dimension info, and point info text boxes.

For information on text boxes see "Text Box Mode" in the "Editing the CAD Display" chapter.

Save Measurement Routine on Execute

The **Save Measurement Routine on Execute** check box tells PC-DMIS to automatically save the current measurement routine whenever it's executed.

Use DMIS Button in Edit Window

The **Use DMIS Button in Edit Window** check box determines whether or not PC-DMIS displays the **DMIS Mode** icon on the **Edit Window** toolbar.

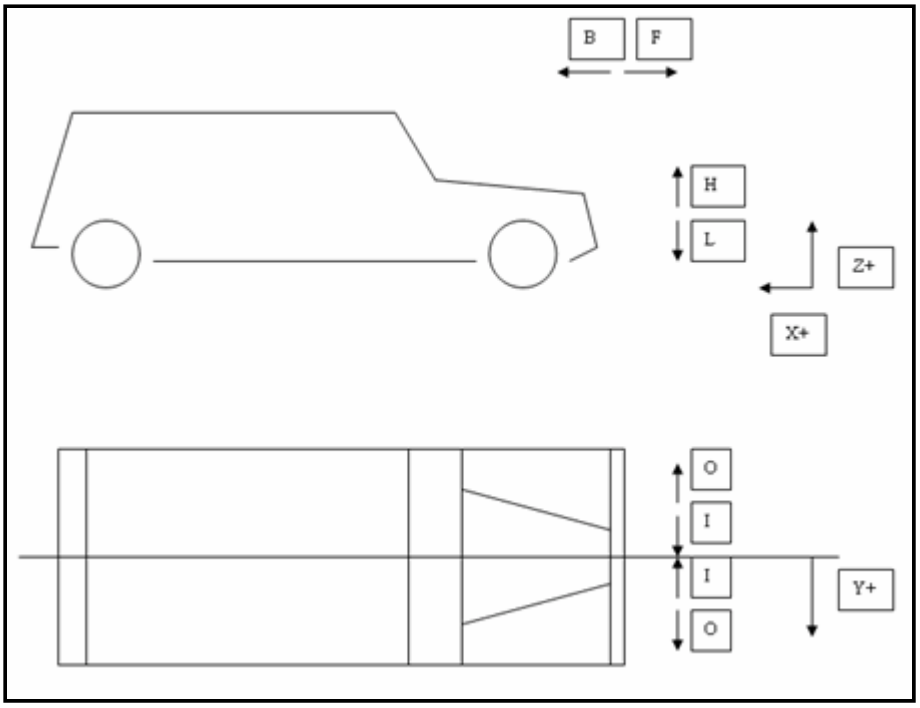
Patch Scans Maintain Last Increment

The **Patch Scans Maintain Last Increment** check box forces each new line of the patch scan to use the last increment from the previous line. If you deselect this check box, the scan will revert to the minimum scan increment when taking the first hit on each line.

Use Automotive Deviation Letters

The **Use Automotive Deviation Letters** check box causes PC-DMIS to add a letter after the deviation number in Location and Position dimension reports. PC-DMIS inserts the letter

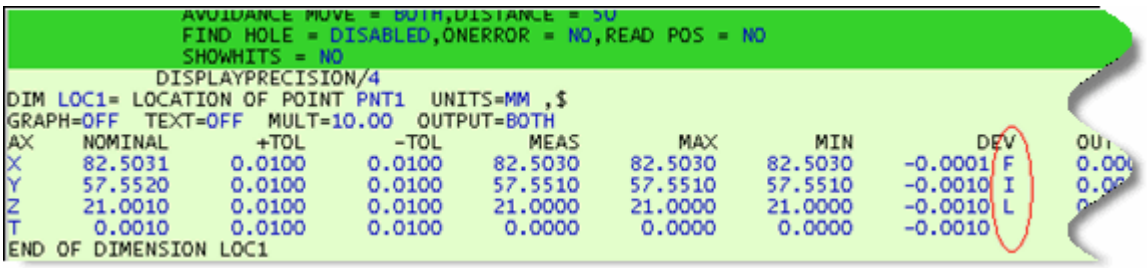
- **F** when the feature deviates in the direction of the front of the car.
- **B** when the feature deviates in the direction of the back of the car.
- **I** when the feature deviates towards the centerline of the car (meaning the car is too narrow).
- **O** when the feature deviates away from the centerline of the car (meaning the car is too wide).
- **H** when the feature deviates towards the top of the car.
- **L** when the feature deviates towards the bottom of the car.



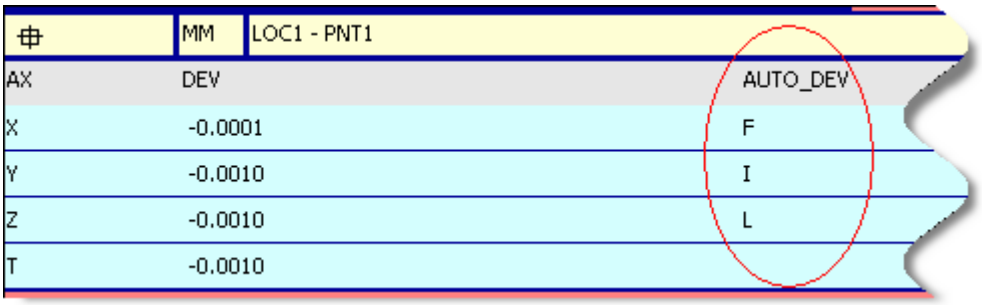
Map of Automotive Deviation Letters

These letters are added to the Location and Position dimensions just following the deviation numbers reported. These deviation letters also appear inside the Report window for the appropriate dimensions.

Automotive Deviation Letters in Command Mode



Automotive Deviation Letters in Report Window



Use Find Nominals Override for Scans

The **Use Find Nominals Override for Scans** check box allows you to override found nominals when searching for nominal values for a scan's measured points when it executes.

> The execution must fail to find at least one of the measured nominal point values for you to override the found nominals.

See "Overriding Found Nominals" in the "Editing a Measurement Routine" chapter.

Use ONLY Priority Surfaces for the Find Nominals During Scans

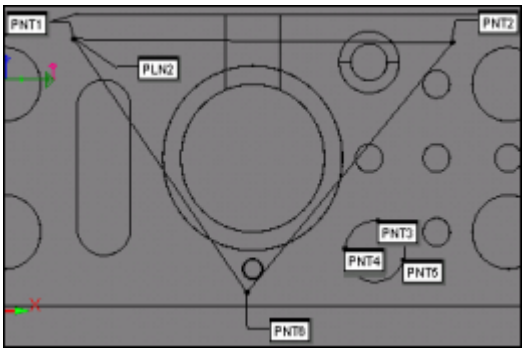
The **Use ONLY Priority Surfaces for the Find Nominals During Scans** check box causes PC-DMIS (during measurement routine execution) to search for nominal values for a scan's measured points on only the priority surfaces set in the **Edit CAD Elements** dialog box.

See "Editing CAD" in the "Editing the CAD Display" chapter.

Display Outline of Plane

The **Display Outline of Plane** check box determines whether or not PC-DMIS, when it creates new measured or constructed plane features, displays actual plane outlines in the Graphic Display window instead of the small symbolic triangle symbol typically used.

For example, if you select this check box and measure three points to construct a plane, the symbolic triangle for the plane is sized so that its vertices are exactly where the measured points are.



Example showing a constructed plane feature when you select the check box.

Whether you select or clear this check box, *existing* outlines aren't redrawn; this only affects planes created from that point. To change existing outlines, you have to do it manually.

To switch the display of existing planes:

1. Access the Edit window.
2. Place it in Command Mode.
3. Go to the command for the plane's display that you want to change. For example:

   ```
   PLN1 = FEAT/PLANE,RECT,TRIANGLE
   ```

4. Press the Tab key until you highlight the `TRIANGLE` or `OUTLINE` field.
5. Press F7 or F8 to switch between the available values for this field.
   - `TRIANGLE` displays the plane as a triangle.
   - `OUTLINE` displays the plane as an actual outline of the points that make up the plane.
   - `NONE` hides that specific drawing of the plane.

   See "Using the Display Area" under the "Constructing a Plane Feature" topic for an example of these display types.
6. Press the Tab key to see the result in the Graphic Display window.

You can also right-click and choose **Edit** to open the plane's dialog box to make the change.

This setting corresponds with the `DisplayOutlineOfPlane` entry in the **Option** section of the PC-DMIS Settings Editor.

Treat Theo Values As if Stored in Part Coordinates

The **Treat theo values as if stored in part coordinates** check box causes PC-DMIS to treat theoretical values as if they're stored in part coordinates. This check box was added for cases where measurement routines perform loops and the alignments are changing within the looping code.

The offset parameters of the LOOP/START command take care of this issue automatically. This is for looping measurement routines that don't use the LOOP/START and LOOP/END commands, such as a WHILE - END/WHILE loop.

Consider this sample pseudo measurement routine:

```
ASSIGN/COUNT=4
ASSIGN/I=1
WHILE/I<4
   LOOPALIGN=START/ALIGN
      ALIGN/TRANSLATION OFFSET,X,50
      END/ALIGN
   MYCIRCLE=MEAS/CIRCLE
      THEO/0,0,0
ASSIGN/I=I+1
END_WHILE/
```

You would expect that each time through the loop the X value would shift by 50 since the alignment is shifting by 50 each time. However, since PC-DMIS doesn't store feature data in part coordinates but in CAD and Machine coordinates, the effect is that the feature doesn't actually move in this situation even though the alignment is changing each time through the loop. This is because the CADTOPART and PARTTOMACHINE transform matrices of the alignment are both altered in the same way, the net effect is that the CADTOMACHINE transform remains unchanged. This means, by default (when this check box is not selected), PC-DMIS ends up measuring in the same spot all four times through the loop.

If PC-DMIS stored coordinates in part coordinates internally, then this would work as expected. This is where this check box comes into play. When you select this check box, PC-DMIS keeps track of the alignment that was used during the initial measurement. On subsequent measurements of a feature, it checks to see if the current alignment differs from the alignment used on the first pass. If it does, then PC-DMIS calculates the differences and shifts the feature by that difference, causing this to function as expected.

Rather than changing how PC-DMIS does things internally, this functionality was provided as a check box to preserve existing measurement routines.

Update Theoretical Values in While Loops

When you use a while loop to measure features multiple times in your measurement routine, during the execution, the software creates multiple copies of those features. The **Update theoretical values in while loops** check box updates the theoretical values for the copied features, so that even if the CAD to part coordinate system changes during execution, the features' CAD values are modified so that they are in the original part coordinate frame used before execution.

Make Variables Globally Visible

Usually, when a subroutine is called, all variables go "out of scope" and are unavailable. The **Make Variables Globally Visible** check box makes any variable globally "visible", or usable, throughout the entire measurement routine.

By default, this check box isn't selected. This means variables in a subroutine won't replace data stored in variables with the same name in the main measurement routine.

```
>ASSIGN/V1=1
>C1=CALLSUB,MYSUB
>.
>.
>.
>SUBROUTINE/MYSUB
>COMMENT/OPER,V1
>END/SUB
```

- If you clear this check box, variables are not globally visible. In the example above, with this check box cleared, the OPER comment shows a value of 0. This is because V1 cannot be seen from inside the subroutine.
- If you select this check box, variables are globally visible. In the example above, with this check box marked, the OPER comment shows a value of 1. This is because V1 can be seen from inside the subroutine.

For information on variables, see the "Using Expressions and Variables" chapter.

Use DMIS Polar Convention

With this option selected, polar angles will follow standard DMIS Polar Convention.

- XY Plane (PLUS Z or MINUS Z) coordinate 'a' is the angle from the x-axis towards the y-axis.

- YZ Plane (PLUS X or MINUS X) coordinate 'a' is the angle from the y-axis towards the z-axis.
- ZX Plane (PLUS Y or MINUS Y) coordinate 'a' is the angle from the z-axis towards the x-axis.

Pass Back Settings from Subroutine

This check box determines whether or not global settings applied within a subroutine are changed permanently for the remainder of the measurement routine when the execution flow returns out of the subroutine.

- If you select this check box, any global settings within the subroutine are "passed back" and used for the remainder of the measurement routine.
- If you clear this check box, any global settings within the subroutine only apply while within that subroutine. When the execution flow returns out of the subroutine, the previous settings are restored.

For example, suppose you have a `SCANSPEED/10` statement before your subroutine and a `SCANSPEED/5` statement somewhere within your subroutine. If you select this check box, when the execution flow leaves the subroutine, the scan speed of 5 is passed back and used for the remainder of the measurement routine. If you don't select this check box, the scan speed will automatically return to its original value of 10 after the execution flow leaves the subroutine.

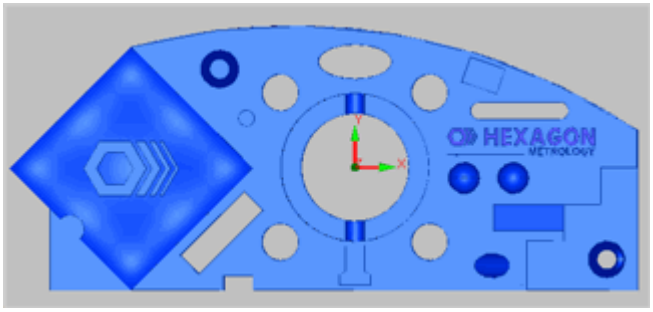
Use Program Layout for Execution

Selecting this check box lets you use the same window layouts at execution time as you have for program time.

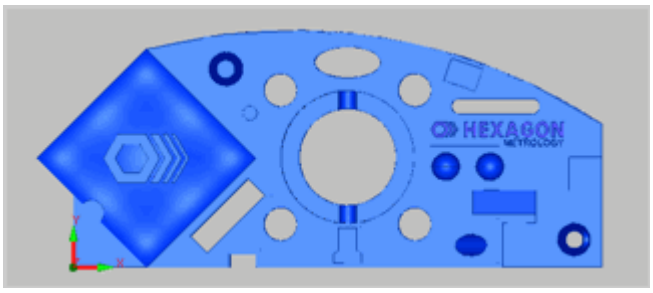
Force Part Alignment in Car Body

The **Force Part Alignment in Car Body** check box forces your alignment to be in car body alignment. If you select this check box, no matter what alignment method you choose, the alignment will always match that of the CAD. Selecting this check box acts as a secondary transformation after the initial alignment.

For example, if you translate to the center hole on the Hexagon test block and select this check box, the origin will still be in the corner, but when you report on that hole, it will be perfect since you used it to set your origin.



Example showing the alignment before using the check box.



Example showing the alignment after using this check box.

Keep Existing Dimension's Axes

The **Keep Existing Dimension's Axes** check box keeps existing axes for dimensioned features from getting updated even if you switch a dimensioned feature to a different feature type. If you don't select this check box, the related dimension axes will update according to the selected feature.

By default, this is not selected.

Select Alignment to Export

This check box determines whether PC-DMIS displays a dialog box to select the alignment to export as a .gds file.

> If you select this check box, PC-DMIS lets you choose an alignment to export to a .gds file.
>
> If you don't select this check box, PC-DMIS exports the last used alignment automatically.

For more information on how to export to a .gds file, see the "Exporting to a Gds File" topic in "Using Advanced File Options".

Gap Only

This check box sets the default value for the **Gap Only** check box used in new Location dimensions for the current measurement routine.

- If you select **Gap Only** then, when the measurement routine next starts, the `GAPONLY/ON` command gets added into the Edit window. In addition, the **Feature Location** dialog box (**Insert | Dimensions | Location**) for Location dimensions automatically has its **Gap Only** check box selected by default whenever you create a new Location dimension.
- If you deselect **Gap Only** then the **Feature Location** dialog box has its **Gap Only** check box also deselected whenever you create a new Location dimension.

Show tracker parameters in offline

If you use a portable Leica Tracker device in Online mode to generate feature commands, PC-DMIS automatically inserts the following information into the Edit window inside those feature commands:

- **RMS** - Root Mean Squared value of each hit.
- **Probe Type** - The type of probe you use to measure the feature.
- **Time Stamp** - The time that PC-DMIS executed or learned the feature. PC-DMIS only updates this time stamp when it actually measures a feature in Online mode.
- **Environmental Conditions** - Information such as temperature, pressure, and humidity.

In Offline mode, PC-DMIS behaves differently. These Leica Tracker items only appear after you select the **Show tracker parameters in offline** check box, and they only appear for new feature commands inserted into the measurement routine after you selected this option. Previously-measured features remain unaffected except for a permanent structure change that added an empty Tracker Parameter group into each feature command.

If you select this check box, PC-DMIS permanently changes your measurement-routine structure for inserted feature commands regardless of whether or not you later clear this check box. For example, if you clear this check box after you've already used it for some features, newly-inserted features still contain a Tracker Parameter group, although that group does not contain any group items.

For information on where and how these items appear in the feature commands, see the PC-DMIS Portable documentation.

Update Report During Execution

This determines whether PC-DMIS creates the report in the Report window during measurement routine execution.

- When marked, the Report window sends update requests during execution, and PC-DMIS generates the report as the execution progresses.

- When cleared, the Report window does not send update requests during or at the end of execution unless you do any of the following:
  - From the Report window's toolbar, click **Redraw the report**. This generates the report in the Report window with the latest data.
  - Choose **File |Printing| Report Window Print**. This generates a report if you configured a file or printer output.
  - Insert a `PRINT/REPORT` command. This generates a report at the time it executes the command if you configured a file or printer output.

Update Summary Mode During Execution

This determines whether or not the Summary Mode should be updated during execution or after execution has been completed. When disabled, the Edit window's Summary Mode will not send requests to be updated during execution.

If marked, information that is typically updated during execution are measured values, deviations, out of tolerance and so forth on dimensions.

Clearing this check box may help speed up report generation.

Suppress Vision Load Probe Dialogs

This setting is only used with optical machines. See the "Available Vision Setup Options" topic in the "PC-DMIS Vision" documentation.

Focus Along Camera Vector

This setting is only used with optical machines. See the "Available Vision Setup Options" topic in the "PC-DMIS Vision" documentation.

Auto Edge Strength

This setting is only used with optical machines. See the "Available Vision Setup Options" topic in the "PC-DMIS Vision" documentation.

Clear Input on Execution

When checked, every time the measurement routine is executed, any input field that is displayed will be empty. If this feature is not checked, any input field that is displayed will display the contents from the previous input.

Do Not Display Plane

When you add measured or constructed plane features, PC-DMIS has the option of displaying a shaded plane as either an outline of the hits comprising the plane or as a triangle. You can mark this check box to hide the drawn shaded plane at creation so that it doesn't clutter up the Graphic Display window. This only hides the drawn plane; PC-DMIS still creates the actual plane feature. This setting corresponds with the `DoNotDisplayPlane` entry in the **Option** section of the PC-DMIS Settings Editor.

Print Background Colors

This **Print Background Colors** check box determines the default state of the **Print background colors** check box in the **Output Configuration** dialog box. See "Output to the Default Printer" for additional information. This setting corresponds to the `PrintBackgroundColors` entry in the **Printing** section of the PC-DMIS Settings Editor.

Use Legacy Profile 2D

Selecting this option toggles between the legacy version of Profile 2D (selected) and the latest version of Profile 2D (de-selected).

The legacy Profile 2D has the options of **Matching Edge and Under Scan** which are no longer available in the latest version of Profile 2D.

For more information on using the Legacy Profile 2D functionality, see the vision topic "**Vision Profile 2D**".

Move to focus position

This check box controls the stage motion at the end of a focus measurement.

- If this check box is selected, the machine will move back to the focus position at the end of the focus scanning motion.
- If this check box is cleared, the machine will not move to the focus position if there is a move or measurement following the focus measurement. Instead, it will move from the top of the focus scan directly to the next machine position.

ClearanceCube uses Tip Vector for Start/End Face

If this check box is selected, new features inserted into the measurement routine will use the probe tip's vector as the default ClearanceCube start face and end face. Otherwise, the ClearanceCube start face or end face is determined by the feature's vector.

Start Points Always Track Execution

The **Start Points Always Track Execution** check box enables start points to always track the execution of a measurement routine. Specifically, it determines if you created the start point or if PC-DMIS created it because the **Cancel** button in the **Execution** dialog box was selected.

If the check box is selected:

- If you select **Cancel** to stop the execution, PC-DMIS adjusts the insertion point in the Edit window to be at the lowest arm number start point that it inserted.
- If you do not select **Cancel**, and the execution successfully completed, PC-DMIS checks the start points. If PC-DMIS inserted them because **Cancel** was previously selected, PC-DMIS clears them. In this case, it restores the insertion point back to where it started from before you executed the measurement routine.
- If you select **Cancel**, PC-DMIS inserts the start point.

If the check box is cleared:

- If you select **Cancel** to stop the execution, and PC-DMIS inserted the start points, PC-DMIS adjusts the insertion point to be at the lowest arm number start point that it inserted.
- If you do not select **Cancel**, and the execution successfully completed, PC-DMIS checks the start points. If PC-DMIS inserted them because **Cancel** was previously selected, it clears them. In this case, the insertion point is restored back to where it started from before you executed the measurement routine.
- If you select **Cancel**, PC-DMIS inserts the start point if there are no start points on an arm, or if it inserted them because **Cancel** was previously selected. If you inserted the start point, PC-DMIS does not adjust it.

For more information about start points, see "Setting Start Points".

Use Scanning Strategies for QuickAlign

The **Use Scanning Strategies for QuickAlign** check box determines whether or not the DCC Mode features during a QuickAlign operation are created using the scanning strategy that is defined in the .ipd (inspection plan defaults) file. However, if you modify that Auto feature type and choose a new strategy prior to generating the alignment, it remembers and uses the newer strategy and no longer uses the defaults in the .ipd file.

The scanning strategies are used for a QuickAlign operation, if the following conditions are true:

- This check box is marked.

- The features in the manual alignment are Auto features.
- The probe type is a scanning probe.

If this check box is unmarked, PC-DMIS uses the default touch trigger strategy.

For information on strategies, see "Working with Measurement Strategies" in the PC-DMIS CMM documentation.

For information on QuickAlign, see "About QuickAlign" in the "Creating and Using Alignments" chapter.

Open report file when generated

The **Open report file when generated** check box determines whether or not any generated .pdf, .rtf, or xls report files are opened once the measurement routine finishes executing. If marked, PC-DMIS marks the **Show Report** check boxes on both the **Report** and **Excel** tabs of the **Output Configuration** dialog box. For more information on the **Output Configuration** dialog box, see the "Printing the Report Window" topic and its sub topics.

Send skipped items to statistics

The **Send skipped items to statistics** check box determines whether or not features that are skipped during execution are sent to any defined statistics database or output files. By default, this is marked and the software sends the skipped features to the database and output files.

Looped Analysis Commands use same CAD Alignment

You may have one or more `ANALYSISVIEW` commands inside of a `LOOP/START` and `LOOP/END` command block. This check box determines if those `ANALYSISVIEW` commands all use the same alignment.

If marked, the commands use the same alignment.

If cleared, the commands may use different alignments based on your measurement routine.

Suppose your measurement routine has this sequence of commands:

```
A1 ALIGNMENT

CIR1

CIR2

CIR3

DIM1

DIM2

DIM3

LOOP/START 1 to 3

    A2 ALIGNMENT

    ANALYSISVIEW, DIM1, DIM2, DIM3

LOOP/END
```

Suppose the loop runs three times. Also, suppose that with each loop, A2 changes the origin to reference a different circle. So that:

- Loop #1 - A2 uses CIR1
- Loop #2 - A2 uses CIR2
- Loop #3 - A2 uses CIR3

If you mark this check box, the `ANALYSISVIEW` command uses the alignment that points to CIR1. It does this for all three iterations.

If you clear this check box, the `ANALYSISVIEW` command uses the three different alignments.

Auto Insert Temperature Compensation in New Routine

You can choose to either automatically insert the `TEMPCOMP` temperature compensation command into a measurement routine or not insert it. For details, see "Automatic Insertion of Temperature Compensation Command".

Use Characteristic ID Naming

This check box determines how PC-DMIS names features that you import from embedded CAD GD&T callouts (PMI).

- If you mark this check box, PC-DMIS uses the characteristic IDs from the native CAD model for the feature naming. PC-DMIS also assigns the characteristic IDs to features and dimensions during the OCR feature creation workflow.
- If you clear this check box, PC-DMIS uses its default feature naming.

For information on importing CAD GD&T callouts, see "Importing CAD GD&T Callouts" under "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.

Don't confuse this characteristic ID naming with characteristic IDs used by **Show Characteristic IDs** on the **Graphic Items** toolbar. The IDs from the toolbar come from an entirely different source. For information on the **Graphic Items** toolbar, see "Graphic Items Toolbar" in the "Using Toolbars" chapter.

Use Measurement Strategy Widget

This check box determines whether the measurement strategy widget appears when you create a QuickFeature or a feature for a GD&T callout. By default, PC-DMIS marks this check box for CMM configurations. For portable configurations, PC-DMIS clears this check box.

- If you mark this check box, the widget always appears when you click on the feature.
- If you clear this check box, the widget does not appear when you click on the feature. PC-DMIS still creates the feature, but it does not show the widget.

For more information on this measurement strategy widget, see "Using the Measurement Strategy Widget" in the "Creating Auto Features" chapter.

Use Measurement Strategy Editor

This check box determines whether to enable or disable the Measurement Strategy Editor (MSE) application. By default, PC-DMIS marks this check box for CMM configurations. For portable configurations, PC-DMIS clears this check box.

- If you mark this check box, the MSE appears when you select the **Edit | Preferences | Measurement Strategy Editor** menu.
- If you clear this check box, the **Edit | Preferences | Measurement Strategy Editor** menu turns gray and you can no longer select it.

For more information on the Measurement Strategy Editor, see "Using the Measurement Strategy Editor" in this chapter.

Show Home Page

This check box determines whether to show the Home page if you don't load any measurement routines. If you clear this check box, the Home page does not appear, and it shows the gray screen from versions before 2019 R2.

For more information on the Home page, see "The Home Page" in the "Navigating the User Interface" chapter.

Show Peek Window

This check box determines whether to enable or disable the Peek window. The Peek window is enabled by default.

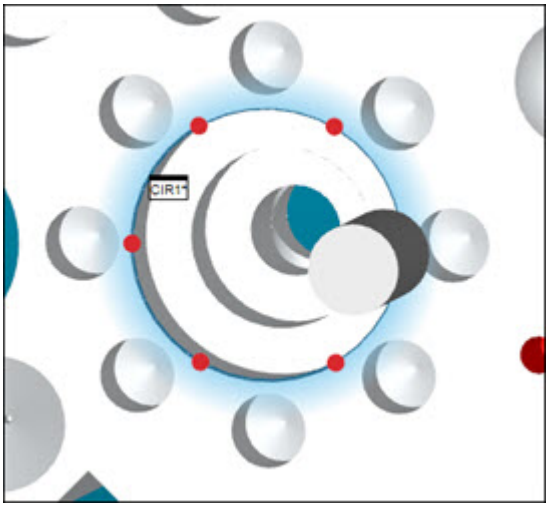
- If you mark this check box, the Peek window displays a command's information when you point to the command in the Edit window in Summary Mode.
- If you clear this check box, the Peek window does not display a command's information when you point to the command in the Edit window in Summary Mode.

For more information on the Peek window, see "Using the Peek Window" in the "Using the Edit Window" chapter.

Auto Scale manual feature during execution

This check box determines whether PC-DMIS automatically rotates the view of the part and zooms to manual features in the Graphic Display window during execution. The rotation is slightly isometric. This option essentially provides a visual guide of what feature you need to measure next with your device. This only works for features that follow a `MODE/MANUAL` command.

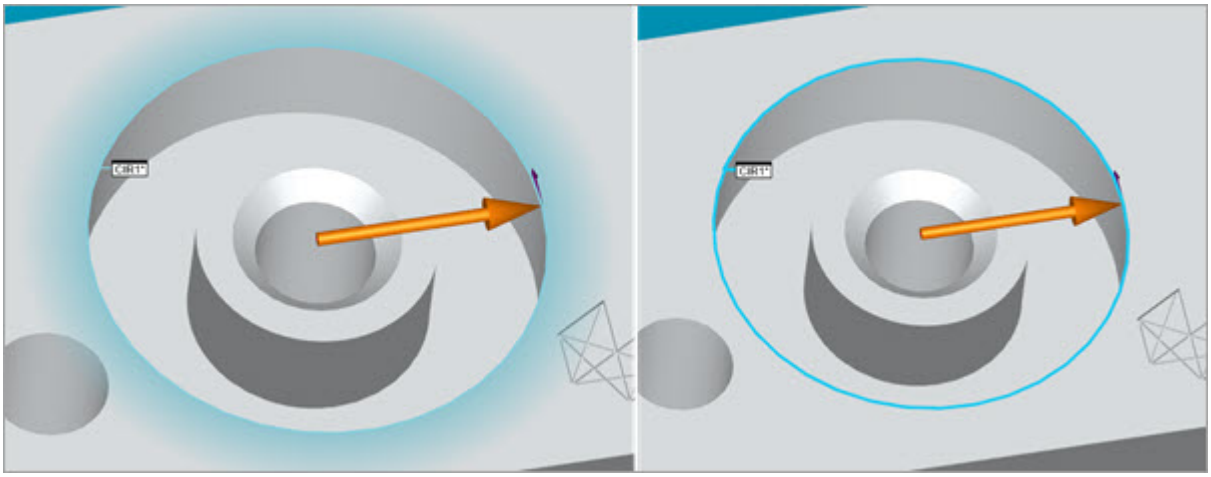
For example, an Auto Circle on the top surface of the Hexagon block looks like this during execution:



For the rotation and scaling to function, you must have a part alignment before manual features.

Enable 2D feature glow

This check box allows you to turn the glow effect for 2D features on and off. PC-DMIS marks this check box by default to allow the glow effect for 2D features.



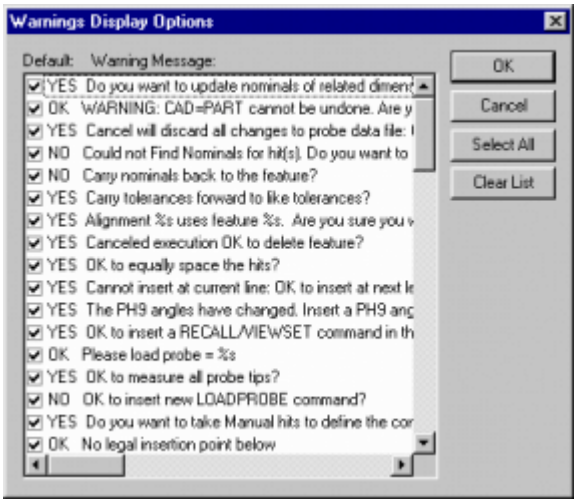
Examples that shows the Enable 2D feature glow option set to On (left) and Off (right).

Here's how this option works:

- If you mark this check box and click the **OK** button on the dialog box, PC-DMIS applies this effect on 2D features only in the currently opened measurement routine. PC-DMIS uses the `Enable2DFeatureGlow` setting in the Settings Editor on all new measurement routine you create to determine whether to apply the effect or not. For details, see "Enable2DFeatureGlow" in the PC-DMIS Settings Editor documentation.
- If you mark this check box and click the **Defaults** button on the dialog box, PC-DMIS applies this effect on 2D features in the currently opened measurement routine and any new measurement routines you create. PC-DMIS sets the Settings Editor entry `Enable2DFeatureGlow` to **TRUE**.
- If you clear this check box and click the **OK** button on the dialog box, PC-DMIS does not apply this effect on 2D features in the currently opened measurement routine. PC-DMIS uses the `Enable2DFeatureGlow` setting in the Settings Editor on all new measurement routines you create to determine whether to apply the effect or not.
- If you clear the check box and click the **Defaults** button, PC-DMIS does not apply this effect on 2D features in the currently opened measurement routine or any new measurement routines you create. PC-DMIS sets the Settings Editor entry `Enable2DFeatureGlow` to **FALSE**.

Warnings

The **Warnings** button on the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) displays the **Warnings Display Options** dialog box.



Warnings Display Options Dialog Box

You can use this dialog box to tell PC-DMIS to re-display warnings that you have already switched off as well as change the default actions for warning messages that are turned off. By default, this dialog box is empty. When a warning message appears, PC-DMIS gives you the option of not receiving that warning again. When you choose to not receive the warning, the warning ends up in this dialog box.

If you need information on specific warnings, consult the "Warn(Option)" topic in the **Option** section of the PC-DMIS Settings Editor documentation.

Warning messages only appear in this interface in the **Warnings Display Options** dialog box once you have turned them off. If no warnings have been turned off, the dialog box remains empty.

To start receiving a warning message again:

1. Access the **Warnings Display Options** dialog box. It shows any warnings that you have turned off.
2. Select the warning and clear its check box.
3. Click **OK** to remove the warning. PC-DMIS starts displaying that warning message again.

The **Clear List** button clears all the check boxes for all items, thereby returning PC-DMIS to its default mode of displaying all the warning messages.

To change the default action for a warning message:

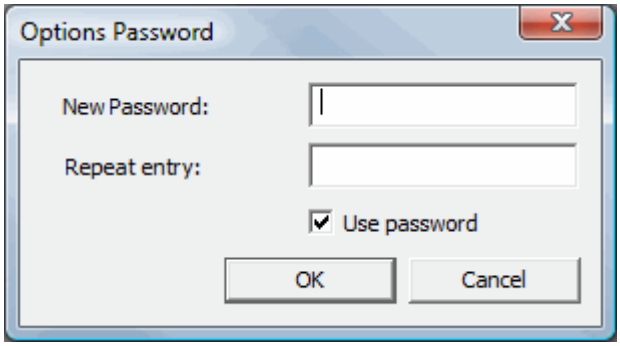
1. Make sure you have clicked the **Don't ask me again** check box when the warning appears. This causes it to go to the **Warnings Display Options** dialog box.
2. Access the **Warnings Display Options** dialog box. It shows any warnings you have turned off.
3. Double-click on the warning message whose default action you want to change. PC-DMIS displays the warning so that you can select the new default action.
4. Select the new default action to update the list with the new action.
5. Click **OK** to save your choice.

Password

The **Password** button allows you to password protect access to the **Setup Options** dialog box.

To password protect your setup options:

1. Access the **Setup Options** dialog box (**Edit | Preferences | Setup**).
2. Select the **General** tab.
3. Click the **Password** button. The **Options Password** dialog box appears.

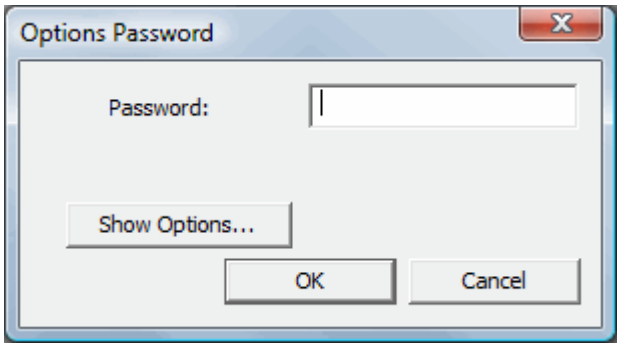


Options Password Dialog Box

4. In the **New Password** box, type the desired password.
5. In the **Repeat entry** box, re-type the same password to confirm your entry.

6. Select the **Use Password** check box.
7. Click **OK**.

The next time you attempt to access the **Setup Options** dialog box, you will be asked to type the password before proceeding. The password is case-sensitive.



Even if you don't have password access to edit the settings, you can always view the available settings by using the **Show Options** button. This displays the **Setup Options** dialog box, but you are unable to make any changes.

> The Protected Mode password supersedes and overwrites any other password you have defined. This means if you have password protection enabled for the **Setup Options** dialog box or for an .ipd file, you need to use the password defined when Protected Mode was enabled.

Other Edit Boxes for the General tab

Use the other boxes on the **General** tab to edit the following options:

Scale factor box

The **Scale factor** box scales the measured data by whatever factor is entered. For example, if a circle is measured and has a diameter of 1.0 inches and the scale factor is 0.95, then the measured value will be reported as 0.95 inches.

Find nominals tolerance box

This box becomes available when you first select the **Find Nominals** check box. See "Find Nominals".

Use the **Find nominals tolerance** box to type the amount of tolerance that PC-DMIS uses when finding the nominals. The default is 10 mm.

When you initially set this value and click **OK** (and whenever you perform a find nominals operation in learn mode), PC-DMIS checks this value against the active tip's radius. If this

value is less than the tip's radius, PC-DMIS will change it to match the diameter of the active tip.

Show deviations tolerance box

Use the **Show deviations tolerance** box to type the amount of tolerance that PC-DMIS uses when PC-DMIS is showing hit deviations. This option is only available when you first select the **Show Hit Deviations** check box.

Deviation arrow multiplier box

The **Deviation arrow multiplier** box is only available when you first select the **Show Hit Deviations** check box. An arrow appears in the Graphic Display window marking each hit taken, showing the deviation. The larger the value you enter in this box, the greater the size of the arrow.

Thickness box

The **Thickness** box works with the **Thickness for Point Only Mode Points** check box. If you select the **Thickness for Point Only Mode Points** check box, the thickness in this box will be applied to each point created by Point Only Mode.

See "Point Only Mode" and "Thickness for Point Only Mode Points" for additional information.

Execution Area

Use the items in the **Execution** area of the **General** tab to edit the following options:

Any order execute tolerance

In the **Any order execute tolerance** box, define the tolerance that the hit must be within for PC-DMIS to measure that feature when executing your measurement routine in any order execute mode.

If your initial hit is more than the specified tolerance away from the feature, PC-DMIS searches backwards and forwards through the measurement routine for the closest feature whose initial hit matches your initial hit and executes that feature instead. For more information, see "Any Order Execution" in the "Using Advanced File Options" chapter.

PC-DMIS stores the tolerance value with your measurement routine. This enables you to have different search zones for different measurement routines if desired.

Execution FindNoms tolerance

In the **Execution FindNoms tolerance** box, type the amount of tolerance that PC-DMIS uses when finding the nominals during measurement routine execution.

This box becomes available when you first select the **Find Nominals during Execution** check box. For more information, see "Find Nominals During Execution".

Support execution time display and Inspect

If you select the **Support execution time display and Inspect** check box, PC-DMIS displays the remaining execution time for a measurement routine or mini routine in the **Execution** dialog box. The time is recorded only for the DCC portion of the execution.

The Inspect software uses this option to display how long it takes to execute a routine. This check box also give Inspect the ability to execute subsets of routines known as "dimension sets". (Inspect is a software built specifically for operators. It runs PC-DMIS in the background and is available with PC-DMIS.)

This check box is not available for Portable machines (the execution time applies only to DCC measurements).

Every time you execute a measurement routine or mini routine, PC-DMIS records and stores its execution time. The next time you start the execution, the last time that was recorded appears in the **Execution** dialog box. This way, if you add or delete features, or decide to decrease or increase speeds in your measurement routine or mini routine, PC-DMIS records the change in execution time.

PC-DMIS records the execution time in the *<name of measurement routine>*.MiniRoutines.xml file. This file is located in the same folder as the measurement routine. If a measurement routine is executed when the file does not exist, then PC-DMIS creates the file. For more information about this file, see "Mini Routines".

You can decide to not record and display the execution time in the **Execution** dialog box. For example, the execution time for a machine that operates only in Manual Mode does not need to be recorded and displayed. To not record and display the execution time, clear the **Record and display execution time** check box.

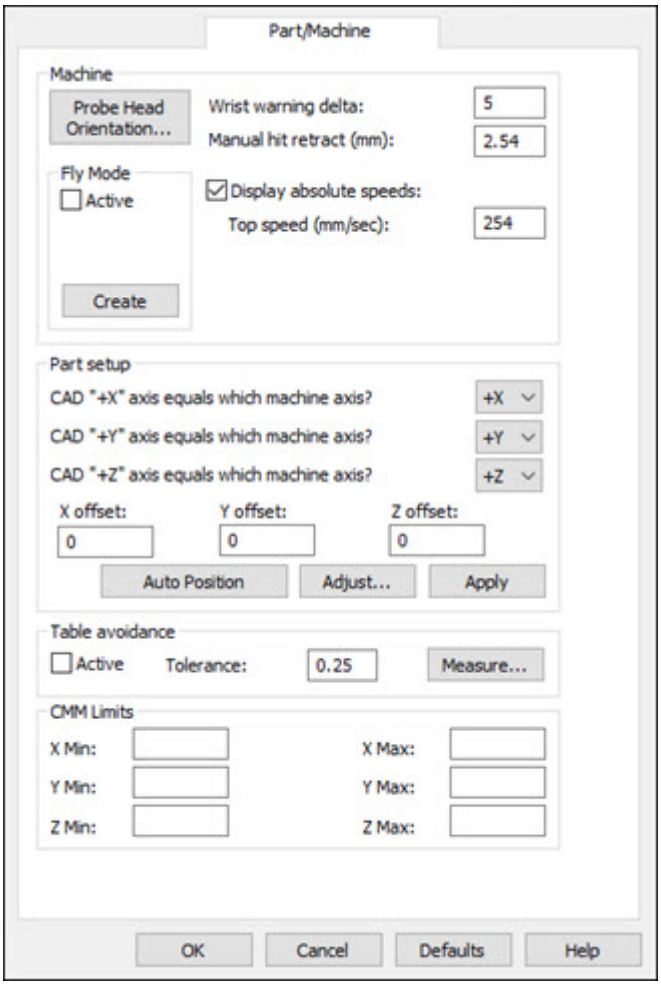
Add to execution time (seconds)

The time required to complete a measurement routine may include other actions in addition to executing the measurement routine or mini routine, such as printing the measurement results. If you select the **Add to execution time (seconds)** check box and enter the time in seconds, PC-DMIS adds the seconds to the actual execution time. The next time you execute the measurement routine or mini routine, the remaining time that appears in the **Execution** dialog box includes this additional time.

Show offline animation speed slider

If you select the **Show offline animation speed slider** check box, PC-DMIS displays the **Offline Animation Speed** slider in the **Execution** dialog box. This slider lets you adjust the execution speed of the measurement routine in Offline Mode as a percentage of the maximum animation speed value.

Setup Options: Part/Machine Tab



Setup Options Dialog Box - Part/Machine Tab

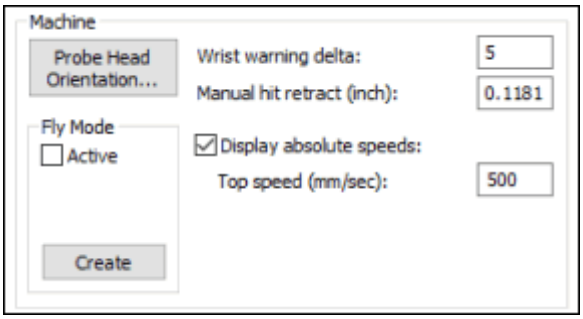
The **Part/Machine** tab lets you define the part setup on the CMM (or Machine) by changing the CAD axes relationship to the machine axes. To access this option, click on the **Part/Machine** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**).

This capability is needed when you use PC-DMIS to create a measurement routine, and the CAD coordinate system differs from the CMM part setup.

A part is set up on the machine with its X+ CAD axis pointing in the same direction as the CMM's Z+ axis. The Z+ CAD axis is pointing in the same direction as the CMM's X- axis. You can use this function to create the proper relationships.

To equate the CAD setup to the part's setup, select the appropriate axes from the drop-down lists. Once you establish this relationship, it becomes easier to create a measurement routine for the part because PC-DMIS properly displays the probe in relation to the part.

Machine area



Machine Area

Probe Head Orientation button

The **Probe Head Orientation** button allows you to configure the probe head wrist AB angles for multiple arms.

To configure the AB wrist angles for multiple arms:

1. Access the **Setup Options** dialog box (**Edit | Preferences | Setup**).
2. Select the **Part/Machine** tab.
3. Click the **Probe Head Orientation** button. The **Probe Head Wrist Angle Configuration** dialog box appears.
4. Select the appropriate axes for the AB angles for the Arm1 and Arm2 arms (if applicable).
5. Click the **OK** button.

Fly Mode area

The **Fly Mode** area provides a way to move the probe around the part in a smooth and non-stopping motion. You must insert a move point into the measurement routine before you use this option. (See "Inserting a Move Point Command" in the "Inserting Move Commands" chapter.) The `Fly` command can only be inserted before or after any feature in the measurement routine.

To use the Fly Mode:

1. Scroll to the location in the Edit window where you want the Fly command.
2. Select the **Active** check box.
3. Click the **Create** button.

   The Fly mode is automatically placed in the current measurement routine at the designated location. The Edit window command line for this option reads: `FLY/TOG1` . The TOG1 toggle field switches between either `ON` or `OFF` . If `ON` , PC-DMIS activates the command. If `OFF` , PC-DMIS skips the command.

In DMIS mode, the Edit window command line that activates the command is: `FLY/1`

If you have selected an illegal position, a message informs you that PC-DMIS cannot insert the command at the current line. The message then asks if it should insert the command at the next legal position.

- If you click the **Yes** button, PC-DMIS moves the `Fly` command down to the end of the current feature in the Edit window.
- If you click the **No** button, PC-DMIS cancels the `Fly` command and returns you to the **Part/Machine** tab.

Wrist warning delta box

This is a numerical value that defines the minimum change in the wrist angle required before PC-DMIS proposes a change to the current wrist position. This affects users with a DCC CMM with all auto wrists.

Manual hit retract box

The **Manual hit retract** box allows you to type the retract distance the CMM automatically travels after you take a manual hit. This value corresponds to the unit type of the measurement routine (inches or millimeters).

If you change the manual hit retract value from either the default or from the last-used value, PC-DMIS places a retract command (shown as `MANRETRACT/` and then the value) in the current measurement routine's Edit window at the cursor's location. As manual hits are taken, the CMM retracts by the new value stated on this command.

For example, if you change the manual hit retract value from the default value of 0.1 to 0.003, PC-DMIS shows the command `MANRETRACT/0.003` in the Edit window.

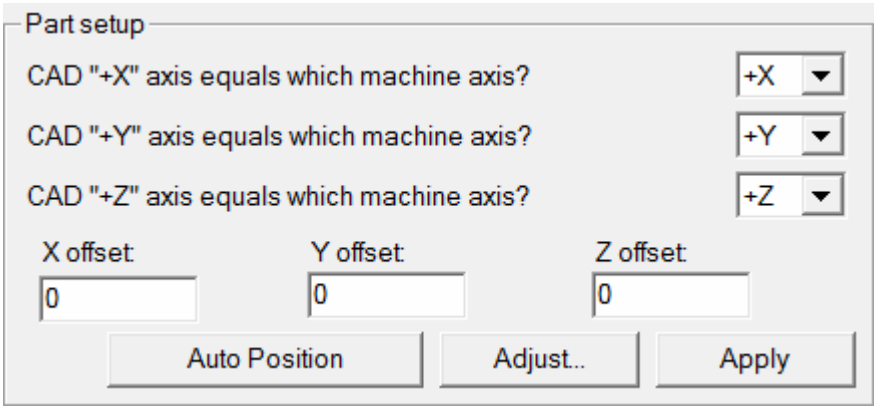
Display absolute speeds box

If you select the **Display absolute speeds** check box, PC-DMIS displays the speed values in other dialog boxes as an absolute value instead of a percentage. This value corresponds to the unit type of the measurement routine (inches or millimeters).

Top speed (mm/sec) box

The **Top speed (mm/sec)** box allows you to reset the top speed that the machine can travel. The value that you specify cannot exceed the machine's designated top speed. The value works with the **Move speed** option.

Part Setup area



Part Setup Area

The **Part Setup** area of the **Part/Machine** tab is useful when you create a measurement routine and the CAD coordinate system differs from the CMM part setup.

CAD "+X" axis equals list

This drop-down list allows you to set the relationship between the CAD X+ axis and the machine axis.

CAD "+Y" axis equals list

This drop-down list allows you to set the relationship between the CAD Y+ axis and the machine axis.

CAD "+Z" axis equals list

This drop-down list allows you to set the relationship between the CAD Z+ axis and the machine axis.

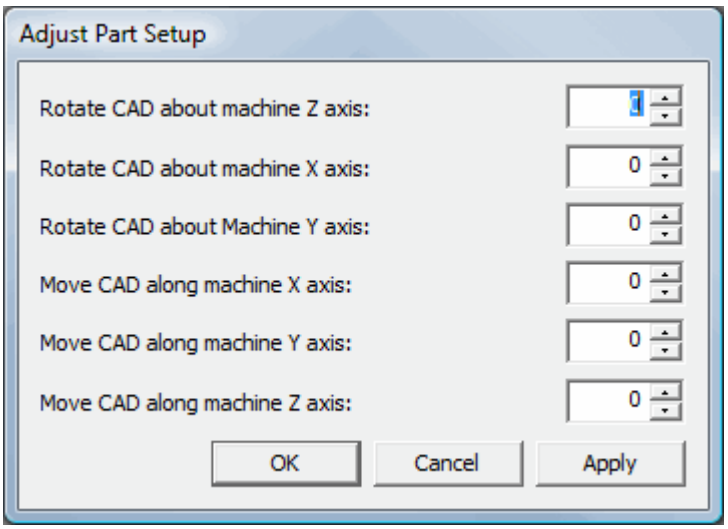
X offset box, Y offset box, Z offset box

These boxes allow you to enter the distance that PC-DMIS offsets the CAD drawing along the X, Y, or Z axis. PC-DMIS shifts the CAD drawing along the X, Y, or Z axis the specified distance. For example, if you enter in .5 in the X field, the entire CAD display in the Graphic Display window shifts the distance by .5 in the X direction.

All features created in the measurement routine *do not* shift along the axis with the CAD drawing.

Adjust button

The **Adjust** button opens the **Adjust Part Setup** dialog box.



Adjust Part Setup Dialog Box

You can use this dialog box to *rotate* or *move* the CAD in increments about and along the machine's XYZ axes.

- *If you're rotating the CAD,* then the values are in angle degrees. A value of 360 has the same effect as 0.
- *If you're moving the CAD,* then the values are in the units of measurement that the measurement routine is in. For example, a value of 2 would mean 2 inches or 2 millimeters, depending on the units of measurement used by your measurement routine.

To adjust the CAD along or about an axis:

1. Click on the appropriate box.
2. Type a new value. PC-DMIS dynamically shows the adjustment in the Graphic Display window.
3. Click **OK** to accept the values and close the dialog box.

PC-DMIS maintains this adjustment unless you re-import the part's CAD model.

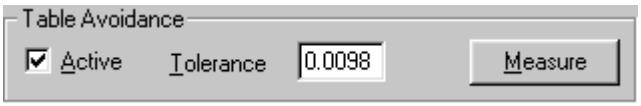
Auto Position button

The **Auto Position** button positions the part onto the graphical representation of the machine's table. Auto position best guesses where to position the part on the graphical representation. You can either let PC-DMIS determine where to put the part in relation to the CMM by using this function, or you can type your own positioning using the XYZ offset fields. (See "Defining Machines" in the "Defining Hardware" chapter for more information on setting up the graphical representation of the CMM.)

Apply button

When you click the **Apply** button, PC-DMIS immediately applies any changes made in the X, Y, or Z Offset fields. It also shifts the drawing along the appropriate axis (axes) while keeping the dialog box open.

Table Avoidance area



The **Table Avoidance** area of the of the **Part/Machine** tab allows PC-DMIS to ascertain if the probe comes in contact with the table (or set plane) while in DCC Mode.

Click the **Measure** button and PC-DMIS asks you to take a hit where the table surface is to be defined. This location defines the threshold for the Z axis. The tolerance field defines a location in the positive Z direction for positive values and in the negative Z direction for negative values, relative to the set plane.

- If a move goes beyond the indicated tolerance, PC-DMIS will display an error message indicating the potential danger.
- If a wrist rotation is requested that will pass through the defined zone, PC-DMIS will display a message alerting you to the error.

Select either **Cancel** or **Continue** to terminate or finish the operation.

A tolerance field of .25 will alert PC-DMIS to avoid the indicated threshold plus the tolerance value. If the threshold is on the table surface, PC-DMIS will alert you if the probe tip comes within one quarter inch (or millimeter depending on the set unit type) of the table.

The **Table Avoidance** option is available with certain interface types only while in DCC Mode.

CMM Limits area

The values in the **CMM Limits** area of the **Part/Machine** tab in the **Setup Options** dialog box enables PC-DMIS to avoid collisions with the legs on Bridge-type CMMs during the calibration of tactile probes.

When you calibrate a tactile probe with long extensions, collisions may occur when the probe rotates in the different tip positions. To avoid collisions, you can enable the Leg Collision check. PC-DMIS runs the check online and just before the probe calibration routine. The Leg Collision check can automatically add safety movements.

When PC-DMIS connects to a supported Hexagon CMM (common controllers and FDC controllers), it automatically gets the CMM limits. You can manually edit the CMM limits. This can be useful if you do not have a Hexagon CMM. In addition, you might want to

reduce the limits to avoid collisions with objects located within the machine volume (tool changers, fixtures, and so on).

To run the Leg Collision check, do the following:

1. If you need to modify or add the limits of the machine, enter the values in the **X Min**, **Y Min**, **Z Min**, **X Max**, **Y Max**, and **Z Max** boxes in the **CMM Limits** area.

2. Calibrate the probe tips. For details, refer to the "Calibrating Probe Tips" topic in the "Setting Up and Using Probes" chapter in the PC-DMIS CMM documentation.

To reset any modified values and obtain the CMM limits from your Hexagon CMM again, do the following:

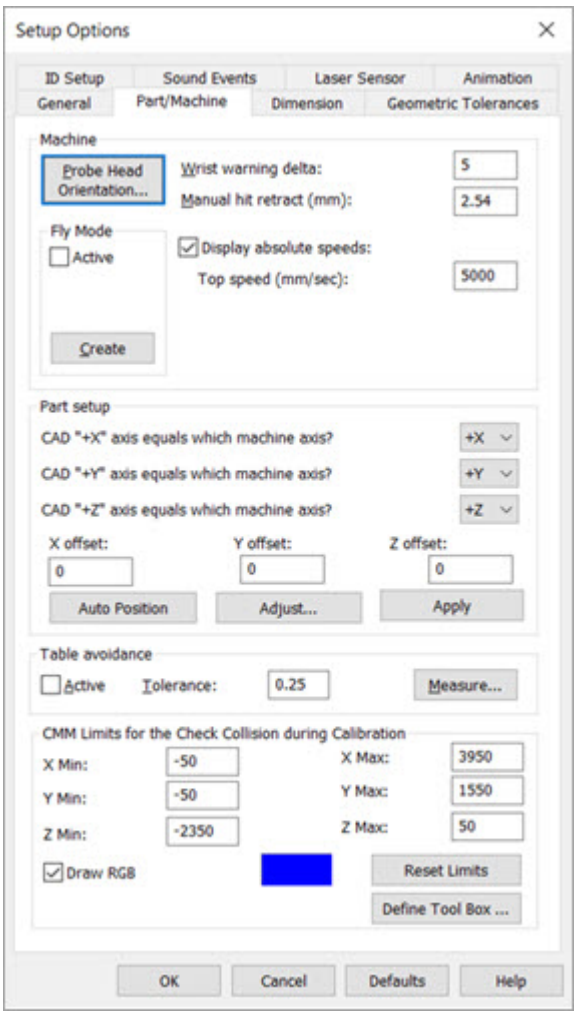
1. Type 0 into the **X Min**, **Y Min**, **Z Min**, **X Max**, **Y Max**, and **Z Max** boxes.

2. Click **OK** to close the **Setup Options** dialog box.

3. Restart the measurement routine, or restart your machine.

Horizontal/Dual Arms

When you define the probe head orientation for a horizontal arm in the **Part/Machine** tab of the **Setup Options** dialog box, the **CMM Limits** area shows additional options:



Setup Options Dialog Box

This example is for Arm1, but you can press F5 to obtain the same for Arm2 in a dual arm system with the Arm2 active. This allows PC-DMIS to read the CMM limits from the Arm2 controller and populate the dialog box with the proper limits.

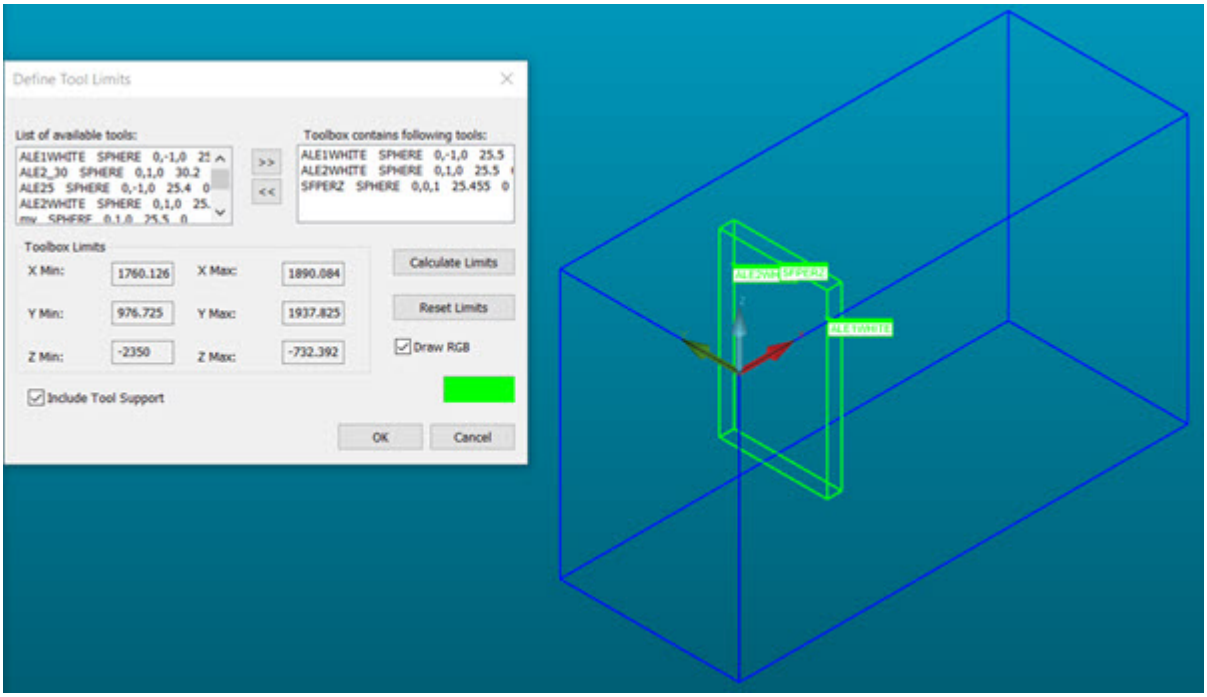
You can reduce the number of CMM limits read in from the computer, for example, to exclude the area where a probe changer is mounted (typically in the -X direction). Press the Reset Limits button to reset the CMM limit values back to the original values from the controller.

PC-DMIS can draw the CMM limits in the Graphic Display window so you can get a visual idea of the setup.

The **Define Tool Box** button allows you to define an additional area that is taken into account by the legs collision engine during the calibration. This is seen in the case of horizontal/dual arm machines where the software uses multiple gauges oriented in Y and Z. In that case, the risk of collisions with the gauges and their support is different compared to Bridge machines, where typically there's only one gauge oriented in Z+.

You can define that additional area by adding the gauges to a list. PC-DMIS then calculates the limits for the tool box to include them all. The defined tool box may or may

not include the tool supports, in which case PC-DMIS extends the calculated tool box using the positions of the selected gauges to the lower limit of the CMM (-Z).



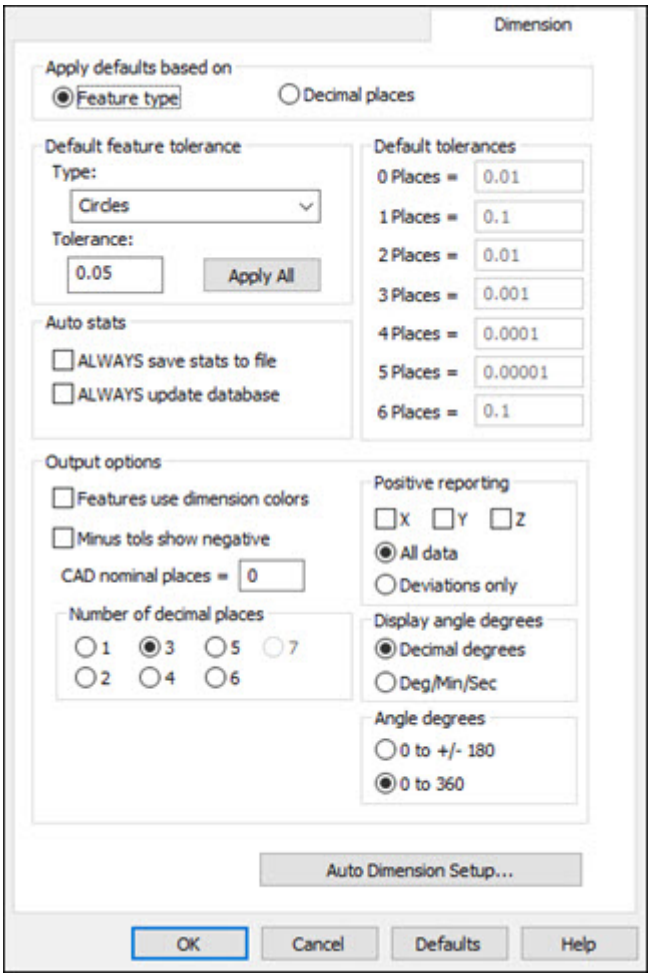
Example showing the defined tool box for a Horizontal or Dual Arm system.

Default button

The **Default** button allows you to update the default settings to several of the **Part/ Machine** tab's options on the **Setup Options** dialog box (**Edit | Preferences | Setup**). When you create a new measurement routine, it reflects any changes made to the accessible parameters, *only* if you click the **Default** button. If you click the **OK** button (without clicking the **Default** button) the defined parameters will only apply to the active measurement routine and will not affect PC-DMIS's entries. The default values are stored in a JSON file. You can update these parameters in the appropriate dialog box or you can use the PC-DMIS Settings Editor application. See the "Modifying Setting Entries" chapter.

If you change any of the parameters and you click the **Default** button, PC-DMIS updates the JSON file. This defines the current values as the default values.

Setup Options: Dimension tab



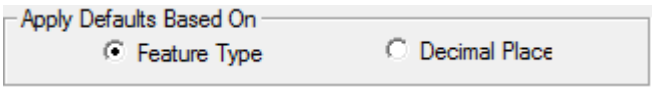
Setup Options Dialog Box - Dimension Tab

The **Dimension** tab lets you access the dimensional printout parameters.

To access the **Dimension** tab:

1. Access the **Setup Options** dialog box (**Edit | Preferences | Setup**).
2. Click the **Dimension** tab.

Apply Defaults Based On



Apply Defaults Based On Area

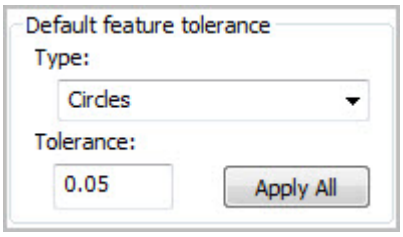
The **Apply Defaults Based On** area lets you apply default dimension tolerances either based on feature type or on the number of decimal places displayed.

- Selecting the **Feature Type** option enables the **Default Feature Tolerance** area, allowing you to define dimension tolerances based on individual feature types. Whenever PC-DMIS creates a dimension automatically, either because the **Auto**

option is selected in the **Location** dialog box (**Insert | Dimension | Location**) or because you used the **Auto Dimension Setup** button, the default dimension tolerance associated with that feature type is used. See the "Default Feature Tolerance" topic.

- Selecting the **Decimal Place** option enables the **Default Tolerances** area, allowing you to define dimension tolerances based on the number of decimal places. This is the old way that PC-DMIS determined dimension tolerances. See "Default Tolerances" topic.

Default Feature Tolerance



Default Feature Tolerance Area

The **Default Feature Tolerance** area lets you define default dimension tolerances based on individual feature types. This area is enabled when you select the **Feature Type** option from the **Apply Defaults Based On** area.

This capability only applies for legacy dimensions.

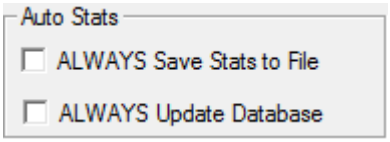
Whenever you manually create a location dimension or whenever PC-DMIS creates any dimension automatically (due to the **Auto Dimension Setup** button), the default dimension tolerance associated with that feature type is used, unless that same feature type already exists in the measurement routine. In that case, PC-DMIS instead uses the tolerance for the existing feature for any new automatic dimensions of that same feature type. This way you only need to modify the tolerance once for a feature type and PC-DMIS uses that same tolerance for other features of the same type throughout the rest of your measurement routine.

Type list
This list displays all the features for which you can set a default tolerance.

Tolerance box
This box defines the default tolerance used for the feature in the **Type** list.

Apply All button
This button lets you apply the current tolerance value in the **Tolerance** box *to all feature types*.

Auto Stats



Auto Stats Area

PC-DMIS can save statistical data to an output file and then update a database from that file. You define where to save stats in the `STATS/ON` command. For more information on the `STATS/ON` command, see the "Tracking Statistical Data" chapter.

The check boxes in the **Auto Stats** area control whether PC-DMIS shows certain prompts when it executes `STATS/ON` commands.

ALWAYS Save Stats to File

- If you clear this check box, PC-DMIS asks if it's okay to save the statistics. This occurs when execution starts and your measurement routine has a `STATS/ON` command in it.
- If you mark this check box, PC-DMIS saves the statistical data without asking you.

ALWAYS Update Database

- If you clear this check box, PC-DMIS asks if you want to update the database. This occurs after PC-DMIS saves the statistical data to the file and is about to update a database.
- If you mark this check box, PC-DMIS updates the database whenever it collects and saves statistical data without asking you.

For details on the locations of PC-DMIS files, see "Understanding File Locations".

Features Use Dimension Colors

The **Features Use Dimension Colors** check box tells PC-DMIS to color features that have an associated dimension. Features are drawn in the Graphic Display window with the same colors that the dimension uses to indicate deviation from the theoretical values.

By default, this only functions with location dimensions. You can use this with non-location dimensions as well by using the `NonLocationDimsSetFeatColor` entry in the **Dimensions** section of the PC-DMIS Settings Editor.

CAD Nominal Places =

The **CAD Nominal Places =** box allows you to enter a numerical value that defines how many decimal places PC-DMIS will use before rounding when it uses CAD data. For example, if a CAD circle has a diameter of 3.9995, and the value is set to 3, PC-DMIS will round its value to 4.000. This option only affects the way PC-DMIS interprets CAD data in the SHEET METAL MEASUREMENT mode. If the value is set to 0, PC-DMIS will not round off any values.

Minus Tols Show Negative

The **Minus Tols Show Negative** check box controls whether or not PC-DMIS displays the minus tolerances of the dimensions with a minus sign. For example, if the dimension is specified as 5.0000 +0.3 (upper tolerance), -0.2 (lower tolerance) and you mark this check box, PC-DMIS displays the dimension line as:

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
Y	5.0000	0.3000	-0.2000	5.0000	0.0000	0.0000	0.0000	0.0000

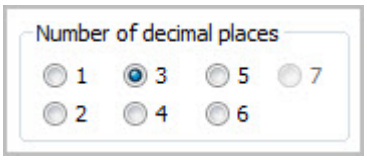
If you do not select this check box, PC-DMIS displays the same dimension line as:

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
Y	5.0000	0.3000	0.2000	5.0000	0.0000	0.0000	0.0000	0.0000

This check box does not affect how PC-DMIS stores the values or how PC-DMIS uses the values in calculations. It only controls how PC-DMIS displays the values according to your preference. This check box is not selected by default.

If you do not select this check box, you can still have a tolerance with a minus sign in front of it. Under usual circumstances, your lower and upper limits are on opposite sides of the nominal. When you do not select this check box, PC-DMIS displays all values as positive values. However, if the upper limit and lower limit are both greater than the nominal - for example, your values actually read +TOL 0.03 and –TOL –0.02 before using this check box - then your minus tol value shows a positive value when you select this check box.

Number of Decimal Places



Number of decimal places Area

The **Number of decimal places** area controls the number of decimal places displayed in the Edit window and the inspection report.

Choose the desired option to determine the number of decimal places displayed.

Each time you change this option in a measurement routine, PC-DMIS places a command: `DISPLAYPRECISION/ #` into the measurement routine. This specifies the precision PC-DMIS displays at this section of the measurement routine. If you do not use this command, then PC-DMIS automatically uses the default value. If you do use this command, the precision stays as specified unless changed by another instance of the command.

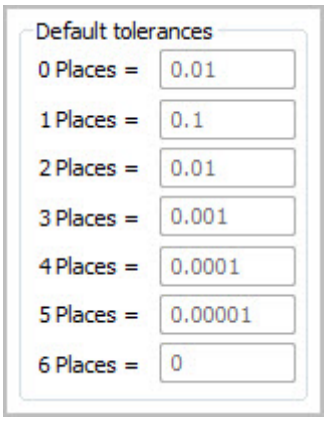
- Millimeter measurement routines allow up to six decimal places.
- Inch measurement routines allow up to seven decimal places.

For example, if you choose **6** for an inch measurement routine and you click **OK**, PC-DMIS inserts this command into the Edit window: `DISPLAYPRECISION/6`

This causes PC-DMIS to display any dimensions listed below this command with six decimal places, as illustrated in this Location dimension:

```
DISPLAYPRECISION/6
DIM LOC2= LOCATION OF POINT PNT1 UNITS=IN ,$
GRAPH=OFF TEXT=OFF MULT=10.00 OUTPUT=BOTH HALF ANGLE=NO
AX      NOMINAL        +TOL        -TOL         MEAS
           DEV          OUTTOL
X     2436.427000    0.001970    0.001970   2436.427000
      0.000000      0.000000 ----#----
Y      229.658000    0.001970    0.001970    229.658000
      0.000000      0.000000 ----#----
Z      849.992000    0.001970    0.001970    849.992000
      0.000000      0.000000 ----#----
T        0.000000    0.001970    0.001970      0.000000
      0.000000      0.000000 ----#----
END OF DIMENSION LOC2
```

Default Tolerances



Default Tolerances Area

The **Default Tolerances** area, enabled when you select the **Decimal Places** option from the **Apply Defaults Based On** area, lets you define the default tolerances PC-DMIS uses when you modify a dimension's nominal value in the Edit window. The default tolerance used is based on the number of decimal places in the nominal value.

For example, if you modify a nominal value to read 6.250, PC-DMIS sets the + and - tolerance to the **3 Places =** default tolerance value, because three decimal places were used. If you instead entered 6.25, PC-DMIS would set the + and - tolerance to the **2 Places =** value, because two decimal places were used.

0-6 Places = boxes
These boxes allow you to set different default tolerances that PC-DMIS applies when you define a nominal value with 0 to 6 decimal places.

Positive Reporting

The check boxes related to the **Positive Reporting** area control the reporting of features to the negative side of the origin. With positive reporting, features that would normally have negative values are always printed with positive values.

- The **X**, **Y**, and **Z** check boxes determine on which axis (or axes) PC-DMIS displays positive numbers.
- The **All Data** option tells PC-DMIS to flip the measured and nominal values of the selected axis (or axes) and to display any negative numbers for nominal axes as positive numbers.
- The **Deviations Only** option tells PC-DMIS to flip only the deviations of the selected axis (or axes) when the nominal values of axes are negative numbers.

When you select these check boxes, PC-DMIS inserts a `POSITIVEREPORTING` command into the measurement routine at the current cursor position. This command has the following format in the Edit window:

```
POSITIVEREPORTING/ a, b, c, TOG1
```

Where:

a = X when you select the X check box, or blank if you clear the X check box.

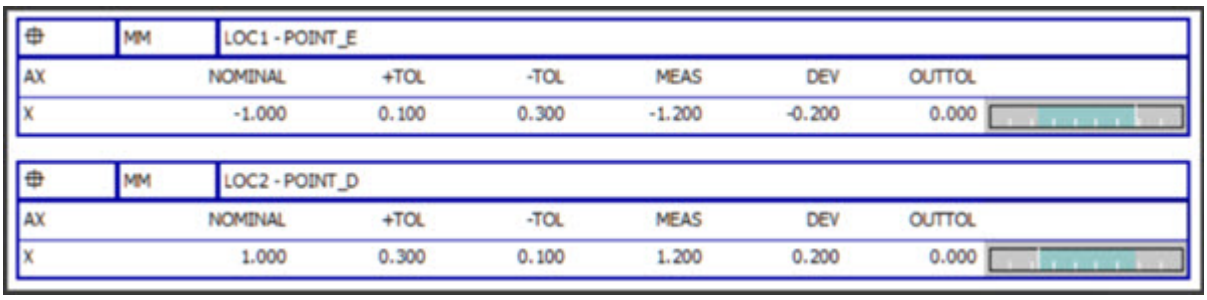
b = Y when you select the Y check box, or blank if you clear the Y check box.

c = Z when you select the Z check box, or blank if you clear the Z check box.

TOG1 = ALLDATA or DEVONLY depending on whether you selected the **All Data** or **Deviations Only** option.

The X, Y, and Z direction may each have positive reporting turned on in any combination of the three. You can use multiple `POSITIVEREPORTING` commands within the same measurement routine, and PC-DMIS displays any dimensions that are in the measurement routine using the `POSITIVEREPORTING` command that precedes the dimensions. If no `POSITIVEREPORTING` command exists in the measurement routine, PC-DMIS reports all dimensions with the options turned OFF in the X, Y, and Z directions.

This is an example of reporting Points E and D with the +0.3/-0.1 tolerance and deviation of 0.2, with Positive Reporting turned off:



	MM	LOC1 - POINT_E					
AX		NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X		-1.000	0.100	0.300	-1.200	-0.200	0.000

	MM	LOC2 - POINT_D					
AX		NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X		1.000	0.300	0.100	1.200	0.200	0.000

Example showing a report with Positive Reporting turned off.

You can see POINT E has a negative nominal value because it is X-. Often, you don't want to see the negative values because on a print it would not show a negative value. Also, as in this case if the tolerances are not the same, you also need to flip the tolerances so Point E is +0.1/-0.3 and point D is +0.3/-0.1 which again can be confusing because the drawing would not show this.

If you turn Positive Reporting on, the same report looks like this:

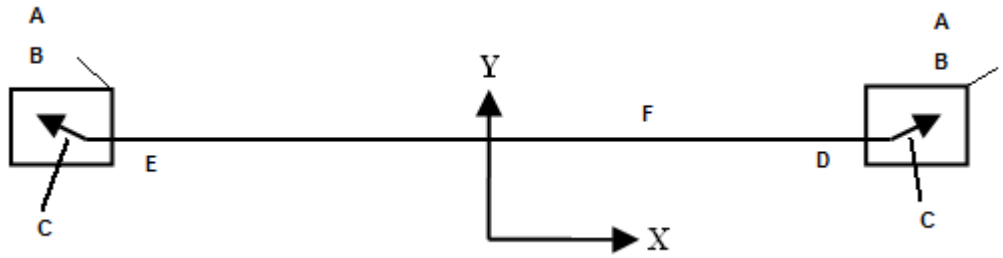
	MM	LOC1 - POINT_E					
AX		NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X		1.000	0.300	0.100	1.200	0.200	0.000

	MM	LOC2 - POINT_D					
AX		NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X		1.000	0.300	0.100	1.200	0.200	0.000

Example showing a report with Positive Reporting turned on.

You can see from this second example that POINT D is unaffected (it was already positive) but POINT E has changed. The nominal is now a positive number and the tolerances have switched.

You can see below how this occurs in the actual coordinate system:



Example showing the effect of Positive Reporting on tolerances.

A = 0.3 +tol in X

B = 0.1 -tol in X

C = 0.2 deviation in X

D = Point 1

E = Point 2

F = 1.0 nominal

Purpose of Positive Reporting

Positive reporting allows you to report features in a symmetrical manner, so that, regardless of which side of the origin the feature exists, deviations *away* from the origin are considered positive, and deviations *toward* the origin are considered negative.

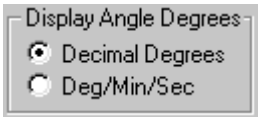
Thus, in the diagram above, both Point1 and Point2 display positive deviations when positive reporting in the X axis. This also means, however, that:

- plus tolerances are applied *away* from the origin.
- minus tolerances are applied *toward* the origin.

Migrating Older Measurement Routines

When migrating measurement routines from an older version of PC-DMIS (such as 3.7) to version 4.x or higher, you may have a problem with the tolerance band being flipped when you use Positive Reporting in your dimensions and you use variables to insert the tolerance values into the PLUS and MINUS tol fields. The `UseLegacyPositiveReporting` entry, located in the **Option** section of the PC-DMIS Settings Editor, lets you use the legacy positive reporting in these cases. This makes it so that PC-DMIS does not flip the tolerance values.

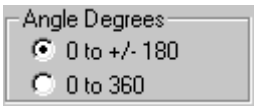
Display Angle Degrees



Display Angle Degrees Area

The **Display Angle Degrees** area allows you to display angle dimensions using decimal degrees or by degree/minute/second. When you select the desired option, PC-DMIS changes any PA (Polar Angle) Location axes and Angle dimensions to display the selected option.

Angle Degrees

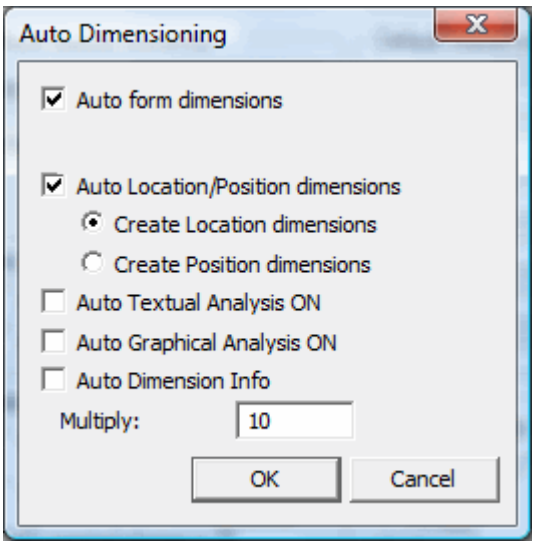


Angle Degrees Area

The **Angle Degrees** area also allows you to display the angle dimension from 0° to +/- 180° or 0° to 360° for the current measurement routine. PC-DMIS changes any PA (Polar Angle) Location axes and Angle dimensions to lie within the selected angle degree range. If you want to change the default value for all future measurement routines, use the `AngleRange0To360` entry in the **Option** section of the PC-DMIS Settings Editor.

Auto Dimension Setup

The **Auto Dimension Setup** button from the **Dimension** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) opens the **Auto Dimensioning** dialog box.



Auto Dimensioning Dialog Box

This dialog box provides you with several options that allow you to determine whether or not PC-DMIS automatically dimensions features immediately after creating them and how it should create those dimensions.

To enable or disable automatic dimension creation, clear or mark the following check boxes:

Auto Form Dimensions
This check box determines whether or not PC-DMIS automatically creates form dimensions for feature types that have a form dimension.

Feature	Associated Form Dimension
Circle	Roundness
Cylinder	Roundness
Cone	Roundness
Sphere	Roundness
Plane	Flatness
Line	Straightness

Auto Location/Position Dimensions
This check box determines whether or not PC-DMIS automatically creates location or position dimensions for feature types that have a location or position dimension.

Create Location Dimensions
If you select **Auto Location/Position** dimensions, this option button tells PC-DMIS to create those dimensions as Location dimensions.

Create Position Dimensions
If you select **Auto Location/Position** dimensions, this option button tells PC-DMIS to create those dimensions as Position dimensions.

Auto Textual Analysis ON
This check box controls whether or not PC-DMIS automatically creates textual analysis of the dimension. It is considered ON if the box is marked. See "Analysis Settings" in the "Using Legacy Dimensions" chapter and "Analysis" in the "Inserting Report Commands" chapter.

Auto Graphical Analysis ON
This check box controls whether or not PC-DMIS automatically creates a graphical analysis of any dimension that is created with **Auto Create Dimensions** or **Auto Roundness**. See "Analysis Settings" in the "Using Legacy Dimensions" section and "Analysis" in the "Inserting Report Commands" chapter.

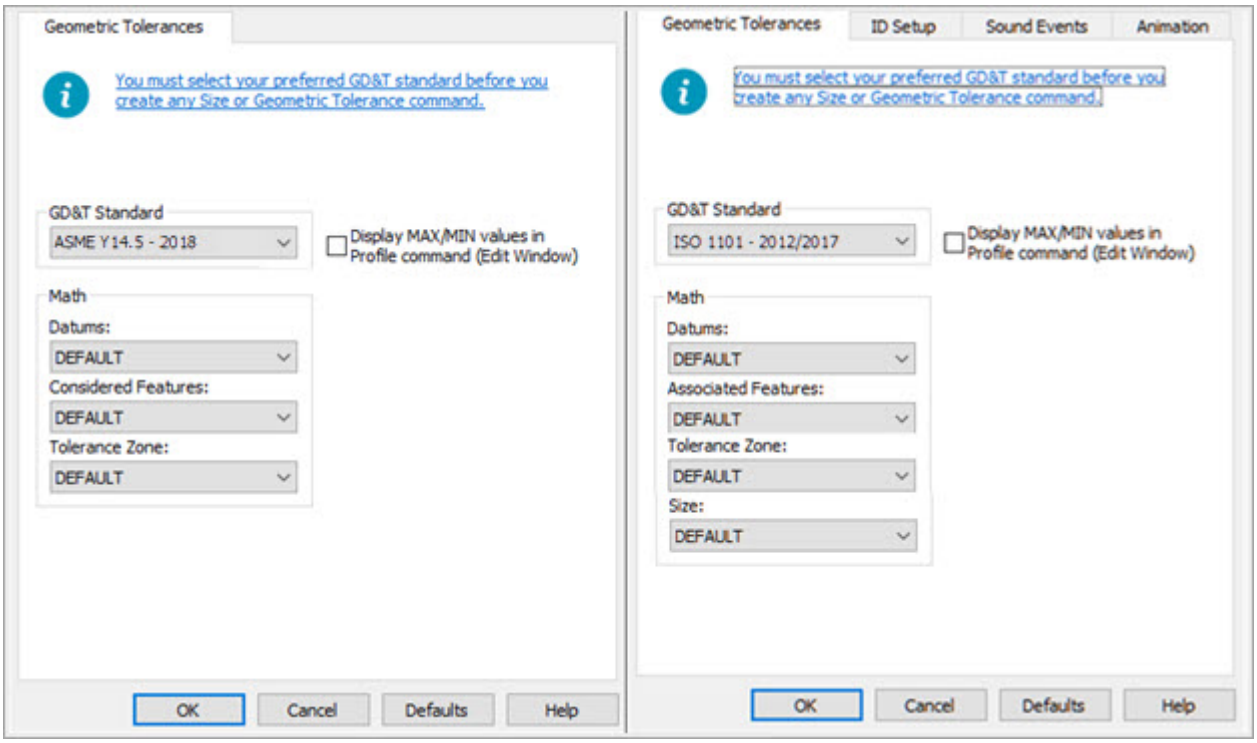
Auto Dimension Info
This check box controls whether or not PC-DMIS automatically creates a **Dimension Information** check box for any dimension created with the **Auto Create Dimensions** check box or the **Auto Roundness** check box. See "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter for information on how to set the default formats for this Dimension Info boxes.

Multiply
The **Multiply** value is a scaling factor that magnifies the arrow and tolerance zone for the graphical analysis mode. If a value of 2.0 is entered, PC-DMIS scales the arrow two times the graphical image.

The **Multiply** box is used for viewing purposes only, and is not reflected in the text printout.

PC-DMIS creates dimensions as either legacy dimensions or Geometric Tolerance dimensions. Mark or clear the **Use Legacy Dimension** check box from the **Insert | Dimensions** submenu to set the appropriate dimension type.

Setup Options: Geometric Tolerances tab



Setup Options dialog box Geometric Tolerances tab for ASME (left) and ISO (right)

The **Geometric Tolerances** tab controls several defaults for the creation of new geometric tolerance commands. For more information on geometric tolerance commands, see the "Using Geometric Tolerances" chapter.

Dialog Box Options

GD&T Standard - This area shows the available standards that you can set as the default standard that you want new geometric tolerance commands to use. When you create a new geometric tolerance command, or access the **Geometric Tolerance** dialog box, PC-DMIS automatically selects the standard that you set here (**ASME Y14.5** or **ISO 1101**).

Starting with PC-DMIS 2023.2, you are no longer able to toggle the selected GD&T standard from within Geometric Tolerance commands. For details, see "Comparison to Past Practices - Referencing the GD&T Standard" section of the "Structuring your Measurement Routine for Geometric Tolerances" topic.

This tab is the only place where you can change the GD&T standard. Starting with 2023.2, PC-DMIS no longer attempts to convert between ASME and ISO standards. In the past, this was somewhat possible due to the limited level of support of ISO functionality in PC-DMIS. With the expanded development of ISO capabilities in PC-DMIS, divergence from the ASME standard has become increasingly significant. In many cases, no equivalency exists between the standards, causing conversion to be impossible.

The GD&T standard affects Size or Geometric Tolerance commands in your measurement routine. The standard you select does not affect legacy dimensions. You can change the selected GD&T standard at any time from the **Geometric Tolerances** tab of the **Setup Options** dialog box. However, because the setting applies to the entire measurement routine, depending on the measurement routine's content at the time of change, it may behave differently.

- If there are no Geometric Tolerance or Size commands in the measurement routine, you can change the GD&T standard from the **Geometric Tolerances** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**), and it becomes the default standard going forward. PC-DMIS updates the program header to reflect the GD&T standard that is now referenced and uses the standard for all new Geometric Tolerance and Size commands that you create.
- If the measurement routine contains existing Geometric Tolerance or Size commands and you attempt to change the GD&T standard from the **Geometric Tolerances** tab of the **Setup Options** dialog, the following rules apply:
    - Changing between ASME Y14.5 publication years:
        - Changing from ASME Y14.5 - 1994 to ASME Y14.5 - 2009: PC-DMIS converts profile tolerances to use the ASME Y14.5.1 - 2019 single value definition. PC-DMIS displays a warning message.
        - Changing from ASME Y14.5 - 2009 to ASME Y14.5 - 1994: PC-DMIS converts profile tolerances to use the ASME Y14.5.1 - 1994 two value definition. Any Geometric Tolerance command that includes a customized datum reference frame, a translation modifier, or a specified material boundary size is rendered invalid. PC-DMIS displays a warning message.
        - Changing from ASME Y14.5 - 1994 to ASME Y14.5 - 2018: PC-DMIS converts profile tolerances to use the ASME Y14.5.1 - 2019 single value definition. Any Concentricity or Symmetry Geometric Tolerance commands are rendered invalid. PC-DMIS displays a warning message.

- Changing from ASME Y14.5 - 2018 to ASME Y14.5 - 1994: PC-DMIS converts profile tolerances to use the ASME Y14.5.1 - 1994 two value definition. Any Geometric Tolerance command that includes a customized datum reference frame, a dynamic profile modifier, a translation modifier, or a specified material boundary size is rendered invalid. PC-DMIS displays a warning message.
- Changing from ASME Y14.5 - 2009 to ASME Y14.5 - 2018: Any Concentricity or Symmetry Geometric Tolerance commands are rendered invalid. PC-DMIS displays a warning message.
- Changing from ASME Y14.5 - 2018 to ASME Y14.5 - 2009: Any Geometric Tolerance commands containing a dynamic profile modifier are rendered invalid. PC-DMIS displays a warning message.

◦ Changing between ISO and ASME is not allowed. All existing Size and Geometric Tolerance commands are rendered invalid. PC-DMIS shows this warning message:

PC-DMIS

All Geometric Tolerances and Size commands within a measurement routine must reference the same GD&T standard. Once you click OK, PC-DMIS disables any pre-existing Geometric Tolerance and Size commands in the measurement routine. After the conversion finishes, you must then recreate these commands. Click OK to continue or click Cancel to abort this operation.

For all the cases above where PC-DMIS displays a warning message:

- If you click **Cancel**, the GD&T standard does not change and your measurement routine remains in its current state.
- If you click **OK** to continue, the GD&T standard for the measurement routine changes to the one you selected. It becomes the default standard for all new Geometric Tolerance and Size commands and the program header updates in the Edit window to reflect this change. PC-DMIS renders invalid all existing Geometric Tolerance and Size commands that could not be converted.

PC-DMIS shows invalid commands in red in the Edit window. You cannot edit or execute these commands. Any command dependent on invalid commands will no longer function.

The main purpose in preserving these commands is to allow you to re-create them and reference them using the newly selected GD&T standard. Once completed, you can then delete the invalid commands and update the dependent commands to reference the new commands.

As discussed in the "Using Geometric Tolerances" chapter, the following geometric tolerance math types are available:

ASME Math Types

- **Datum** math options
- **Considered Feature** math options
- **Tolerance Zone** math options

ISO Math Types

- **Datum** math options
- **Associated Feature** math options
- **Tolerance Zone** math options
- **Size** math options

This **Math** area lets you define what math options you want PC-DMIS to use for new geometric tolerance commands. You can change the math options in individual geometric tolerance commands, and it does not affect future geometric tolerances that you create, nor does it affect what you set here.

For information on how to choose the math options for your application, see the "Using Geometric Tolerances" chapter.

Profiles display MAX/MIN values in Profile command (Edit Window) - This check box defines whether PC-DMIS displays the minimum and maximum deviation values in profile tolerance commands inside the Edit window.

For ISO 1101 2012/2017, ASME Y14.5 – 2009 & ASME Y14.5 - 2018

- If you clear this check box, geometric tolerance commands that represent profile tolerances display a single measured value. This value is based on the single actual value defined by the GD&T standard you selected.
- If you select this check box, profile tolerances show the minimum and maximum deviation values instead of the single measured value.

For ASME Y14.5 - 1994

- This control is greyed out. The check box is always selected and cannot be cleared. Profile tolerances show the minimum and maximum deviation values (see note below).

ASME Y14.5-2009 and ASME Y14.5-2018 use the ASME Y14.5.1-2019 math standard which defines how to calculate an "actual value" of a profile tolerance. Section 8.3.1 (Uniform Tolerance Zones) of the Y14.5.1-2019 standard states, “This concept applies in the same way to equally disposed, unequally disposed and unilateral profile specifications" (see Figures 8-5 thru 8-7). "The actual value of a profile tolerance for a single feature is the value of t_0+2g for the enveloping zone to which the actual surface will conform." This results in the single measured value, equivalent to twice the farthest deviation from the centre of the tolerance zone that PC-DMIS reports.

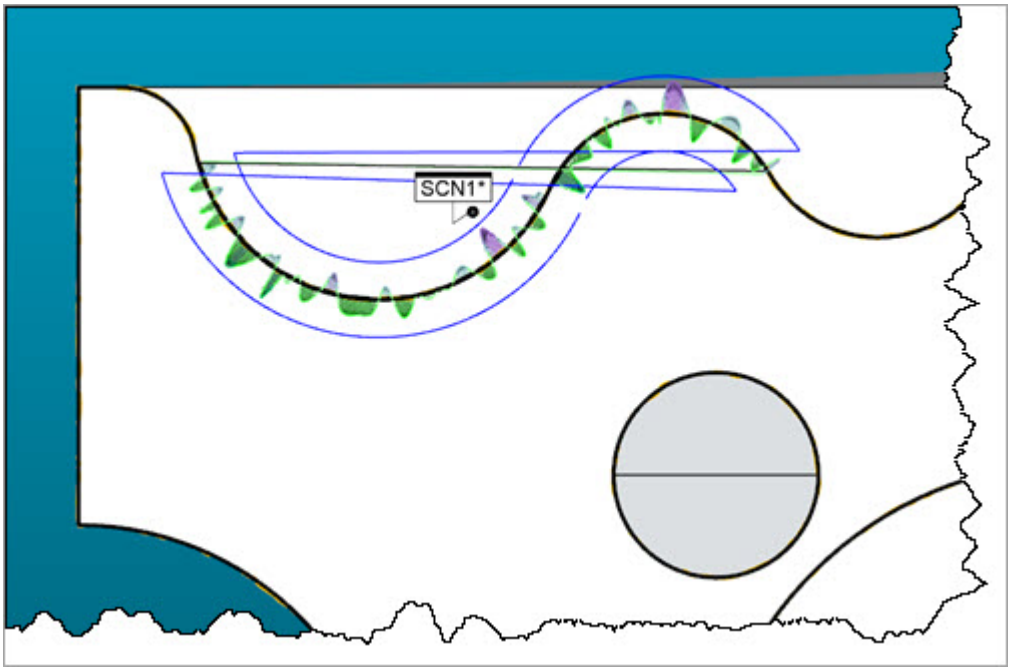
ASME Y14.5 1994 uses the ASME Y14.5.1M-1994 math standard, which states, "For both unilateral and bilateral profile tolerances two actual values are necessarily calculated: one for surface variations in the positive direction and one for the negative direction. For each direction, the actual value of profile is the smallest intermediate tolerance to which the surface conforms. Note that no single actual value may be calculated for comparison to the tolerance value in the feature control frame, except in the case of unilateral profile tolerances." This means that when you select ASME Y14.5 1994 as your GD&T standard, you no longer get a single measured value, you get the minimum and maximum values instead. The only real difference is the way in which the information is presented, tolerance boundaries and conformance are unaffected. For more information, download the "ProfileReporting_Handout_V2" document from the PC-DMIS Knowledgebase repository.

For example, if you select this option and create a Profile, PC-DMIS shows the **MAX** and **MIN** values as part of the MEASURED component of the associated code snippet as seen here:

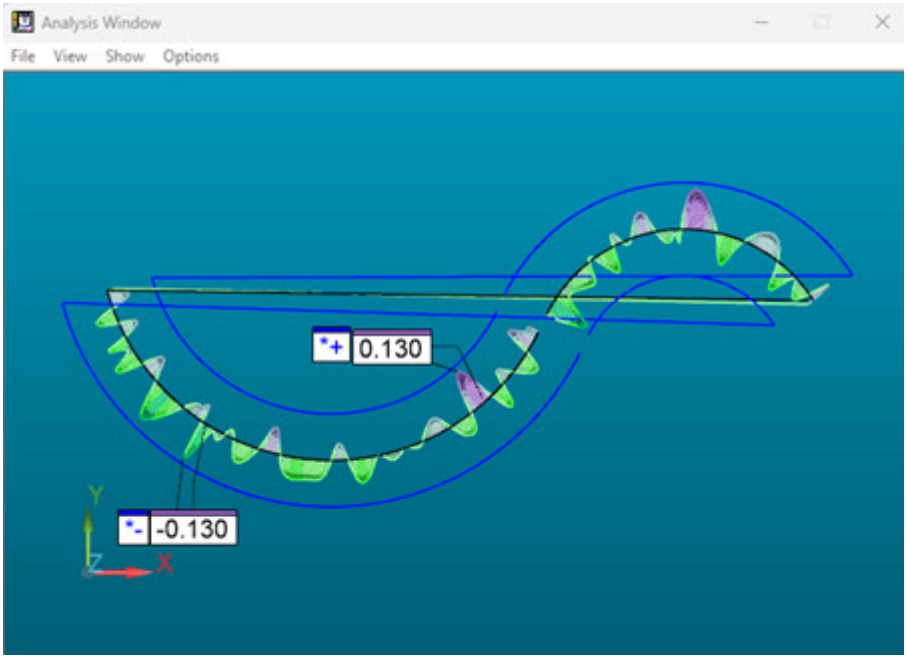
```
FCFPROF1    =GEOMETRIC_TOLERANCE/STANDARD=ASME Y14.5 - 2018,SHOWEXPANDED=YES,
             DESCRIPTION=ON,,
             DISPLAY_COORDS=DRF,
             UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=10,ITERATEANDREPIERCECAD=YES,
             SEGMENT_1,PROFILE_SURFACE,0.3,__,<dat>,<dat>,<dat>,TOL_ZONE_MATH=DEFAULT,
             TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF,MULT=1,
               MEASURED:
                 SCN1:0.130,-0.130,
             ADD
             FEATURES/SCN1,,
```

The first value (0.130) is the **MAX** value, the second value (-0.130) is the **MIN** value.

You should view these values in conjunction with the CAD graphics or a graphical analysis command in order to better understand the deviation direction with respect to the CAD surface.



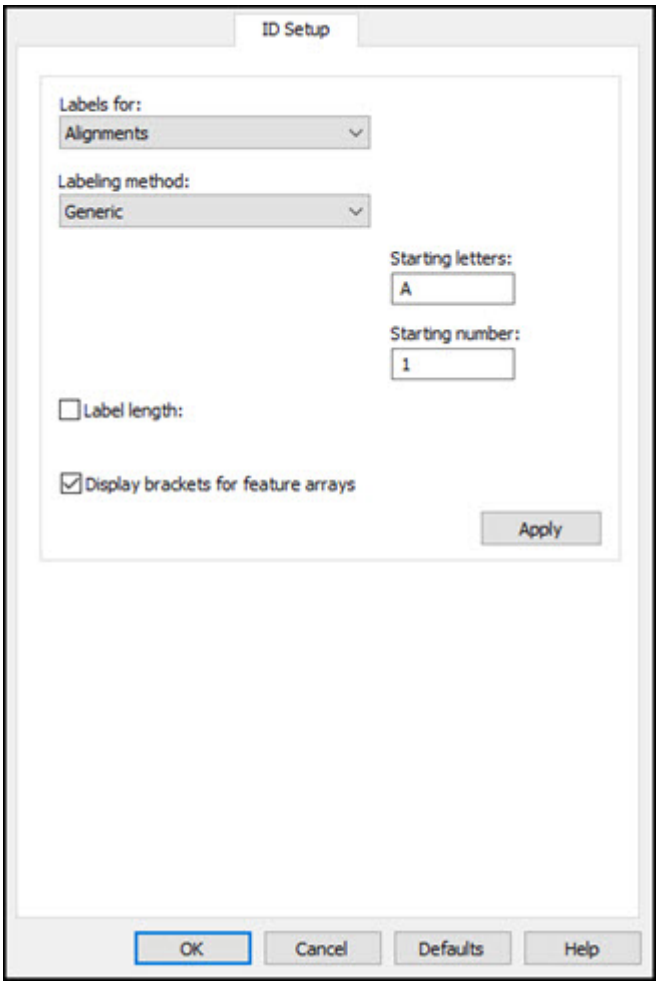
CAD model in the Graphic Display window showing the Profile for the above code snippet



CAD model in the Analysis Window showing the Profile for the above code snippet

This setting does not affect the report. Geometric Tolerance report labels for Profile of a Surface and Profile of a Line always report the single measured value, regardless of the **Profiles display MAX/MIN values in Profile command** setting.

Setup Options: ID Setup tab



Setup Options Dialog Box - ID Setup Tab

Be sure to keep track of IDs already set when changing an identification. It is possible to have ID duplication through multiple changes to this option.

The **ID Setup** tab allows you to alter the format used to identify alignments, dimensions, features, comments, labels, variables, and other items.

To access this option:

1. Access the **Setup Options** dialog box (**Edit | Preferences | Setup**).
2. Select the **ID Setup** tab.

The default **Labeling method** is **Generic**. As you create each feature, PC-DMIS assigns it an ID beginning with the letter **F**, followed by a number (incrementally set from the starting number of one). Type a new value in the appropriate box on this tab to override this setting.

ID Rules

When you create or modify an ID for any command (feature, alignment, dimension, comment, label, variable, and so on), follow these rules:

- Use only letters, numbers, and the underscore character (_). Do not use any other character, such as @ # $ % & * ( ) + - = / \ [ ] { }
- Start the ID with a letter.
- Do not use spaces. Use an underscore instead.
- Do not use the same ID for two different commands, even when the commands are of different types. For example, an alignment should not have the same ID as a feature or a label.
- Do not use names that are identical to PC-DMIS keywords or commands (ALIGN, PROBE, OFFSET, and many more).

While PC-DMIS does not prevent you from breaking most of these rules, doing so may cause problems with expressions, reporting, or how that ID functions with routines that use other products (like DataPage+, Microsoft Excel, and so on).

Apply these same ID rules when you name a probe file. PC-DMIS does not allow a period or any of these characters in a probe file name: < > : " / \ |

For more information, see "Probe File Name" in the "Defining Hardware" chapter.

Options in the ID Setup Tab

Labels for - The **Labels for** drop-down list is used to select the identification used for these items: (See "Labeling Method" below.)

Alignments
Generic is the only method available for identifying alignments.

Comments
Generic is the only method available for identifying comments.

Dimensions
You can identify Dimensions with either the **Generic** or **By Type** method. If you use the **By Type** method, identifications can be similar to each dimension type, or different based on individual need.

Features
You can identify Features with either the **Generic** or **By Type** method.

If you select the **By Type** method, you can also change the color used to display the feature ID.

If you select the **Show All ID** check box and you click **Apply**, PC-DMIS displays all feature ID labels in the Graphic Display window for all features from the current cursor location to the end of the measurement routine. Newly created features also show their ID. If this check box is deselected and you click **Apply**, all features IDs from current cursor location to the end of the measurement routine are hidden. Newly created features still get created in the Edit window, but the ID label doesn't appear in the Graphic Display window.

If the **Color** option is selected (marked), all features created after the **Apply** button is pressed will be affected. (Features created prior to the color change will not be replaced.)

Labels
Generic is the only method available for identifying labels.

Variables
Generic is the only method available for identifying variables.

Call Subs
Generic is the only method available for identifying Called Subroutines.

Spc Queries
You can identify Spc queries with either the **Generic** or **By Type** method.

Geometric Tolerances
You can identify Geometric Tolerances with either the **Generic** or **By Type** method.

Pointcloud Operators
You can identify Pointcloud Operators with either the **By Type** or **Generic** method. If you use the **By Type** method, identifications can be similar to each operator type, or different based on individual need.

Labeling Method - You can use this drop-down window to select between **By Type** and **Generic** identification methods.

By Type
This lets you set the identification for each element type (for example: circle, cone, cylinder, line, plane, point, and sphere).

Generic
This applies the same identification system regardless of feature (dimension) type.

PC-DMIS has no inherent limit on the number of letters used for identifications. However, the Graphic Display area and Edit window have limits on the ID length. Even if the Edit window does not show the complete ID, PC-DMIS internally keeps track of the complete identification.

Starting Letters - This box determines the first letter or letters used in the identification process. PC-DMIS always displays the ID using capital letters.

In various dialog boxes where the ID is displayed, if you modify the ID there, PC-DMIS displays a prompt to ask you if you would like to change the default ID for that item type.

Starting Number - The Starting Number box determines the first number used in the identification process. Any number between 1 and 9999 can be entered.

In various dialog boxes where the ID is displayed, if you modify only the number portion of an ID, the counter may be reset to a desired count.

Label length - The **Label length** check box determines the length of the identification. Selecting it displays a small edit box, into which you can type a number value. The check box must be selected in order for this option to be active. If this length is set, PC-DMIS appends zeros to the ID letters to make up the necessary length.

ID Length = 10, ID Letters = CIRCLE.

PC-DMIS generates an ID = CIRCLE0001, etc. This is done only if the Length is set.

Display brackets for feature arrays - The **Display brackets for feature arrays** check box determines whether array brackets are displayed with IDs for commands executed more than once. When selected, the inspection report displays which instance of the command's execution is being referenced.

F1[3]=POINT MEASURED FROM 1 HIT

Shows that the feature, F1, is measured here for the third (designated by the number three in brackets) time.

You can control the format of the bracketed expression with the array_indices object. See "Array Indices Object" in the "Using Expressions and Variables" chapter.

Apply - The **Apply** button applies the changes described in "Labeling Method" to any feature identifications. These changes apply only to feature IDs. If the **Apply** button is not

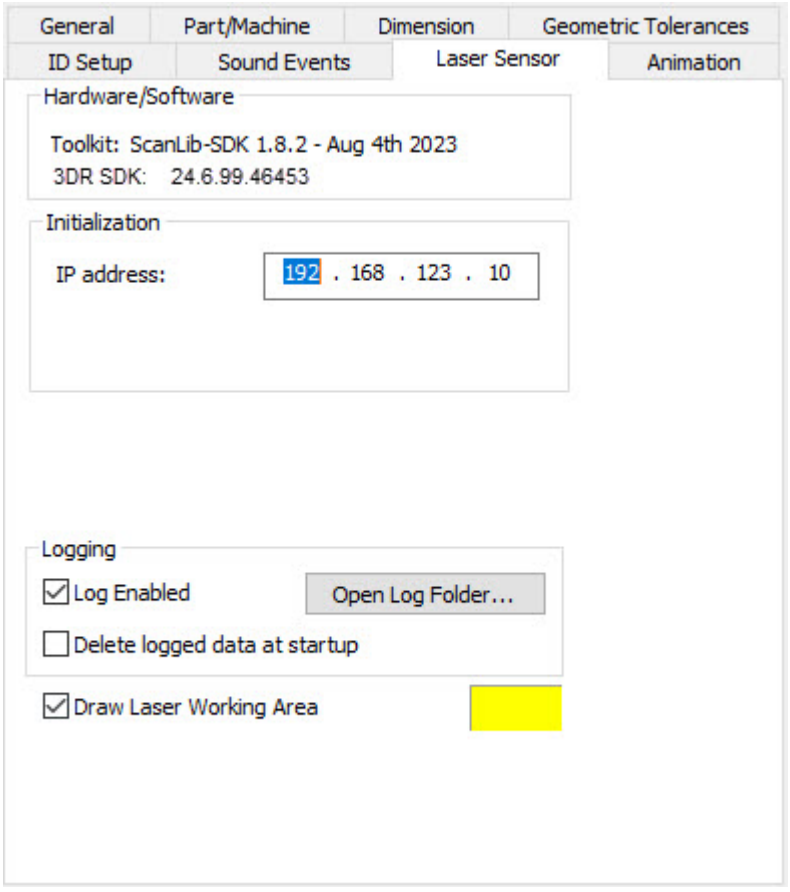
pressed, PC-DMIS will continue to assign feature identifications using the previous method listed.

Each feature, dimension, and other command should have a unique ID.

Default - The **Default** button allows you to update the default settings to all of the ID Setup parameters. When a new measurement routine is created, it will reflect any changes made to the parameters, *only* if the **Default** button is clicked. For additional information on **Default** buttons, see the "Default" topic in the "Navigating the User Interface" chapter

Always press the **Apply** button after you make a change (before you press the **OK** or **Default** buttons).

Setup Options: Laser Sensor tab



Setup Options Dialog Box - Laser Sensor Tab

The **Laser Sensor** tab (**Edit | Preferences | Setup**) contains information and controls to define the defaults settings for your laser sensor.

Hardware/Software - This area provides you with the current versions of the listed hardware and software.

Initialization - This shows the IP address for your laser sensor.

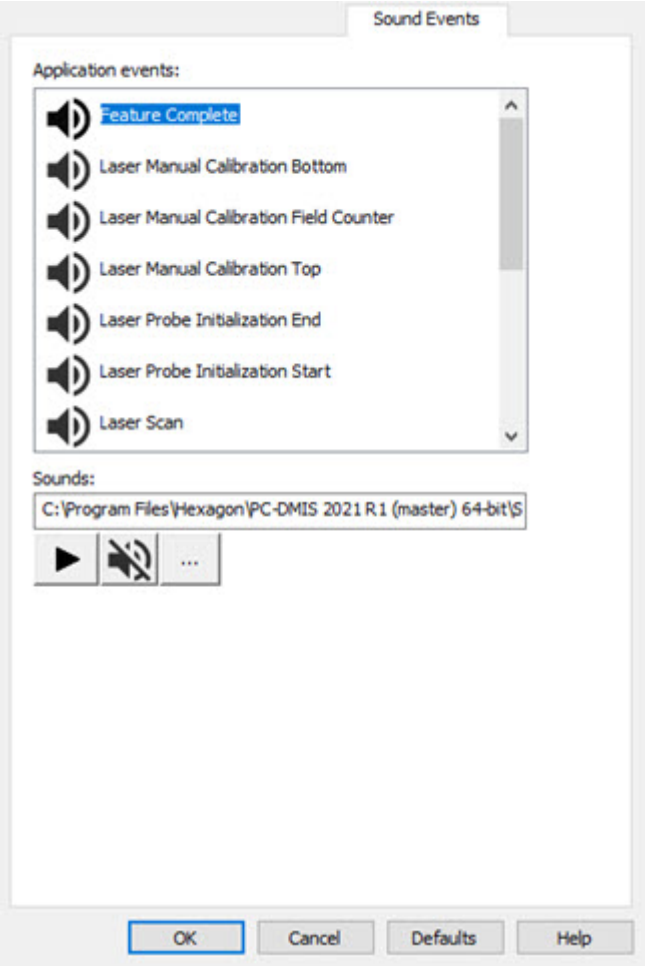
Logging - This area provides you with the following options to perform data logging.

> **Log Enabled** check box - Select this check box to enable data logging. You can click the **Open Log Folder** button to view the location of the generated log file.
>
> **Delete logged data at startup** check box - This option is only available when you select the **Log Enabled** check box. When you select this option, PC-DMIS deletes the log data at startup.

Draw Laser Working Area check box- Select this check box and then use the color patch to select a color. The next time you scan a CAD model, PC-DMIS colors the scan working areas with the selected color.

The **Laser Sensor** tab is available only when you are licensed with the laser option and when using a laser probe. For additional information on this tab, see the "Step 3: Define Setup Options for the Laser Sensor" in the PC-DMIS Laser documentation.

Setup Options: Sound Events tab



Setup Options Dialog Box - Sound Events Tab

The **Sound Events** tab (**Edit | Preferences | Setup**) contains a list of application events that you can associate with a sound file of your choice. When the event occurs, PC-DMIS automatically plays the associated sound.

Application Events list
This list shows the application events to which you can associate sound files.

Sounds box
This box lists the path to the sound file for the selected application event.

Play Sound button

This button tests the specified sound file. It is disabled if no sound is associated with the event.

Turn Sound Off button

This button turns off the specified sound file.

Browse button

This button is used for navigating to and selecting a sound file for the selected application event.

Associating a Sound

To associate a custom sound file with an event:

1. Select the event from the **Applications Events** list.
2. Click the **Browse** button ( ).
3. Navigate to the directory containing your sound file. PC-DMIS only supports the playing of .wav files.
4. Select the .wav file and click **Open**. The **Sounds** list box displays the path to the selected sound file.
5. Test the file by pressing the **Play** button ( ).
6. Click **OK** to save your changes.

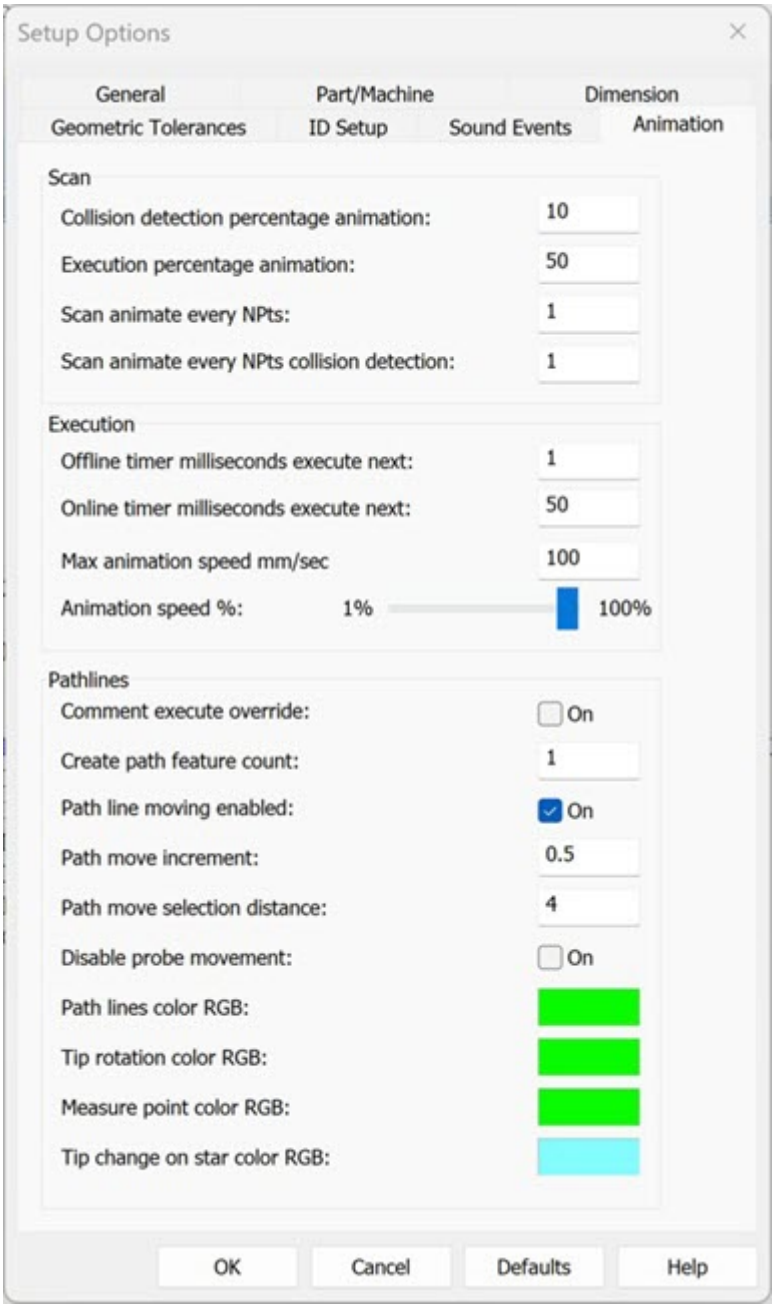
Turning Off a Sound

You can turn off an associated sound from an event. To turn off a sound:

1. Select the event from the **Applications Events** list.
2. Click the **Sound Off** button ( ).

   The speaker icon ( ) to the left of the event changes to show a circle with a line through it ( ), indicating that the event is turned off.
3. Click **OK** to save your changes. The sound is removed from the event.

Setup Options: Animation tab



Setup Options Dialog Box - Animation Tab

The **Animation** tab (**Edit | Preferences | Setup**) contains offline animation and path line settings.

Scan area

Collision detection percentage animation: - This defines the percentage of probe animation that occurs during a collision detection operation.

- A value of 100 indicates that the display is redrawn at its maximum rate.
- A value of 0 indicates that the display is not redrawn at all.
- The default value is 50.

Execution percentage animation: - This defines the percentage of probe animation that occurs during normal measurement routine execution.

- A value of 100 indicates that the display is redrawn at its maximum rate.
- A value of 0 indicates that the display is not redrawn at all.
- The default value is 50.

Scan animate every NPts: - This limits the number of points that PC-DMIS uses for the animation. For example, if you set this value to 10, PC-DMIS only takes every 10th point, plus the first and last points. PC-DMIS uses this value for probe animation during execution. The default value is 50.

Scan animate every NPts collision detection: - This limits the number of points that PC-DMIS uses for the collision detection animation. For example, if you set this value to 10, PC-DMIS only takes every 10th point, plus the first and last points. PC-DMIS uses this value for probe animation during collision detection.

Execution area

Offline timer milliseconds execute next: - This sets the frequency of how often PC-DMIS processes commands during execution in Offline mode. This is a number of milliseconds. The default is 50.

Online timer milliseconds execute next: - This sets the frequency of how often PC-DMIS processes commands during online execution. This is a number of milliseconds. The default is 50.

For example if these are set to 1 then PC-DMIS would try to process a command during execution every millisecond.

Max animation speed (mm/sec): - This lets you define the maximum animation speed that the animated probe uses in the Graphic Display window during measurement routine execution. The speed is in mm per second. You may find it useful to alter this value for complex measurement routines that cause the animation to render too slowly. To increase the duration between redrawn views of the animation, increase this value. This causes PC-DMIS to draw fewer animation steps.

Animation speed %: - The slider allows you to quickly and easily adjust the actual percentage used of the **Max animation speed** value.

The **Animation speed %** slider is directly linked to the **Offline Animation Speed** slider on the **Execution** dialog box. The value you set this slider to, for example, is the same value set for the slider on the **Execution** dialog box. When you close and then reopen the measurement routine, the slider value in both locations is set back to the default value of 100%.

For details on the **Execution** dialog box, see the "Using the Execution Dialog Box" topic in the "Using Advanced File Options" chapter of the PC-DMIS Core documentation.

Changing Animation Speeds: If you want to further fine-tune your offline animation speeds, see the "Executing and Debugging Measurement Routines Offline" topic in the "Working in Offline Mode" chapter.

Pathlines area

Comment execute override: - This check box determines whether or not PC-DMIS executes `COMMENT` commands when you generate path lines. If you mark this check box then they are executed. This check box is cleared by default.

Create path feature count: - This box is used when you select **View | Path Lines from Cursor**. It defines how many features above and below the cursor location are used. For example, if you set this to 3, PC-DMIS would use three features above and three features below the cursor location. The default value is 1, which means that PC-DMIS draws path lines for one feature that precedes the current feature and one feature that follows the current feature. See "Showing, Animating, and Moving Path Lines" in the "Editing the CAD Display" chapter.

Path lines color RGB: - This box defines the color for generated path lines in the Graphic Display window. Clicking on the color opens up the standard **Color** dialog box, from which you can choose a new color.

Path line moving enabled: - This check box determines whether or not path-line moving is enabled. If marked, you can click on a path line to insert a `MOVE/POINT` command at that location. See "Moving Path Lines" in the "Editing the CAD Display" chapter.

Path move increment - This defines the increment distance for moving path lines in the **Move Path Line** dialog box. See "Moving Path Lines" in the "Editing the CAD Display" chapter.

Path move selection distance - This determines the selection distance from a start and end point of a path line. If the **Path line moving enabled** check box is marked, and you click on a path line within the specified distance inside the Graphic Display window, PC-

DMIS searches for an existing `MOVE/POINT` command to modify instead of inserting a new `MOVE/POINT` command that splits the selected path line.

Disable probe movement - This check box is cleared by default, which causes the probe to move during path line creation. If this box is checked, the probe no longer moves during path line creation.

Path lines color RGB - This defines the main path line color as a probe moves between features.

Tip rotation color RGB - This defines the color for a probe's path lines as it rotates its tip to a new angle.

Measure point color RGB - This defines the color for a probe's path lines as it contacts with the part to measure points that define the feature.

TIP change on star color RGB - This defines the color of the path lines that PC-DMIS displays when you change the TIP in star configuration. The machine or wrist does not move. The path line indicates the change in the hot spot from one TIP to another.

Modifying Report and Motion Parameters

The **Edit | Preferences | Parameters** menu option allows you to modify the content, form, and method of calculation used in any report. Additionally, it allows you to modify the machine motion parameters of a DCC CMM. This menu option opens the **Parameter Settings** dialog box.

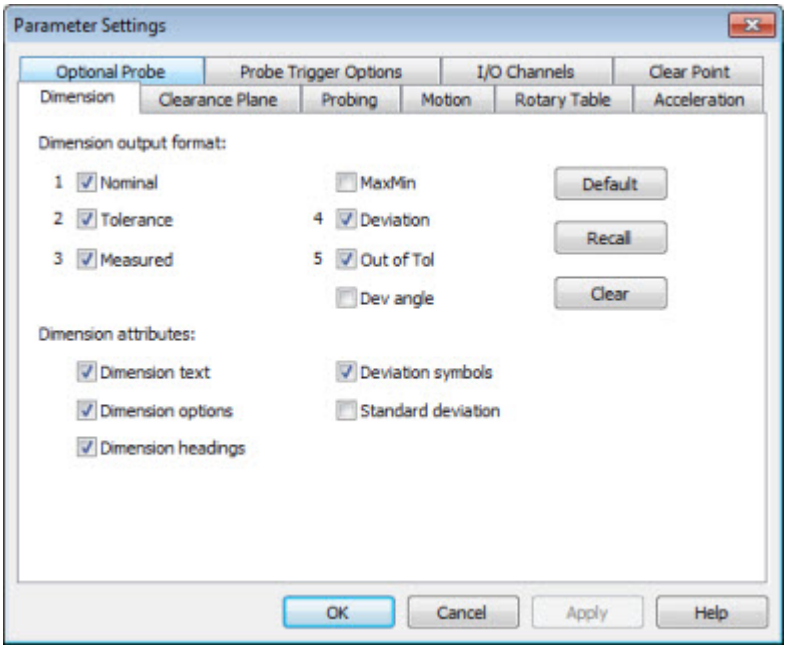
In addition to accessing the **Parameter Settings** dialog box, you can insert the commands from this dialog box directly into the measurement routine by selecting them from the **Insert | Parameter Change** submenu.

The following tabs are available on the **Parameter Settings** dialog box.

- Parameter Settings: Dimension Tab
- Parameter Settings: Clearance Plane Tab
- Parameter Settings: Probing Tab
- Parameter Settings: Motion Tab
- Parameter Settings: Rotary Table Tab
- Parameter Settings: Acceleration Tab
- Parameter Settings: Probe Options Tab
- Parameter Settings: Probe Trigger Options Tab

- Parameter Settings: I/O Channels Tab
- Parameter Settings: Clear Point Tab

Parameter Settings: Dimension Tab



Parameter Settings Dialog Box - Dimension Tab

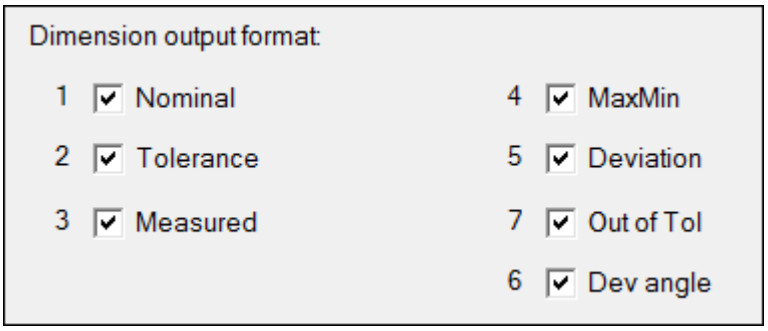
Use the **Dimension** tab to vary the format of the dimension output and alter the printed report. The settings on this tab apply only to labels in dimension templates.

The reports from DEFAULT.RPT and TEXTONLY.RPT reflect any changes made to these settings.

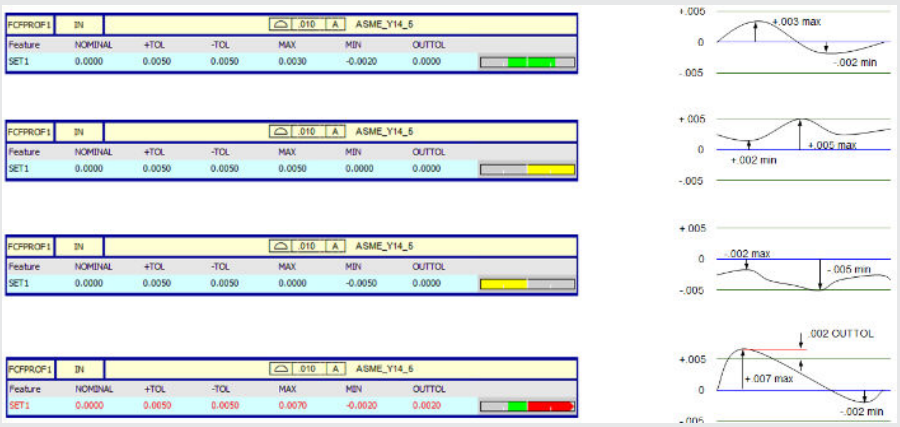
To access the **Dimension** tab:

1. Access the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Click the **Dimension** tab.

Dimension Output Format



The `FORMAT/TEXT` command controls the PC-DMIS dimension output format. To change the format, select the desired check boxes.

Available Formats	Description
Nominal	Displays the nominal values for all dimensions.
Tolerance	Displays the tolerance values for all dimensions.
Measured	Displays the measured values for all dimensions.
MaxMin	Displays the maximum and minimum deviation values from the points that make up Line Profile and Surface Profile dimensions.  *Example of the Report window showing MaxMin values on four different profile dimensions.*
Deviation	Displays the deviation values for all dimensions.
Out of Tol	Displays the out of tolerance values for all dimensions.
Dev Angle	Displays the deviation angles on the Position dimensions.

When switching from box tolerancing to position (and back), it is important to check the format for correctness.

PC-DMIS offers the same print formats for box tolerancing and positioning tolerancing, although their columns look slightly different due to additional columns for position dimensions.

PC-DMIS displays a number to the left of the check box to indicate the order of the output selection. This enables you to alter the order of the format to meet your needs.

The Edit window command line for this option reads:

```
FORMAT/TEXT,OPTIONS,HEADINGS,SYMBOLS, SD; "DIMENSION OUTPUT"
```

DIMENSION OUTPUT = The format of the output is based on the order of selection. The default output displays the entire selection, in the order indicated.

Dimension Text

The **Dimension Text** check box controls whether or not the dimension text will be displayed in the Edit window for any dimensions that follow the command.

The Edit window command line for this option reads:

```
FORMAT/TEXT, , , , ;NOM,TOL,MEAS,MAXMIN,DEV,OUTTOL,DEVANG
```

For a description the options on this command, see "Dimension Output Format".

Dimension Options

The **Dimension Options** check box controls whether or not the dimension options will be displayed in the Edit window for any dimensions that follow the command.

These options include:

- Units (see "Units" in the "Using Legacy Dimensions" chapter)
- Graphical analysis (see "Analysis Settings" in the "Using Legacy Dimensions" chapter)
- Textual analysis (see "Analysis Settings" in the "Using Legacy Dimensions" chapter)
- Arrow multiplier (see "Analysis Settings" in the "Using Legacy Dimensions" chapter)
- Output options (see "Analysis Settings" in the "Using Legacy Dimensions" chapter)

The Edit window command line for this check box reads:

```
FORMAT/OPTIONS, , , , ;NOM,TOL,MEAS,MAXMIN,DEV,OUTTOL
```

Dimension Headings

The **Dimension Headings** check box controls the column headings on the inspection report. If this check box is not selected, then PC-DMIS does not print any column headings.

Deviation Symbols

The **Deviation Symbols** check box shows the deviation within the set range. If the out of tolerance range is high, PC-DMIS indicates the deviation using the "greater than" symbol (>) on the right side of the line. If the out of tolerance range is low, PC-DMIS indicates the deviation using the "less than" symbol (<).

For example:

Nominal = 0.00

Measured = 0.02

Positive Tolerance = 0.10

Negative Tolerance = 0.20

Total Tolerance Range = (.10 - (-.20)) = .30

Percentage = 100*(.02 - (-.20))/.3 = 73.3 %

- - - - - - # - - looks at the % and shifts based on the %.

DIM D1= LOCATION OF CIR F5 GRAPH=OFF TEXT=OFF MULT=1.00

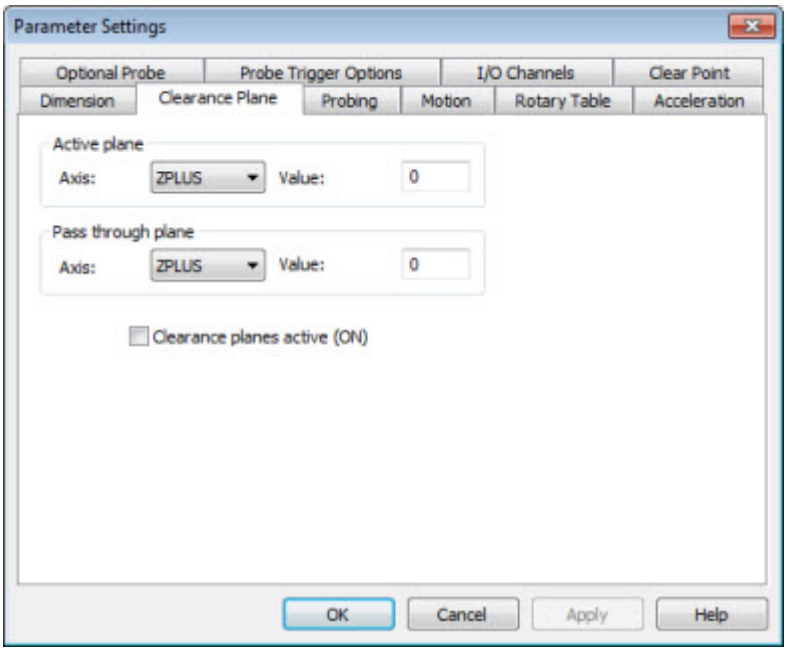
AX	NOM	+TOL	-TOL	MEAS	DEV	OUT-TOL	
x	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	----#----
y	2.0000	0.0100	0.0100	2.0000	0.0000	0.0000	----#----
z	-0.2500	0.0100	0.0100	-0.2500	0.0000	0.0000	----#----
d	2.0000	0.0100	0.0100	2.0000	0.0000	0.0000	
v	i	j	k				

END OF DIMENSION D1

Standard Deviation

The **Standard Deviation** check box displays the standard deviation of features.

Parameter Settings: Clearance Plane Tab

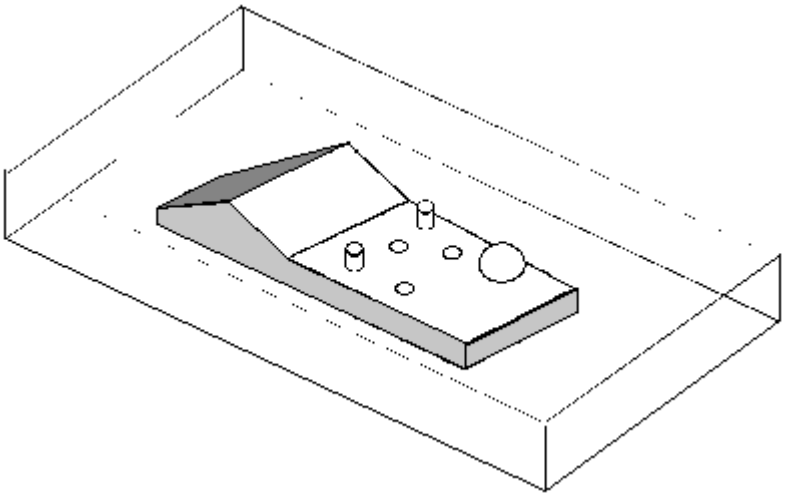


Parameter Settings Dialog Box - Clearance Plane Tab

The **Clearance Plane** tab provides the means to define and add clearance planes. Clearance planes, in essence, create an envelope around a part. The probe stays on this envelope when it moves from one feature to another.

PC-DMIS moves the probe out from the part at a predetermined distance, relative to the coordinate system in which it was defined.

After the last hit on the feature is measured, the probe stays at probe depth until called to the next feature. This can help reduce routine creation time because you don't have to define as many intermediate moves. In addition, properly-defined clearance planes can help protect your probe from accidental collisions with the part.



Example showing a part with an imaginary envelope from clearance planes.

To use clearance planes:

1. Open the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**) and then select the **Clearance Plane** tab.
2. Select the clearance plane through the **Active plane** and **Pass through plane** areas, and use the respective **Value** boxes to specify the clearance distances.
3. Click **OK** to finish defining the clearance plane. PC-DMIS inserts a `CLEARP` command that contains the clearance plane information into the Edit window. The finished command looks something like this:

```
CLEARP/ACTIVE_PLANE, n, PASS_THROUGH_PLANE, n,TOG1
```

   - `ACTIVE_PLANE` and `PASS_THROUGH_PLANE` refer to the axes selected.
   - `n` refers to the offset distances specified.
   - `TOG1` is an ON/OFF toggle field that determines whether the clearance plane is active and automatically used for newly-created Measured and Auto features.

4. You can then insert `MOVE/CLEARPLANE` commands into your measurement routine. A `MOVE/CLEARPLANE` command does not in itself create motion to move the probe to the clearance plane. Rather, when PC-DMIS encounters a `MOVE/CLEARPLANE` command during execution, that command essentially gives permission to move to the predefined clearance plane on the next move, measure, tip selection, or Auto feature command. When PC-DMIS is about to execute one of those motion commands, the probe moves to the specified distance away from the selected active plane.
5. If you define another `CLEARP` command, the very next `MOVE/CLEARPLANE` command moves first to the old clearance plane, then to the Pass Through plane, and then to the new clearance plane.

To display the current clearance plane as a translucent image in the Graphic Display window, from the **Graphic Items** toolbar, select the **Show Clearance Plane** icon ( ). For more information, see "Viewing Clearance Planes" in the "Editing the CAD Display" chapter.

Active Plane Area

The **Active Plane** area defines the plane (or axis) that PC-DMIS locates and measures features. The **Value** box defines the clearance plane as an offset distance in the current units of measurement away from the specified plane. To define a clearance plane, select the plane from the **Axis** list and then enter a new value in the **Value** box.

Pass Through Plane Area

The **Pass Through Plane** defines a clearance plane that the probe moves to and then passes through to get to the next active clearance plane after a probe's `TIP` command. The new `CLEARP` definition command must immediately follow the `TIP` command to properly define the pass through plane. When PC-DMIS encounters the next `MOVE/CLEARPLANE` command, it moves to the pass through plane and remains at that offset distance until it reaches the next active clearance plane.

If you add additional moves or a loadprobe command before a tip change command, PC-DMIS deactivates the pass through plane.

When you make adjustments to commands in your measurement routine, be sure to review the path lines. This allows you to see the effect your changes make without having to run the measurement routine.

Clearance Plane Active (ON) Check Box

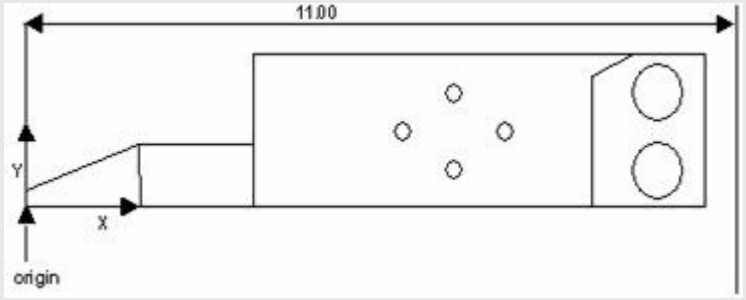
If you select the **Clearance planes active (ON)** check box, PC-DMIS automatically inserts a `MOVE/CLEARPLANE` command before any Measured or Auto features you insert into the Edit window from that point on.

Notes on Clearance Planes

Be aware of the sign of a clearance plane when you enter its distance value. The sign must correspond to the positive or negative end of the normal axis that defines the plane. For example, to define a top clearance plane, enter a positive value, and to define a bottom clearance plane, enter a negative value.

Movement from one clearance plane to another affects the position of the probe. Make sure that the set clearance plane is sufficient to clear the part.

PC-DMIS defines a clearance plane relative to the current coordinate system and part origin. Therefore, you must be careful when you define a clearance plane to ensure adequate clearance around the part.



Clearance Plane example

Using the above drawing, suppose your part is ten inches long, and it is lined up close to the machine axes with the X origin at the bottom left-hand corner. To define a one-inch clearance from the right side of the part, set the XPLUS clearance plane to 11 inches.

Always define clearance planes relative to the current coordinate system. When you create a new coordinate system, the clearance planes still relate to the first alignment. If you wish to associate the clearance planes with the new coordinate system, you must redefine them.

PC-DMIS does not use clearance planes when taking sample hits. It is therefore important, when you measure pins, to set the spacer value to a distance that will allow the probe to move around the pin.

For a clearance plane example, see "Clearance Plane Example".

Clearance Plane Example

This example provides a complete measurement routine with comments in the routine that provide explanation. Below the code sample, there are screen captures of the clearance planes.

Code Sample

```
PART NAME  : test
REV NUMBER :
SER NUMBER :
STATS COUNT : 1
```

```
STARTUP    =ALIGNMENT/START,RECALL:USE_PART_SETUP,LIST=YES
            ALIGNMENT/END
$$ NO,
            ----------------------------
            ROUTINE STARTS IN MANUAL MODE
            ----------------------------

            MODE/MANUAL
            PREHIT/0.0394
            RETRACT/0.0394
            MOVESPEED/ 500
            MANRETRACT/0
            FORMAT/
TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;NOM,TOL,MEAS,DEV,OUTTOL, ,

            TEMPCOMP/METHOD = AUTOMATIC,MATERIAL = Zerodur;
Nexcera,CTE=0
              ,SET WARNING LIMIT = FALSE,MINIMUM =
10,MAXIMUM = 40
              ,PART SENSOR NUM=DEFAULT,X SCALE= 20,Y SCALE=
20,Z SCALE= 20,PART TEMP=20
            LOADPROBE/INDEXABLE
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
MAN_ALIGN  =GROUP/SHOWALLPARAMS=YES,EXECUTION CONTROL=AS
MARKED
PLN_A_MAN   =FEAT/PLANE,CARTESIAN,TRIANGLE
             THEO/<1.5084,1.1159,0>,<0,0,1>
             ACTL/<1.5084,1.1159,0>,<0,0,1>
             MEAS/PLANE,4
               HIT/
BASIC,NORMAL,<0.2631,1.9109,0>,<0,0,1>,<0.2631,1.9109,0>,USE
THEO=YES
               HIT/
BASIC,NORMAL,<2.5538,2.186,0>,<0,0,1>,<2.5538,2.186,0>,USE
THEO=YES
               HIT/
BASIC,NORMAL,<3.1128,0.2527,0>,<0,0,1>,<3.1128,0.2527,0>,USE
THEO=YES
               HIT/
BASIC,NORMAL,<0.1038,0.1142,0>,<0,0,1>,<0.1038,0.1142,0>,USE
THEO=YES
             ENDMEAS/
A1          =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
               ALIGNMENT/LEVEL,ZPLUS,PLN_A_MAN
               ALIGNMENT/TRANS,ZAXIS,PLN_A_MAN
             ALIGNMENT/END
LIN_B_MAN   =FEAT/LINE,CARTESIAN,UNBOUNDED
             THEO/<1.2188,0,-0.117>,<1,0,0>
             ACTL/<1.2188,0,-0.117>,<1,0,0>
             MEAS/LINE,3,ZPLUS
               HIT/
BASIC,NORMAL,<1.2188,0,-0.112>,<0,-1,0>,<1.2188,0,-0.112>,US
E THEO=YES
               HIT/
BASIC,NORMAL,<2.3457,0,-0.1143>,<0,-1,0>,<2.3457,0,-0.1143>,
```

```
      USE THEO=YES
                     HIT/
      BASIC,NORMAL,<3.2475,0,-0.1247>,<0,-1,0>,<3.2475,0,-0.1247>,
      USE THEO=YES
                   ENDMEAS/
      A2           =ALIGNMENT/START,RECALL:A1,LIST=YES
                     ALIGNMENT/LEVEL,ZPLUS,PLN_A_MAN
                     ALIGNMENT/TRANS,ZAXIS,PLN_A_MAN
                     ALIGNMENT/
      ROTATE,XPLUS,TO,LIN_B_MAN,ABOUT,ZPLUS
                     ALIGNMENT/TRANS,YAXIS,LIN_B_MAN
                   ALIGNMENT/END
      PNT_C_MAN    =FEAT/POINT,CARTESIAN
                   THEO/<0,0.3496,-0.3075>,<-1,0,0>
                   ACTL/<0,0.3496,-0.3075>,<-1,0,0>
                   MEAS/POINT,1,WORKPLANE
                     HIT/
      BASIC,NORMAL,<0,0.3496,-0.3075>,<-1,0,0>,<0,0.3496,-0.3075>,
      USE THEO=YES
                   ENDMEAS/
      A3           =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
                     ALIGNMENT/LEVEL,ZPLUS,PLN_A_MAN
                     ALIGNMENT/TRANS,ZAXIS,PLN_A_MAN
                     ALIGNMENT/
      ROTATE,XPLUS,TO,LIN_B_MAN,ABOUT,ZPLUS
                     ALIGNMENT/TRANS,YAXIS,LIN_B_MAN
                     ALIGNMENT/TRANS,XAXIS,PNT_C_MAN
                   ALIGNMENT/END
                ENDGROUP/ID=MAN_ALIGN
      DCC_ALIGN  =GROUP/SHOWALLPARAMS=YES,EXECUTION CONTROL=AS
      MARKED
        $$ NO,
                   ----------------------------
                   ROUTINE ENTERS DCC MODE
                   ----------------------------

                   COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
      CONTINUE=NO,
                   Changing to DCC Mode! Do not continue unless
      avoidance moves have been added.
                   MODE/DCC
        $$ NO,
                   ------------------------------------------------
      ------
                   CLEARANCE PLANE IS DEFINED  (2 UNITS ABOVE TOP
      FACE)
                   ------------------------------------------------
      ------

                   CLEARP/ZPLUS,2,ZPLUS,0,ON
      PLNA_DCC     =FEAT/PLANE,CARTESIAN,TRIANGLE
                   THEO/<1.5084,1.1159,0>,<0,0,1>
                   ACTL/<1.5084,1.1159,0>,<0,0,1>
                   MEAS/PLANE,4
          $$ NO,
                     ------------------------------------------
```

```
---------------------------
                PROBE MOVES TO CLEARANCE PLANE
                ---------------------------------------------
---------------------------

                MOVE/CLEARPLANE
                HIT/
BASIC,NORMAL,<0.2631,1.9109,0>,<0,0,1>,<0.2631,1.9109,0>,USE
THEO=YES
                HIT/
BASIC,NORMAL,<2.5538,2.186,0>,<0,0,1>,<2.5538,2.186,0>,USE
THEO=YES
                HIT/
BASIC,NORMAL,<3.1128,0.2527,0>,<0,0,1>,<3.1128,0.2527,0>,USE
THEO=YES
                HIT/
BASIC,NORMAL,<0.1038,0.1142,0>,<0,0,1>,<0.1038,0.1142,0>,USE
THEO=YES
              ENDMEAS/
A3_DCC1      =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
                ALIGNMENT/LEVEL,ZPLUS,PLNA_DCC
                ALIGNMENT/TRANS,ZAXIS,PLNA_DCC
                ALIGNMENT/
ROTATE,XPLUS,TO,LIN_B_MAN,ABOUT,ZPLUS
                ALIGNMENT/TRANS,YAXIS,LIN_B_MAN
                ALIGNMENT/TRANS,XAXIS,PNT_C_MAN
              ALIGNMENT/END
LINB_DCC     =FEAT/LINE,CARTESIAN,UNBOUNDED
              THEO/<1.2188,0,-0.117>,<1,0,0>
              ACTL/<1.2188,0,-0.117>,<1,0,0>
              MEAS/LINE,3,ZPLUS
    $$ NO,
                ---------------------------------------------
---------------------------
                PROBE MOVES TO CLEARANCE PLANE
                ---------------------------------------------
---------------------------

                MOVE/CLEARPLANE
                HIT/
BASIC,NORMAL,<1.2188,0,-0.112>,<0,-1,0>,<1.2188,0,-0.112>,US
E THEO=YES
                HIT/
BASIC,NORMAL,<2.3457,0,-0.1143>,<0,-1,0>,<2.3457,0,-0.1143>,
USE THEO=YES
                HIT/
BASIC,NORMAL,<3.2475,0,-0.1247>,<0,-1,0>,<3.2475,0,-0.1247>,
USE THEO=YES
              ENDMEAS/
A3_DCC2      =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
                ALIGNMENT/LEVEL,ZPLUS,PLNA_DCC
                ALIGNMENT/TRANS,ZAXIS,PLNA_DCC
                ALIGNMENT/
ROTATE,XPLUS,TO,LINB_DCC,ABOUT,ZPLUS
                ALIGNMENT/TRANS,YAXIS,LINB_DCC
                ALIGNMENT/TRANS,XAXIS,PNT_C_MAN
```

```
                ALIGNMENT/END
PNTC_DCC       =FEAT/POINT,CARTESIAN
                THEO/<0,0.3496,-0.3075>,<-1,0,0>
                ACTL/<0,0.3496,-0.3075>,<-1,0,0>
                MEAS/POINT,1,WORKPLANE
    $$ NO,
                  ------------------------------------------
----------------------------
                  PROBE MOVES TO CLEARANCE PLANE
                  ------------------------------------------
----------------------------

                  MOVE/CLEARPLANE
                  HIT/
BASIC,NORMAL,<0,0.3496,-0.3075>,<-1,0,0>,<0,0.3496,-0.3075>,
USE THEO=YES
                ENDMEAS/
A3_DCC3        =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
                  ALIGNMENT/LEVEL,ZPLUS,PLNA_DCC
                  ALIGNMENT/TRANS,ZAXIS,PLNA_DCC
                  ALIGNMENT/
ROTATE,XPLUS,TO,LINB_DCC,ABOUT,ZPLUS
                  ALIGNMENT/TRANS,YAXIS,LINB_DCC
                  ALIGNMENT/TRANS,XAXIS,PNTC_DCC
                ALIGNMENT/END
              ENDGROUP/ID=DCC_ALIGN
$$ NO,
              ------------------------------------------------
-----------------------
              PROBE MOVES TO CLEARANCE PLANE
              ------------------------------------------------
-----------------------

              MOVE/CLEARPLANE
PLN1         =FEAT/CONTACT/PLANE/
DEFAULT,CARTESIAN,TRIANGLE,LEAST_SQR
              THEO/<1.5748,1.1811,-0.3937>,<0,0,1>
              ACTL/<1.5748,1.1811,-0.3937>,<0,0,1>
              TARG/<1.5748,1.1811,-0.3937>,<0,0,1>
              ANGLE VEC=<1,0,0>,RADIAL
              SHOW FEATURE PARAMETERS=NO
              SHOW CONTACT PARAMETERS=NO
$$ NO,
              ------------------------------------------------
-----------------------
              PROBE MOVES TO CLEARANCE PLANE
              ------------------------------------------------
-----------------------

              MOVE/CLEARPLANE
CON1         =FEAT/CONTACT/CONE/DEFAULT,CARTESIAN,IN
              THEO/<0.3937,0.3937,0>,<0,0,1>,90,-0.2756,0.5906
              ACTL/<0.3937,0.3937,0>,<0,0,1>,90,-0.2756,0.5906
              TARG/<0.3937,0.3937,0>,<0,0,1>
              START ANG=0,END ANG=360
```

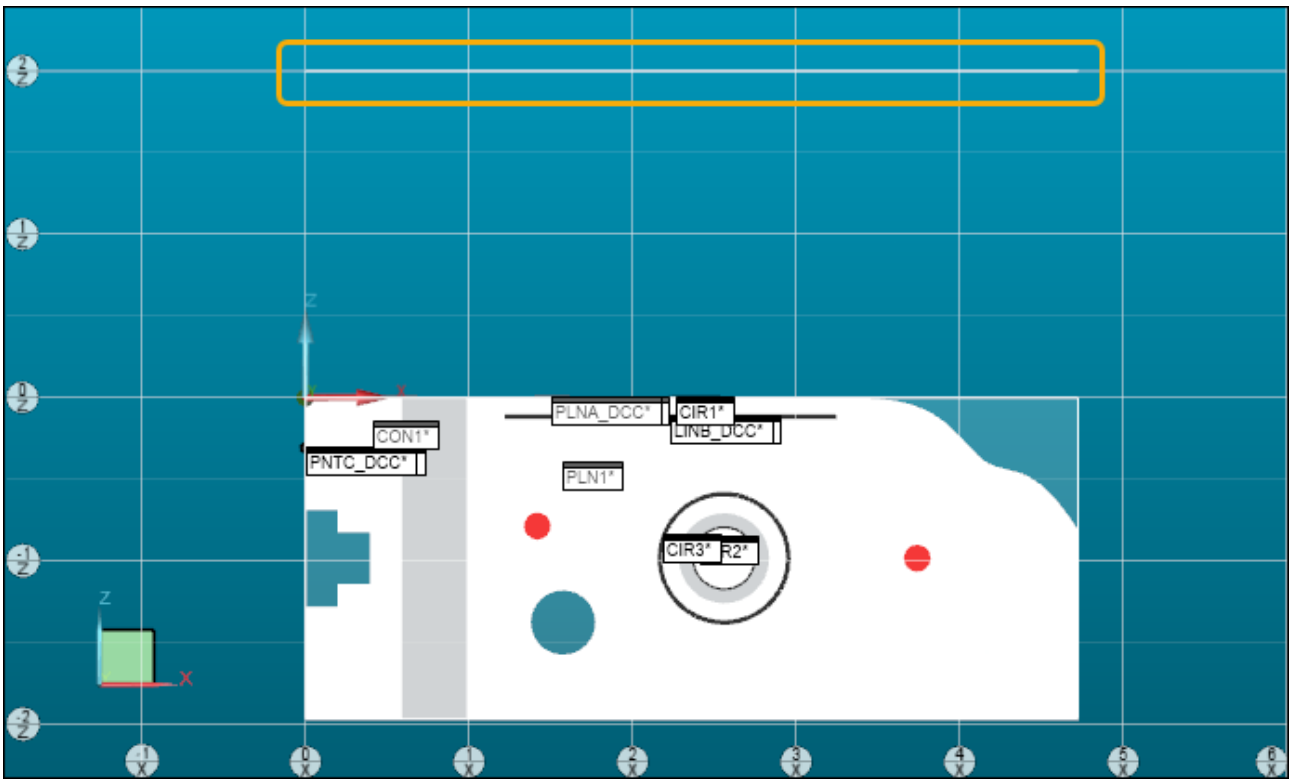
```
            ANGLE VEC=<1,0,0>
            SHOW FEATURE PARAMETERS=NO
            SHOW CONTACT PARAMETERS=NO
$$ NO,
            ------------------------------------------------
------------------------
            PROBE MOVES TO CLEARANCE PLANE
            ------------------------------------------------
------------------------

            MOVE/CLEARPLANE
CIR1       =FEAT/CONTACT/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
            THEO/<2.3819,1.1811,0>,<0,0,1>,0.3228
            ACTL/<2.3819,1.1811,0>,<0,0,1>,0.3228
            TARG/<2.3819,1.1811,0>,<0,0,1>
            START ANG=0,END ANG=360
            ANGLE VEC=<1,0,0>
            DIRECTION=CCW
            SHOW FEATURE PARAMETERS=NO
            SHOW CONTACT PARAMETERS=NO
$$ NO,
            ------------------------------------------------
------------------------
            PROBE MOVES TO CLEARANCE PLANE.
            IT THEN PERFORMS A TIP ROTATION PRIOR TO
MEASURING FRONT FACE.
            ------------------------------------------------
------------------------

            MOVE/CLEARPLANE
            TIP/T1A90B-180, SHANKIJK=0, -1, 0, ANGLE=180
$$ NO,
            ------------------------------------------------
------------------------
            A NEW CLEARANCE PLANE IS DEFINED AT 3 UNITS AWAY
FROM FRONT FACE
            A PASSTHROUGH PLANE IS SET AT 2 UNITS ABOVE THE
TOP FACE
            ------------------------------------------------
------------------------

            CLEARP/YMINUS,-3,ZPLUS,2,ON
            MOVE/CLEARPLANE
$$ NO,
            ------------------------------------------------
------------------------
            PROBE MOVES ALONG THE PASS THROUGH PLANE TO 2ND
CLEARANCE PLANE
            ------------------------------------------------
------------------------

CIR2       =FEAT/CONTACT/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
            THEO/<2.5591,0.0787,-0.9843>,<0,-1,0>,0.3937
```

```
            ACTL/<2.5591,0.0787,-0.9843>,<0,-1,0>,0.3937
            TARG/<2.5591,0.0787,-0.9843>,<0,-1,0>
            START ANG=0,END ANG=360
            ANGLE VEC=<1,0,0>
            DIRECTION=CCW
            SHOW FEATURE PARAMETERS=NO
            SHOW CONTACT PARAMETERS=NO
$$ NO,
            ------------------------------------------------
------------------------
            PROBE MOVES TO 2ND CLEARANCE PLANE
            ------------------------------------------------
------------------------

            MOVE/CLEARPLANE
CIR3       =FEAT/CONTACT/CIRCLE/
DEFAULT,CARTESIAN,OUT,LEAST_SQR
            THEO/<2.5591,0,-0.9843>,<0,-1,0>,0.7874,0.25
            ACTL/<2.5591,0,-0.9843>,<0,-1,0>,0.7874,0.25
            TARG/<2.5591,0,-0.9843>,<0,-1,0>
            START ANG=0,END ANG=360
            ANGLE VEC=<1,0,0>
            DIRECTION=CCW
            SHOW FEATURE PARAMETERS=NO
            SHOW CONTACT PARAMETERS=NO
$$ NO,
            ------------------------------------------------
------------------------
            PROBE MOVES TO 2ND CLEARANCE PLANE
            ------------------------------------------------
------------------------

            MOVE/CLEARPLANE
            MOVE/INCREMENT,<0,0,6>
```

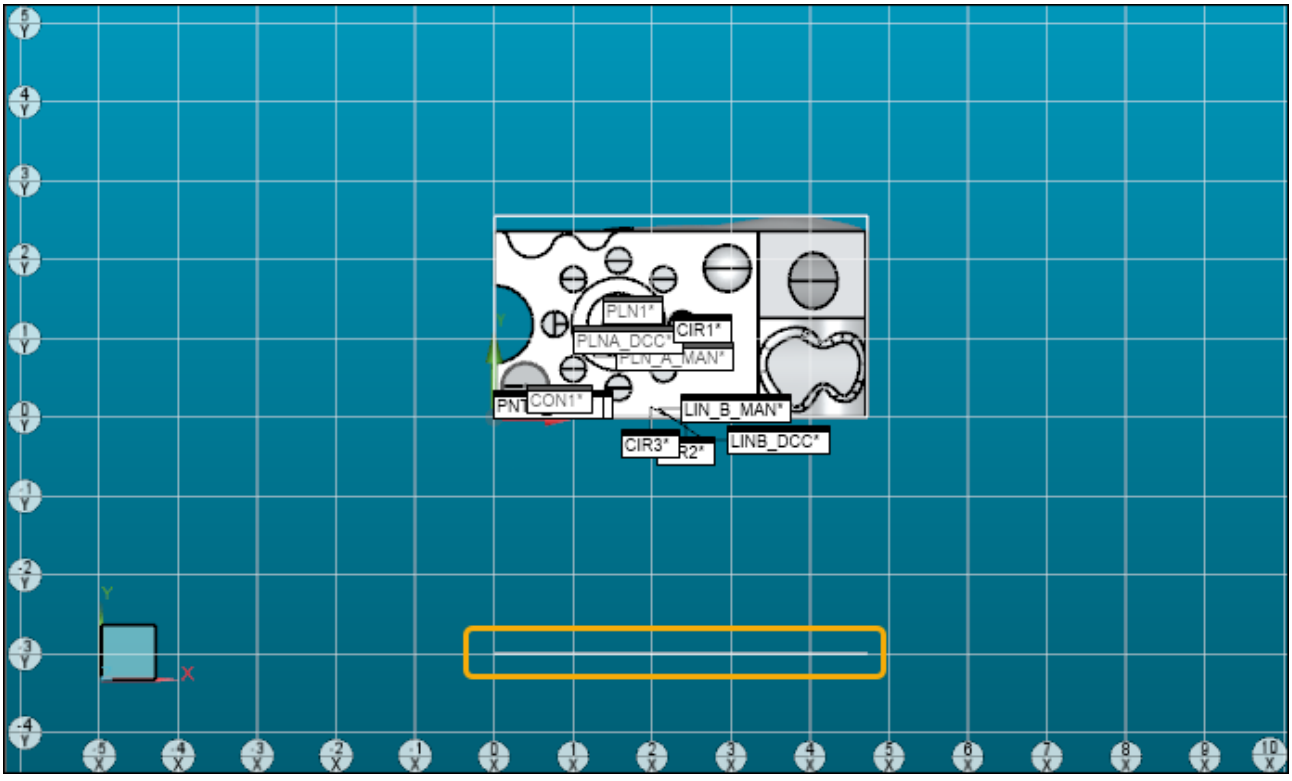
If you add additional moves or a loadprobe command before a tip change command, PC-DMIS deactivates the pass through plane.

When you make adjustments to commands in your measurement routine, be sure to review the path lines. This allows you to see the effect your changes make without having to run the measurement routine.

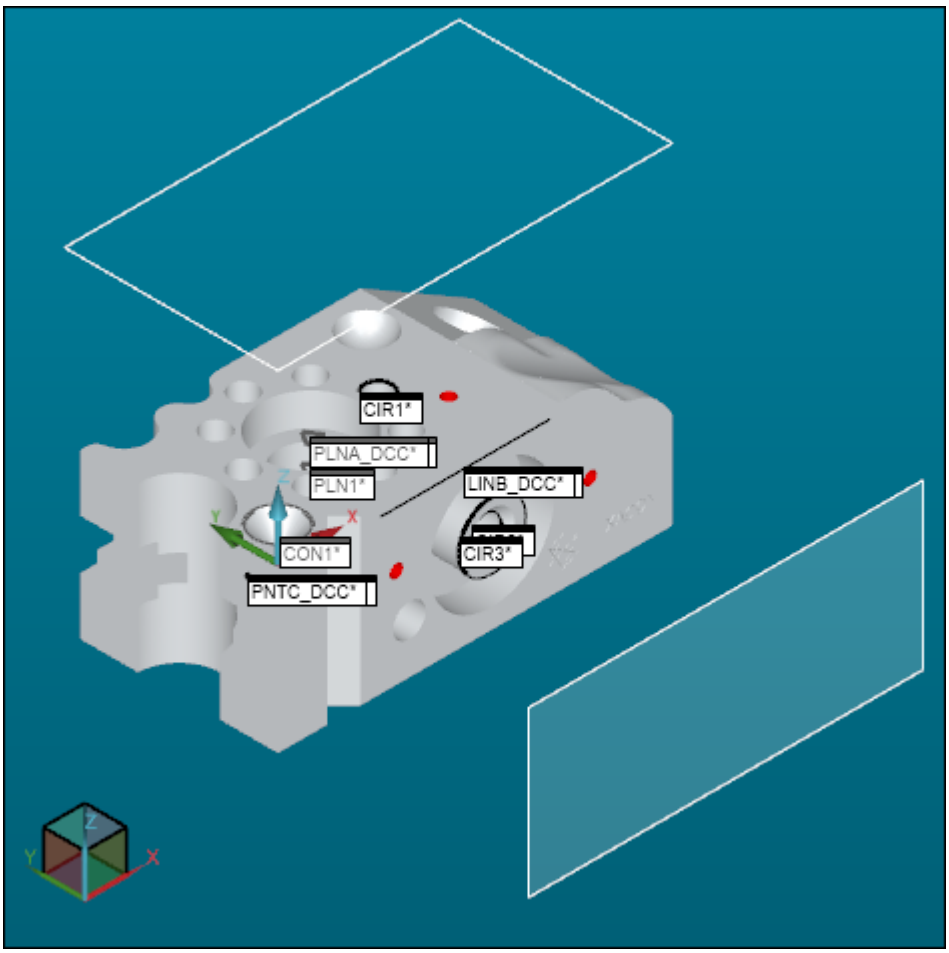
First Clearance Plane - YMINUS View



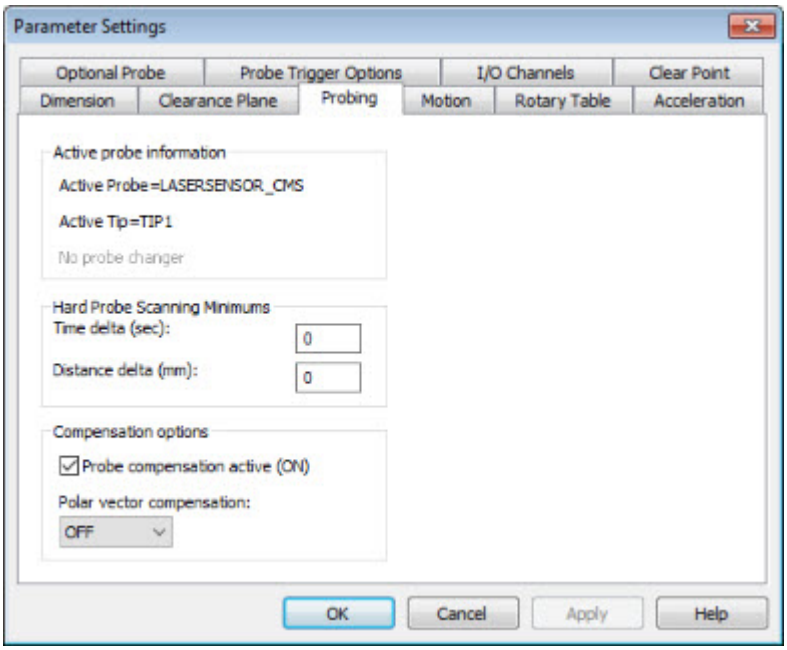
Second Clearance Plane - ZPLUS View



Both Clearance Planes - Isometric View



Parameter Settings: Probing Tab



Parameter Settings Dialog Box - Probing Tab

The **Probing** tab displays the current probe file, active tip, and probe port (if being used). It also allows you to set the **Hard Probe Scanning Minimums** for time delta and distance

delta, select the **Probe compensation active (ON)** check box, and select the **Polar vector compensation**.

The **Hard Probe Scanning Minimums** area is disabled for trackers that do not support this functionality.

To access the **Probing** tab:

1. Open the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Select the **Probing** tab.

Active Probe Information area

Active Probe

Active Probe=PH9

This line on the **Probing** tab displays the current probe file. For more information on selecting a different probe file or creating a new probe file name, see "Probe File Name" in the "Defining Hardware" chapter.

Active Tip

Active Tip=T1A0B0

This line on the **Probing** tab displays the current and active tip. For more information on the values and selecting, creating, and deleting tips, see "Active Tip List" in the "Defining Hardware" chapter.

The Edit window command line for this option reads:

```
TIP/active_tip_name
```

Probe in Port

This line on the **Probing** tab indicates the port number on the probe changer of the probe or stylus that you are currently using. This option is only available if you set up a probe changer.

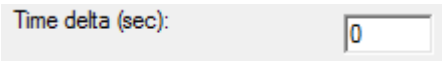
If you aren't using a probe changer, or if your current probe isn't in a probe changer, then PC-DMIS displays text informing you that your current probe or stylus isn't in the probe changer or that a probe changer isn't being used.

For information on adding probes or styli to a probe changer, see "Ports Tab" in the "Setting Up Probe Changer Options" topic.

Hard Probe Scanning Minimums Area

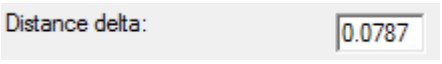
The **Hard Probe Scanning Minimums** area is disabled for trackers that do not support this functionality.

Time Delta (sec) box



This value allows PC-DMIS to reduce the amount of points that are scanned by deleting hits that are read in faster than the specified time delay in milliseconds.

Distance Delta box



This value allows you to reduce the measured data by deleting hits that are closer than the distance specified in millimeters. The reduction of hits happens as data comes from the machine. PC-DMIS only keeps the points that are set apart by more than the specified increments.

Notes

If Both are Greater than Zero

If both the time delta and distance delta values are greater than zero, PC-DMIS checks the amount of time elapsed and the distance and the probe moves. Whenever both the time AND the distance exceed the values specified, PC-DMIS accepts a hit.

If Both Are Zero

If both the time delta and distance delta values are set to zero, PC-DMIS uses the maximum-allowed sampling rate of the measurement device to accept hits.

Compensation Options area

Probe Compensation Active (On)

The **Probe Compensation Active (On)** check box allows PC-DMIS to compensate for the probe radius. If marked, and you click **OK**, this check box inserts a `PROBECOM/ON` command into the Edit window. PC-DMIS will compensate for the probe radius on each feature it measures. If cleared, then the command is set to `PROBECOM/OFF` in the Edit window. If you're using a Portable device, you can use **Insert | Parameter Change | Probe | Probe Compensation** to toggle this on or off as well.

Polar Vector Compensation

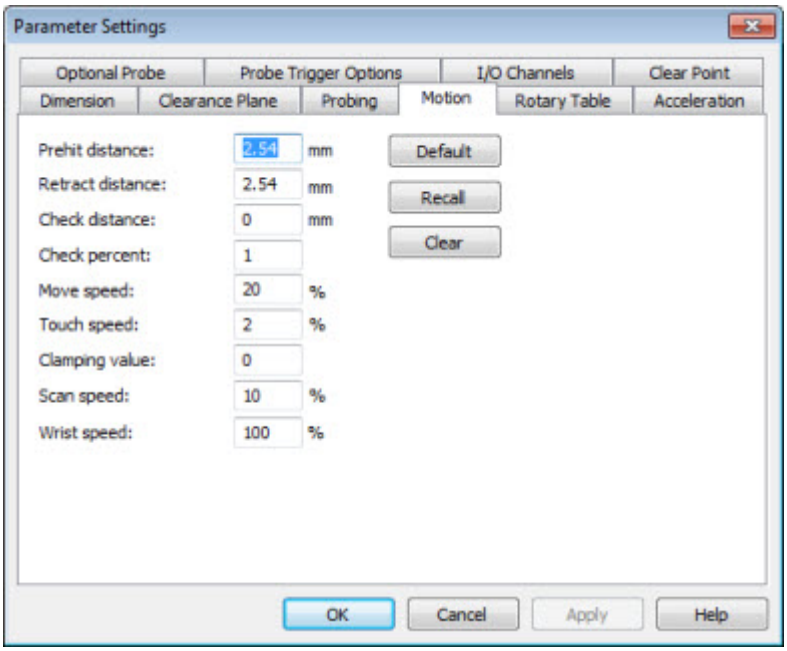
The **Polar Vector Compensation** drop-down list allows the measurement of Vector and Surface points to always be compensated along a Polar vector. The following options are available:

- **OFF** - Vector and Surface points behave normally.
- **XYPL** - Vector compensations for each Vector and Surface point will be 2D in the XY plane along a vector from the point to the current origin.
- **YZPL** - Vector compensations for each Vector and Surface point will be 2D in the YZ plane along a vector from the point to the current origin.
- **ZXPL** - Vector compensations for each Vector and Surface point will be 2D in the ZX plane along a vector from the point to the current origin.
- **3D** - Causes a polar vector compensation along a 3D vector from the point to the current origin.

The Edit window command lines for these options read:

```
POLARVECTORCOMP/ OFF
POLARVECTORCOMP/ XYPL
POLARVECTORCOMP/ YZPL
POLARVECTORCOMP/ ZXPL
POLARVECTORCOMP/ 3D
```

Parameter Settings: Motion Tab



Parameter Settings Dialog Box - Motion Tab

The **Motion** tab allows you to change the distances that the probe is allowed to travel when taking a hit. It also allows you to set the speed that PC-DMIS uses to take a hit and move from point to point.

To determine whether the speed boxes (**Move Speed**, **Touch Speed**, and **Scan Speed**) list the values as either **mm/sec** or as a percentage of the total allowed speed, select or clear the **Display Absolute Speeds** check box on the **Part/Machine** tab in the **Setup Options** dialog box.

To edit information in the **Motion** tab:

1. Access the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Click on the **Motion** tab. You'll see a variety of boxes.
3. Highlight the value you want to change.
4. Type a new value.
5. Click **Apply** or **OK**. PC-DMIS inserts any changed command into your measurement routine.

To reset the motion values to the original factory setting, click the **Recall** button. The **Recall** button returns the displayed motion values to the values stored in the PC-DMIS Settings Editor. If you click the **Defaults** button, the displayed values are saved to the

Settings Editor. For information on the PC-DMIS Settings Editor, view the "Modifying Setting Entries" section.

Changing Animation Speeds: If you want to modify your offline animation speeds, see the **Execution** area on the **Animation** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). In addition, see the "Executing and Debugging Measurement Routines Offline" topic in the "Working in Offline Mode" chapter.

Prehit Distance

The **Prehit Distance** box lets you enter the value for the machine prehit distance. This value determines the distance away from the theoretical hit location on the surface where PC-DMIS starts searching for the part. The machine travels at touch speed while traversing this distance to search for the part.

If necessary, PC-DMIS can automatically change this when the software takes hits within an arc or circle.

The Edit window command line for this option reads:

```
PREHIT/nnn.nnnn
```

Where `nnn.nnnn` is a number value for the distance.

For an example of how **Prehit Distance** and **Check Distance** work together, see the example in the "Check Distance" topic.

The Probe Readout window shows the probe location based on the probe's center. However, the machine uses the outer diameter of the probe to drive to the surface. This means the values in the Probe Readout always show a probe radius less than what you might expect when it moves to this distance.

Retract Distance

The **Retract Distance** box lets you enter the distance the probe retracts from the surface after taking a hit. If necessary, PC-DMIS can automatically change this value when it takes hits on an arc or circle.

The Edit window command line for this option reads:

```
RETRACT/nnn.nnnn
```

Where `nnn.nnnn` is a number value for the distance.

How the probe retracts depends on the CMM controller:

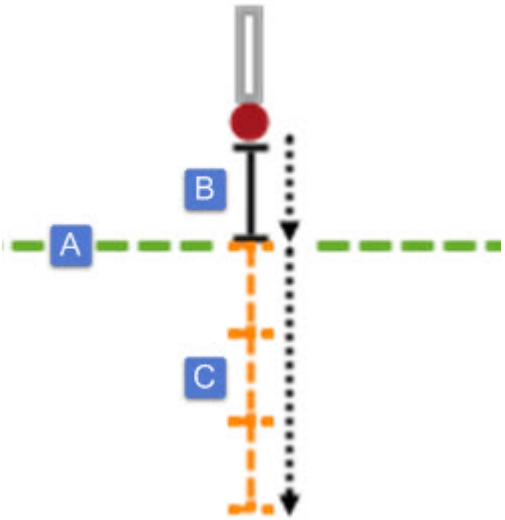
- **Some controllers handle the retract internally.** After a hit, the controller automatically retracts the probe, but it may only know the prehit distance, not the retract distance. In these cases, the controller retracts the probe to the prehit standoff, and PC-DMIS then issues an additional move to reposition the probe at the retract distance. If the retract distance is less than or equal to the prehit distance, the probe remains at the prehit standoff; PC-DMIS does not move the probe back toward the part after the controller's retract. *The retract value takes effect only when it is greater than the prehit.*
- **Some controllers do not retract on their own.** In these cases, PC-DMIS issues the move to perform the retract, and the distance is based on the probe tip's ball surface to the part's theoretical hit location.

When the controller does perform its own retract, the distance may be calculated from either the ball surface or the ball center to either the theoretical or measured hit location, depending on the particular controller.

Check Distance

The **Check Distance** box defines the distance past the theoretical hit location that the machine continues to search or check for the surface of the part. This distance is after it traverses the **Prehit Distance** value. The default value is zero.

- If this is zero, after the prehit distance, the machine searches to a maximum distance of three times the **Prehit Distance** value. For example:

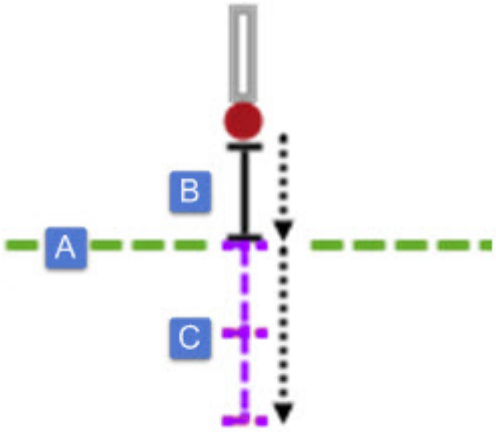


A. *Theoretical surface location.*

B. *Prehit distance.*

C. *Check distance (three times the prehit)*

- If this is a positive, non-zero number, after the prehit distance, the machine searches up to the maximum distance specified by the **Check Distance** value. For example, if the **Check Distance** is set to 2, the probe moves by two units:



A. *Theoretical surface location.*

B. *Prehit distance.*

C. *Check distance (moves the specified Check Distance value; in this example, two units)*

The Probe Readout window shows the probe location based on the probe's center. However, the machine uses the outer diameter of the probe to drive to the surface. This means the values in the Probe Readout always show a probe radius less than what you might expect when it moves to this distance.

The measurement units for the distance depend on the measurement system used for the part

The Edit window command line for this option reads:

```
CHECK/distance,percentage
```

Percentage Movement During Find Hole Operations

When defining the check distance for a Find Hole operation, you can tell PC-DMIS to move by a percentage of the check distance.

To do this:

1. Access the Edit window and place it in Command Mode.
2. Click on the `CHECK` command in the Edit window.
3. Press the Tab key to move to the second number.
4. Type a new percentage value. The default value is 1 which means 100% of the check distance. Thus .1=10%, .2=20%, .3=30% and so on.

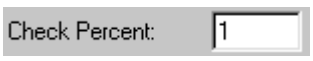
For example, in the following code, the .3 value represents 30% of the total check distance of twenty units:

```
CHECK/20,.3
```

For information on the Edit window command see "Check Distance" in the "Using the Edit Window" chapter.

For additional information on Find Hole, see the "Working with Contact Find Hole Properties" topic in the "PC-DMIS CMM" documentation.

Check Percent



The **Check Percent** value determines the percent of the total distance moved when performing a Find Hole operation. Be aware that if you type **1**, the value is equal to 100 percent. Thus, 100% would be entered as **1**, 25% would be entered as **.25**, and 10% would be **.10**.

Move Speed %

The **Move Speed** box lets you change the point to point positioning speed of the CMM. Depending on the state of the **Display absolute speeds** check box in the **Part/Machine** tab of the **Setup Options** dialog box, this will either be an absolute speed (mm/sec) or a percentage of the machine's defined top speed.

(See the "Measure" topic under "Defining Probes" in the "Defining Hardware" chapter for information regarding the move speed option and probe calibration.)

The Edit window command line for this option reads:

```
MOVESPEED/ nnn.nnnn
```

Where `nnn.nnnn` is a number value for the speed.

Touch Speed %

The **Touch Speed** box lets you change the speed at which the CMM takes hits. Depending on the state of the **Display absolute speeds** check box in the **Part/Machine** tab of the **Setup Options** dialog box, this is either be an absolute speed (mm/sec) or a percentage of the machine's defined top speed. This cannot exceed twenty percent.

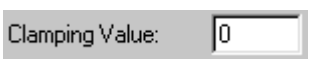
The Edit window command line for this option reads:

```
TOUCHSPEED/nnn.nnnn
```

Where `nnn.nnnn` is a number value for the speed.

> To understand the effects of changing either PREHIT or RETRACT distances and MOVE or TOUCH SPEED, it's important to understand how the options interact during DCC feature measurement. The order of events is that the machine moves toward the feature at MOVE SPEED. When it reaches PREHIT distance, it approaches the feature at TOUCH SPEED to take the hit. After it takes the hit, it backs away at TOUCH SPEED until it reaches the RETRACT DISTANCE. At this point, it proceeds to the next location at MOVE SPEED.

Clamping Value



The **Clamping Value** only works with the Leitz controller and the Leitz brand TTP. This value tells the controller how firmly to hold the probe on a Leitz CMM.

Depending on the weight of the probe tips, you may need to increase or decrease the clamping value.

- For a large tip weight, you may need to increase the clamping value.
- For a small tip weight, you may need to decrease the clamping value.

Scan Speed %

The **Scan Speed** box lets you change the speed at which the CMM scans the part. Depending on the state of the **Display absolute speeds** check box in the **Part/Machine** tab of the **Setup Options** dialog box, this is either an absolute speed (mm/sec) or a percentage of the machine's defined top speed.

The Edit window command line for this option reads:

```
SCANSPEED/nnn.nnnn
```

Where `nnn.nnnn` is a number value for the speed.

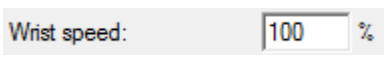
When scanning using the **DEFINED** mode of execution (see "Defined" discussed in the "Exec Controls area" topic in the "Scanning your Part" chapter), **Scan Speed** plays an important role in getting back data at increments specified by you. When you specify a very high **Scan Speed**, the CMM may execute the scan at the speed you requested, but the data from the CMM may not be spaced at the increments you specify.

> Assume the CMM gathers data at the rate of one Hit per 20 milliseconds in the **DEFINED** mode. If you specify an increment (minimum distance between hits) of 0.5 mm and a **Scan Speed** of 75 mm/sec, the CMM would give back points at every 1.5 mm. To avoid this you could reduce the **Scan Speed** to 15 mm/sec or 20 mm/sec which, at the rate of 1hit/20ms, would satisfy your requirements.

PC-DMIS will warn you about this case by displaying a message informing you that the supplied increment couldn't be measured at the specified scan speed. You will then be required to reduce your **Scan Speed** or increase the increment.

You can then select the **Scan Speed** setting and change to a suitable scan speed.

Wrist Speed %



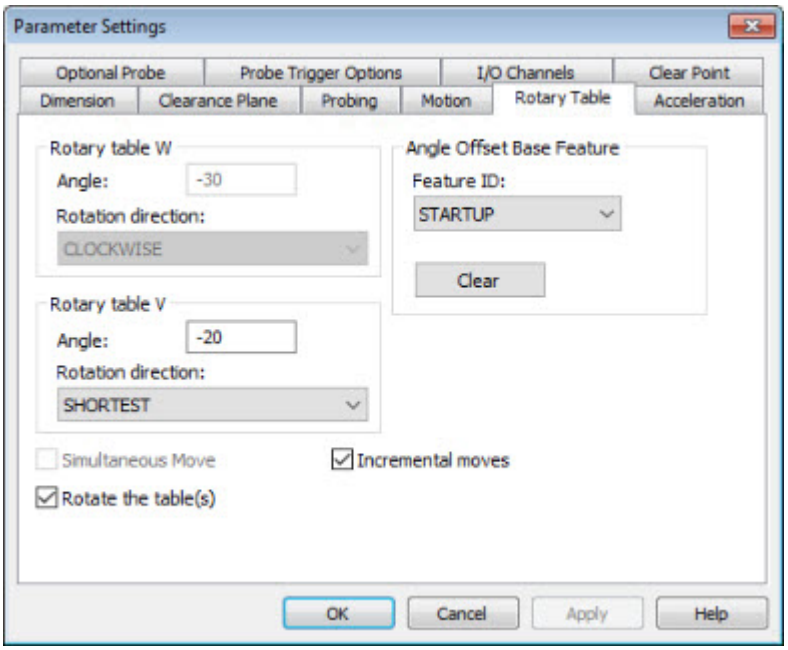
The **Wrist Speed** box lets you change the speed your measuring machine will use when rotating a probe's wrist (such as a CW43L). Depending on the state of the **Display absolute speeds** check box in the **Part/Machine** tab of the **Setup Options** dialog box,

this will either be an absolute speed (mm/sec) or a percentage of the machine's defined top speed.

The Edit window command line for this option reads:

```
WRISTSPEED/ nnn.nnnn
```

Parameter Settings: Rotary Table Tab



Parameter Settings Dialog Box - Rotary Table Tab

The **Rotary Table** tab allows you to rotate the current active rotary table by a defined angle and direction. You can also use it to automatically set the rotation based on a specified feature or alignment; or, you can combine the two and rotate to a specific feature or alignment and then rotate a relative angle on an offset based on the feature or alignment.

The **Rotary Table** tab becomes available when you select **Single Rotary Table**, **Dual Rotary Tables**, or **Stacked Rotary Tables** from the **Rotary Table Setup** dialog box. See "Defining the Rotary Table".

To insert a `MOVE/ROTAB` command that rotates the table, follow this procedure:

1. Access the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Select the **Rotary Table** tab.
3. Choose to rotate by a specific angle or to rotate to a feature (or both).
    - If you want to rotate by a specific angle, complete the **Rotary Table W** or **Rotary Table V** areas, and define the **Angle** and the **Rotation Direction**.

- If you want to rotate to a specific feature or alignment, complete the **Angle Offset Base Feature** area.

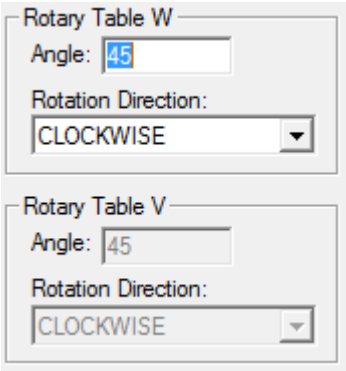
4. If you want to combine the individual movements of the W (rotary axis) and V (swivel axis) axes of a stacked rotary table into a single joint movement, select the **Simultaneous Move** check box.
5. If you want to rotate the table immediately, select the **Rotate the Table(s)** check box.
6. Click the **Apply** button. PC-DMIS inserts a `MOVE/ROTAB` command into the Edit window.

The Edit window command line for this option reads:
`MOVE/ROTAB, angle, DIRECTION,feature`

If you have a stacked configuration, the Edit window command line for this option reads:
`MOVE/ROTAB, angle, DIRECTION, angle2, DIRECTION2,feature`

Additionally, the **Rotary Table Setup** menu option is only available if your PC-DMIS license is programmed for rotary tables.

Rotary Table W / Rotary Table V



Parameter Settings Dialog Box - Rotary Table W and Rotary Table V Area

The **Rotary Table W** area and the **Rotary Table V** area let you control up to two rotary tables, table W and table V. PC-DMIS activates the area associated with the currently active rotary table. If you have a stacked rotary configuration, PC-DMIS activates both areas allowing you to type the angle and define the rotation direction for both tables at the same time. See "Defining the Rotary Table".

These areas contain the same options:

Angle box
This defines the angle of the table's rotation.

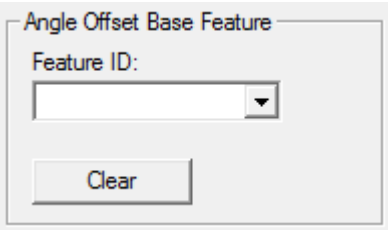
Rotation Direction list
In the **Rotation Direction** drop down list you can select the direction you want the table to rotate. Available options are:

Clock Wise: Rotates the table in a clockwise direction until it reaches the angle entered in the Rotate Table Angle box.

Counterclockwise: Rotates the table in a counterclockwise direction until it reaches the angle entered in the Rotate Table Angle box.

Shortest: Rotates taking the shortest route (either Clockwise or Counterclockwise) until it reaches the angle entered in the Rotate Table Angle box.

Angle Offset Base Feature



Parameter Settings Dialog Box - Angle Offset Base Feature Area

The **Angle Offset Base Feature** area lets you select a feature or alignment from the **Feature ID** list. The angle where the feature normal matches the machine Z (or where it matches as closely as possible for the hardware configuration) will become the 0 angle on the rotary table. This allows you to rotate to the desired feature or alignment without defining an initial angle. You simply specify the desired feature or alignment. The selected feature or alignment becomes the base feature (or the 0 angle) from which PC-DMIS rotates the table by a relative angle. Relative measurements like these are especially useful in camera-based vision environments where initial starting angles may not be known.

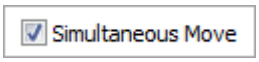
This works for both single and stacked rotary tables.

Feature ID
This list contains all the features and alignments in the measurement routine. It allows you to choose a feature or alignment to which the table will rotate.

Clear
This button clears the selected feature or alignment.

You can use the `MOVE/ROTAB` command to rotate the table to the feature or alignment during measurement routine execution. See "Inserting a Move Rotary Table Command" in the "Inserting Move Commands" chapter.

Simultaneous Move



Parameter Settings Dialog Box - Simultaneous Move Option

The **Simultaneous Move** option allows you to combine the individual movements of the W (rotary axis) and V (swivel axis) axes of a stacked rotary table into a single joint movement.

The result is a faster positioning and thus an overall faster processing of the measurement routine.

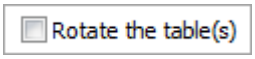
This option is only available if you have an FDC and an activated stacked rotary table.

If you select the **Simultaneous Move** check box, the individual paths of the W and V axes must both have either the SHORTEST direction selected or not selected for the simultaneous movement of the tables. That is, if the **Simultaneous Move** check box is selected, but one of the axis has the SHORTEST direction selected and the other axis has CLOCKWISE or COUNTERCLOCKWISE direction selected, PC-DMIS executes the rotary table moves one at a time.

If you do not select the **Simultaneous Move** check box, the software executes the rotary table moves individually.

The V axis (swivel axis) cannot rotate 360° due to the software limitations in the FDC configuration. This restriction does not always allow clockwise or counterclockwise movement.

Rotate the Table(s)



Parameter Settings Dialog Box - Rotate the table(s) Option

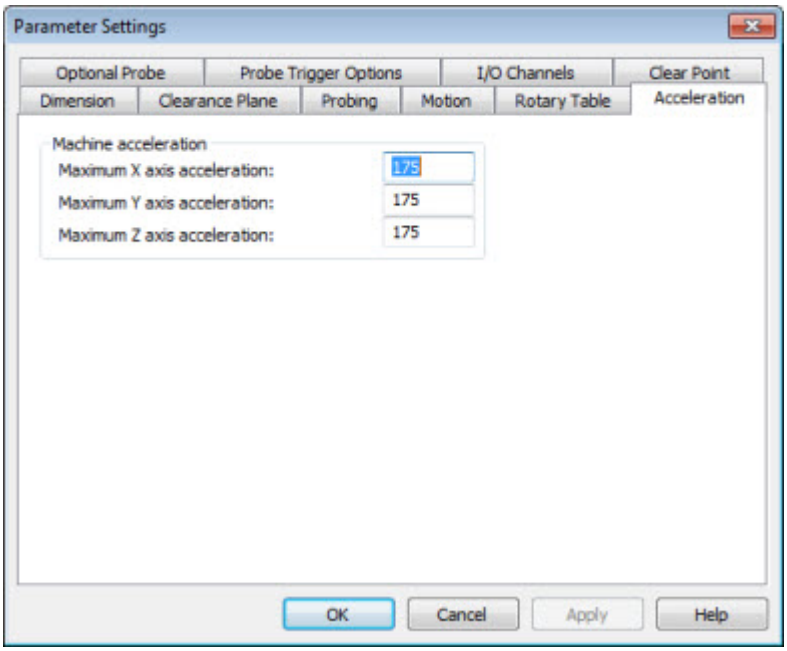
The **Rotate the Table(s)** check box enables you to activate the current table's rotation by the specified **Angle** value once you select the **Apply** or **OK** button.

Incremental Moves

Parameter Settings Dialog Box - Incremental moves Check Box

The **Incremental moves** option applies to the `MOVE/ROTAB` command. When you select this option, instead of using the NORMAL parameter for absolute values, you can use incremental values for the angle.

Parameter Settings: Acceleration Tab



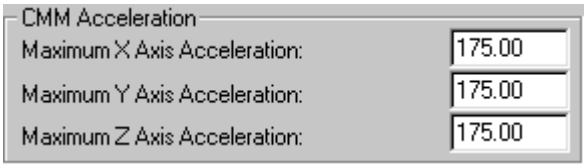
Parameter Settings Dialog Box - Acceleration Tab

The **Acceleration** tab displays additional editing capabilities for both CMM and Table motion.

To access the **Acceleration** tab:

1. Access the **Parameter Settings** dialog box from the main menu (**Edit | Preferences | Parameters**).
2. Select the **Acceleration** tab.

CMM Acceleration



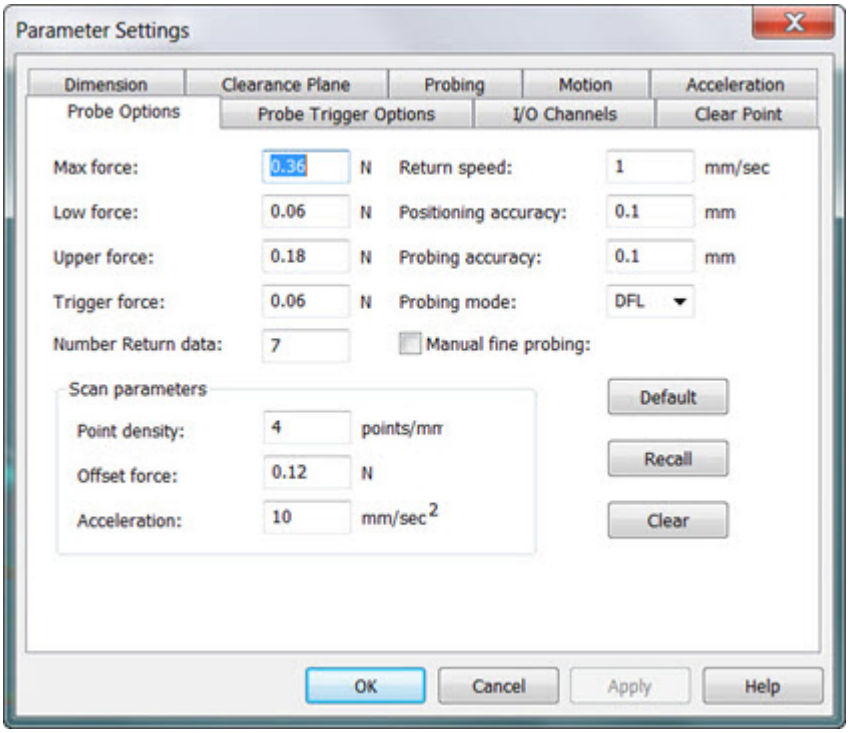
The **CMM Acceleration** area of the **Acceleration** tab allows you to change the maximum acceleration (at mm/sec2) along the X, Y, or Z axis of the CMM. Available options are:

Maximum X Axis Acceleration
The number in this box represents the maximum acceleration that the CMM will take when traveling along the X axis.

Maximum Y Axis Acceleration
The number in this box represents the maximum acceleration that the CMM will take when traveling along the Y axis.

Maximum Z Axis Acceleration
The number in this box represents the maximum acceleration that the CMM will take when traveling along the Z axis.

Parameter Settings: Probe Options Tab



Parameter Settings Dialog Box - Probe Options Tab

The **Probe Options** tab gives you additional capabilities for controlling analog probes. These capabilities include the following probing-related values:

- Max force
- Low force
- Upper force
- Trigger force
- Number Return data
- Return speed
- Positioning accuracy
- Probing accuracy
- Probing mode
- Manual fine probing

It also includes the following scan-related values and general purpose buttons:

- Point density
- Offset force
- Acceleration
- **Default** button
- **Recall** button
- **Clear** button

Each of these is described below in greater detail.

To access the **Probe Options** tab:

1. Access the **Parameter Settings** dialog box from the main menu (**Edit | Preferences | Parameters**).
2. Select the **Probe Options** tab.

The values on the **Probe Options** tab are machine-specific. With the exception of the **Manual fine probing** check box and the **Point density** box, *they generally shouldn't be changed*. Check with your machine's manufacturer before you make changes.

Click **Apply** or **OK** to insert an `OPTIONPROBE` command into the Edit window.

Max Force

The **Max Force** box allows you to enter the maximum force that a probe takes before an error occurs, thus stopping measurement.

A "newton" is a unit of force. One newton is the force needed to accelerate a mass of 1 kilogram to a speed of 1 meter per second per second.

During an analog probing cycle, when the probe first touches the part it continues to move into the part until it reaches this Max Force value. It then reverses direction and moves away from the part. This movement into the part, after touching the part, is sometimes referred to as the *contact force*. The value is specified in newtons. In a normal deflection mode (DFL) probing cycle, the controller collects data as the probe moves away from the part.

Low Force

The **Low Force** box allows you to enter the minimum force required to determine when the machine is in contact with the object being measured.

For a normal deflection mode (DFL) probing cycle, this is the force at which the controller stops collecting data. The value is specified in newtons.

Upper Force

The **Upper Force** box is the upper limit for a measurement. When this force is reached the machine moves back, away from the object being measured.

For a normal deflection mode (DFL) probing cycle, this is the force at which the controller starts collecting data. The value is specified in newtons.

Trigger Force

The **Trigger Force** box allows you to enter the force at which a measurement reading is taken.

For a normal deflection mode (DFL) probing cycle, this is the force at which the Actual Point (APT) will be calculated and returned to PC-DMIS. The value is specified in newtons. Not all analog probes/controllers use this input.

Return Data

The **Number Return data** box allows you to enter the number of readings taken when the machine is moving away from the object being measured.

This value defines the minimum number of data to be collected within the **Probe Options** tab defined by the **Upper Force** and **Lower Force** values.

Return Speed

The **Return Speed** box allows you to set the value, which determines the speed at which the probe comes off the object being measured. This is specified in mm/sec.

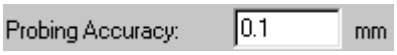
Positioning Accuracy

The **Positioning Accuracy** box is a parameter specific to the Leitz interface. The value submitted tells the CMM how carefully it should remain on the approach vector as the probe is moved into the part for a measurement.

With smaller values the difficulty for the machine to acquire the desired location increases. However, using a smaller value provides a more accurate measurement. This value is always in millimeters.

This should normally be left at the default value.

Probing Accuracy



The **Probing Accuracy** box allows you to determine the accuracy required to take a measurement. If this value is not met, then no measurement is taken, and an error is given. It is specified in mm and should normally be left at the default value.

Probing Mode

This box specifies the type of probing cycle used. The most common cycle is deflection mode (DFL). Other cycles, such as soft probing (SFT), may also be supported by some analog probes/controllers. In some cases, the probe /controller may not have multiple modes, and this value is ignored.

Type or select the probing cycle that you want to use.

Manual Fine Probing

If the **Manual Fine Probing** check box is selected, then when a manual probing point is taken, the controller automatically switches to DCC Mode while moving away from the part in order to use the normal deflection probing cycle. This may result in slower manual probing but improves accuracy.

While recent machines with analog probing systems may support **Manual Fine Probing**, not all analog probes/controllers support this mode of manual probing. In those cases, PC-DMIS ignores this check mark. Your CMM controller's manufacturer will know if your controller supports this option.

Point Density

The **Point Density** box allows you to set the number of readings to take per millimeter of measurement during a scan.

If you give a scan increment smaller than the **Point Density** defined in the entry in the Settings Editor, PC-DMIS displays a warning explaining that the minimum increment is smaller than Point Density for scans. You are then asked to verify increment settings in the dialog box.

You can then change the scan's **Point Density** value to a suitable number.

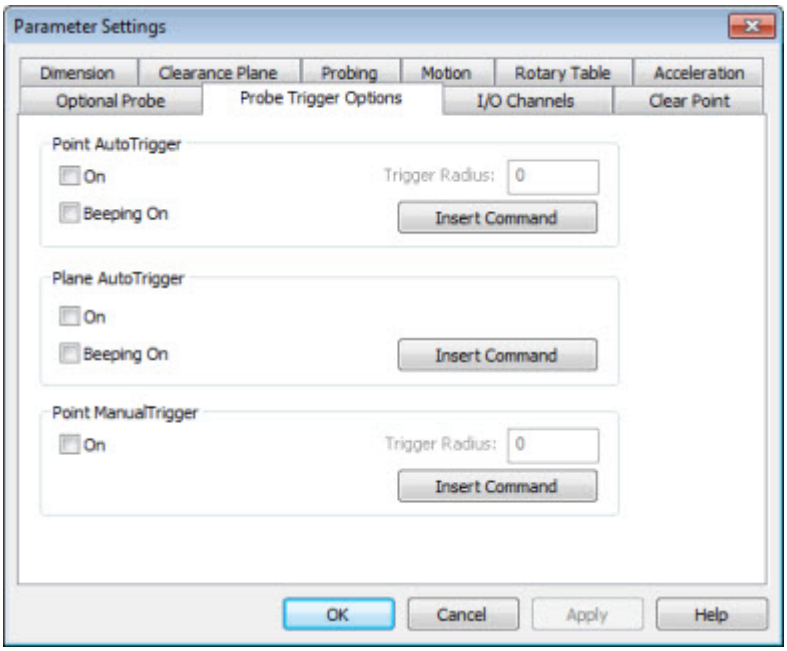
Offset Force

This allows you to specify what level of force to maintain during a scan. The value is specified in newtons.

Acceleration

This allows you to specify what acceleration to use during a scan. The value is specified in mm/sec/sec.

Parameter Settings: Probe Trigger Options Tab



Parameter Settings Dialog Box - Probe Trigger Options Tab

The **Probe Trigger Options** tab allows you to determine tolerance zones and then insert `POINT AUTOTRIGGER`, `PLANE AUTOTRIGGER`, and `POINT MANUAL TRIGGER` commands into the Edit window. These commands trigger a hit when certain conditions are met.

Only manual CMMs with certain interfaces support these probe trigger options. These interfaces include Faro, Romer, Garda, Leica, and Polar.

To access this tab:

1. Select **Edit | Preferences | Parameters** to display the **Parameter Settings** dialog box.
2. Select the **Probe Trigger Options** tab.

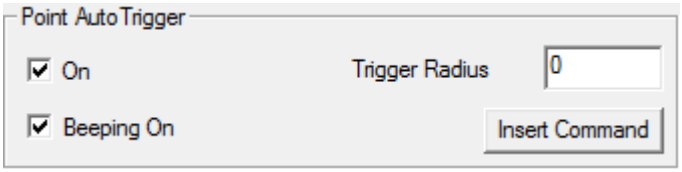
Supported Features

These trigger commands function with these supported features:

- Auto Features: Circle, Ellipse, Edge Point, Round Slot, Square Slot, Notch Slot, and Polygon
- Measured Features: Circle, Line, and Round Slot

In addition, the `POINT AUTOTRIGGER` command supports the Auto Vector Point and Measured Point features.

Point AutoTrigger area



With the **Point AutoTrigger** area, you can insert `POINT AUTOTRIGGER/` commands into the Edit window with a tolerance zone.

The `POINT AUTOTRIGGER/` command tells PC-DMIS to automatically take a hit when the probe enters a tolerance zone at a specified distance from the original hit location. For example, if the tolerance zone, the **Radius** value, is set to 2 mm, a hit is taken when the probe is within 2 mm of the hit location.

You can use this option with manual machines; instead of pressing a button to take a hit, you can place `POINT AUTOTRIGGER/` commands at any standard location in the Edit window.

On

Selecting the **On** check box activates the point auto trigger command. Commands in the Edit window that follow the inserted `POINT AUTOTRIGGER/` command and require you to take a hit will then have the hit automatically taken when the probe enters the defined tolerance zone. The **Trigger Radius** capability is disabled until you turn the option back on.

If you do *not* select this check box and you click the **Insert Command** button, PC-DMIS inserts the command line into the Edit window, but it doesn't activate the command.

Beeping On

Selecting the **Beeping On** check box activates a beeping sound associated with your `POINT AUTOTRIGGER/` command. The closer you get to the target, the more frequently your computer emits beeps.

Trigger Radius

The **Trigger Radius** box allows you to type a tolerance zone value. When the probe moves into this tolerance zone, it automatically and immediately takes a hit.

Insert Command

Clicking the **Insert Command** button inserts the `POINT AUTOTRIGGER/` command into the Edit window for the current measurement routine. This command line reads:

```
POINT AUTOTRIGGER/ TOG1, TOG2, RAD
```

TOG1: This toggle field corresponds to the **On** check box. It displays either ON or OFF.

TOG2: This toggle field corresponds to the **Beeping On** check box. It displays either ON or OFF.

RAD: The radius field contains the value for the tolerance zone, and it corresponds to the **Trigger Radius** box. This value is the distance from the actual point that PC-DMIS takes the hit.

Plane AutoTrigger area

With the **Plane AutoTrigger** area, you can insert a `PLANE AUTOTRIGGER/` command into the Edit window. The `PLANE AUTOTRIGGER/` command tells PC-DMIS to automatically take a hit when the probe passes the plane defined by the surface normal of a supported feature at the level of the defined depth. For Auto features, this defined location is adjusted based on options such as sample hits or RMEAS features. As the probe center passes from one side of the plane to the other, the probe triggers and the hit is taken.

You can use this command with manual machines; instead of pressing a button to take a hit, you can place `PLANE AUTOTRIGGER/` commands at any standard location within the Edit window.

This command only works in Online mode. If you use the `POINT AUTOTRIGGER/` command, it takes precedence over the `PLANE AUTOTRIGGER/` command.

As defined above, PC-DMIS automatically takes a hit when the probe passes the plane. However, if you are using a Faro or a Romer machine, the probe does not trigger again until you press the **Accept** or **Release** button. You must press this button after each registered hit to continue.

On

Selecting the **On** check box activates the `PLANE AUTOTRIGGER/` command. Commands in the Edit window that follow the inserted `PLANE AUTOTRIGGER/` command and require you to take a hit then have the hit automatically taken when the probe center passes the plane defined by the surface normal and depth of the feature.

If you do *not* select this check box and you click the **Insert Command** button, PC-DMIS inserts the command line into the Edit window but doesn't activate the command. The `PLANE AUTOTRIGGER/` command does not function until the option is turned on.

Beeping On

Selecting the **Beeping On** check box activates a beeping sound associated with your `PLANE AUTOTRIGGER/` command. The closer the probe gets to the target, the computer emits the beeping sound more frequently.

Insert Command

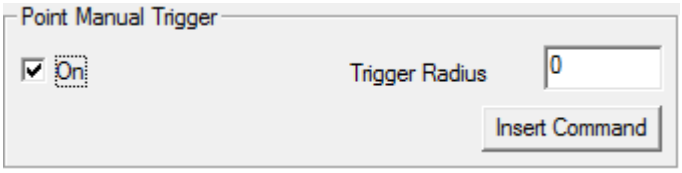
Clicking the **Insert Command** button inserts the `PLANE AUTOTRIGGER/` command into the Edit window for the current measurement routine.

```
PLANE AUTOTRIGGER/ TOG1,TOG2
```

TOG1: This toggle field corresponds to the **On** check box. It displays either ON or OFF.

TOG2: This toggle field corresponds to the **Beeping On** check box. It displays either ON or OFF.

Point Manual Trigger area



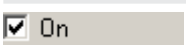
With the **Point Manual Trigger** area, you can insert a `POINT MANUAL TRIGGER/` command into the Edit window.

The `POINT MANUAL TRIGGER/` command tells PC-DMIS to only accept a manual hit when it is within the specified tolerance zone.

You can use this option with manual machines; when PC-DMIS prompts you to take a hit, trigger the probe as you wish. Each trigger will be evaluated to see if it is within the cylindrical trigger tolerance zone. If it is *not,* you will receive an error in the **Machine Errors** list of the **Execution** dialog box. PC-DMIS will then ask you to take the hit again. You can place `POINT MANUAL TRIGGER/` commands at any standard location within the Edit window.

This option works only in Online mode.

On



Selecting the **On** check box activates the `POINT MANUAL TRIGGER/` command. Commands in the Edit window that follow the inserted `POINT MANUAL TRIGGER/` command and require you to take a hit will then only accept the hit when the probe enters

the defined tolerance zone. The **Trigger Radius** capability is disabled until the option is turned on.

If you do *not* select this check box and you click the **Insert Command** button, PC-DMIS inserts the command line into the Edit window but doesn't activate the command. **Insert Command**

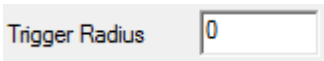
Clicking the **Insert Command** button inserts the `POINT MANUAL TRIGGER/` command into the Edit window for the current measurement routine with the following options.

```
POINT MANUAL TRIGGER/ TOG1, RAD
```

TOG1: This toggle field corresponds to the **Use Trigger Tolerance** check box. It displays either ON or OFF.

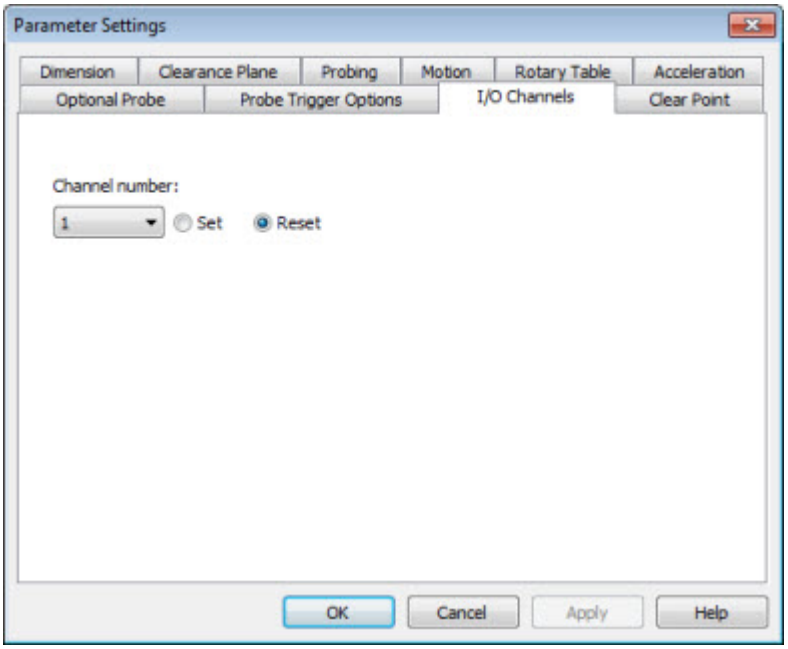
RAD: The radius field contains the value for the tolerance zone. This corresponds to the **Trigger Radius** box. This value is the distance from the actual point that PC-DMIS accepts the hit.

Trigger Radius



The **Trigger Radius** box allows you to type a tolerance radius value. When the probe is triggered, PC-DMIS will check to see if the probe is within this tolerance zone. If it is, the hit will be accepted. It is it *not*, you will be asked to take another hit.

Parameter Settings: I/O Channels Tab



Parameter Settings Dialog Box - I/O Channels Tab

*Currently, the options on the **I/O Channels** tab only function on DEA machines. Other machine types may be added in the future.*

With the **I/O Channels** tab, you can select options related to the use of controller I/O channels and place an `IOCHANNEL/` command into the Edit window that define the state of the controller.

Some machine controllers are equipped with I/O channels that can be `SET` to an ON state (a value of 1) or `RESET` to an OFF state (a value of 0). The `IOCHANNEL/` command tells PC-DMIS to set the state as specified.

To edit information in the **I/O Channels** tab:

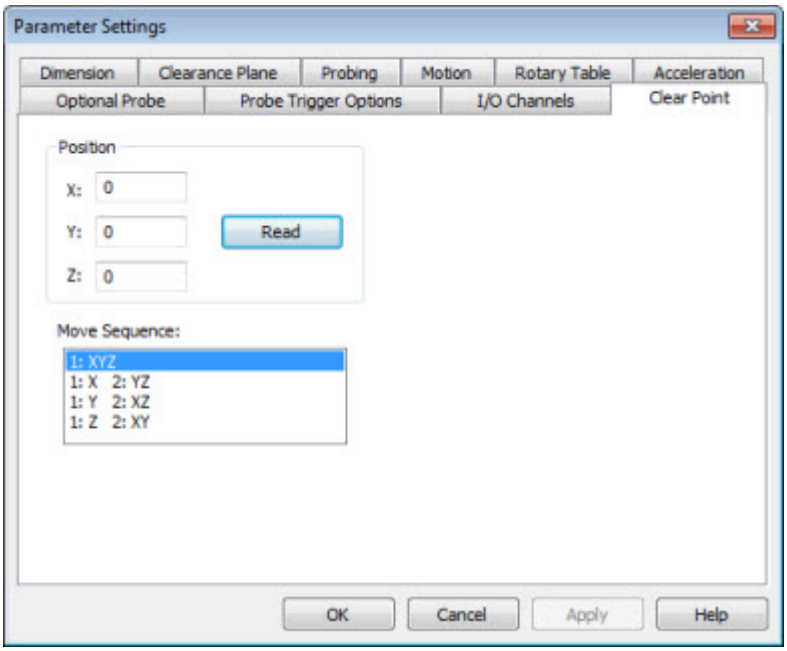
1. Access the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Click on the **I/O Channels** tab.
3. Make any changes.
4. Click **Apply** or **OK**.

Channel - Specifies the channel number that you will **Set** or **Reset**.

Set - Inserts an `IOCHANNEL/SET` command into the measurement routine. When PC-DMIS executes this command, the state of the specified channel number is set to 1.

Reset - This option inserts an `IOCHANNEL/RESET` command into the measurement routine. When PC-DMIS executes this command, the state of the specified channel number is set to 0.

Parameter Settings: Clear Point Tab



Parameter Settings Dialog Box - Clear Point Tab

The **Clear Point** tab lets you define a single move point position in machine coordinates, called a clear point, for your machine. This causes your machine to move the end of the arm to the defined position. This is intended to be a safety position the arm moves to when using a probe changer. This differs from standard MOVE/POINT command in that it uses **Move Sequence** list to specify the form of the move and the move position is absolute to the machine.

Position area - The **XYZ** boxes define the location of the move point. When you click the **Read** button, PC-DMIS reads the machine's current position and enters the coordinates into the **XYZ** fields.

Move Sequence list - This list lets you choose the form of the move to take to arrive at the clear point. It contains these options:

1: XYZ - The machine moves in a straight line to the clear point position.

1: X 2: YZ - The machine first moves the X axis and then the YZ axis.

1: Y 2: XZ - The machine first moves the Y axis and then the XZ axis.

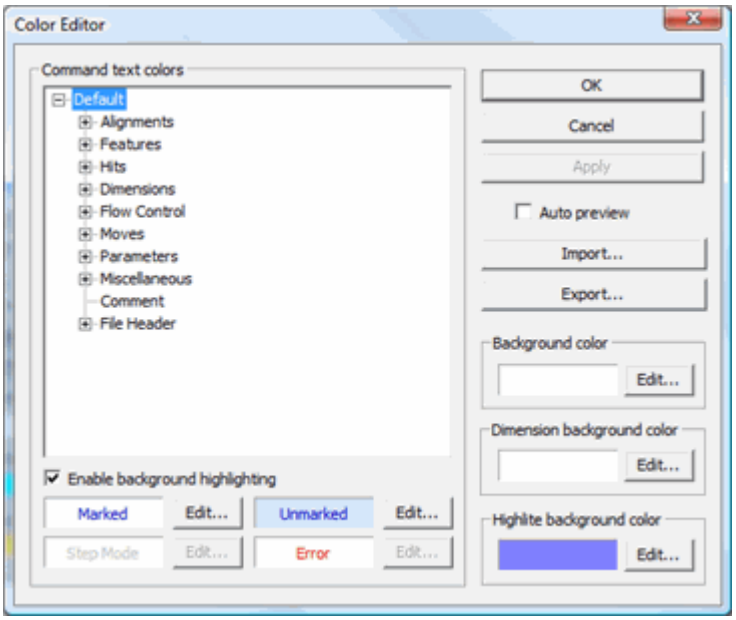
1: Z 2: XY - The machine first moves the Z axis and then the XY axis.

To insert this command, select the **Insert | Move | Clear Point** menu item. PC-DMIS inserts a `MOVE/CLEARPOINT` command in the Edit window. When you execute this command, PC-DMIS uses the selected move sequence to move to the defined clear point.

Setting Up the Edit Window

PC-DMIS lets you determine the Edit window's appearance as well as what information to display within some of the Edit window's modes.

Defining Edit Window Colors



Color Editor Dialog Box

The **Edit | Preferences | Edit Window Colors** menu option displays the **Color Editor** dialog box. You must be in Command Mode or DMIS Mode to select this menu item.

The **Color Editor** dialog box allows you to define the colors used for the Edit window text and background when you have the Edit window in Command and DMIS Modes. You can also export your color scheme to use on different computer systems or import external color schemes.

The **Color Editor** dialog box only specifies the colors in textual modes of the Edit window such as the Command and DMIS.

There are four principal colors that you can configure for all commands or for individual commands. To determine whether to show these colors as a highlighted background color or as text color, select or clear the **Enable Background Highlighting** check box. These are listed in the four boxes in the **Command Text Colors** area.

- Marked
- Unmarked
- Step Mode

- Error

By default, PC-DMIS selects the **Enable Background Highlighting** check box which makes the boxes appear with these colors:

If you clear this check box to enable text coloring instead, PC-DMIS shows the boxes as follows:

Color Editor dialog box showing the principal colors.

Marked - PC-DMIS uses this color on features that you mark for execution. PC-DMIS always executes some features such as alignments are always executed and are always displayed in the Marked color.

Unmarked - PC-DMIS uses this color for unmarked features. This color is the default color. If no other condition is present, then PC-DMIS uses the default color.

Step Mode - PC-DMIS uses this color to highlight the next execution line of code that occurs when executing a measurement routine that contains breakpoints.

Error - PC-DMIS uses this color for commands that are in error or measurements that are outside of their tolerance limits. For example, if the measurement routine calls for a tip that is not defined in the probe database, PC-DMIS colors this tip text with the Error color.

You can also change background colors for the Edit window, dimensions, and the highlight color.

Background Color - This sets the Edit window's background color.

Dimension Background Color - This sets the dimension's reporting boxes' background color.

Highlight Background Color - This sets the background color when dragging the mouse to select a command or group of commands.

You might be interested in color schemes created by other users. You can search for those on the PC-DMIS Community Group. For example, to get an unsupported dark theme created by one of our writers, go to this post.

Related Topics:

Understanding the Default Edit Window Colors and Formatting

To change the command text colors used

1. Select **Edit | Preferences | Edit Window Colors** from the menu bar. The **Color Editor** dialog box opens.
2. Click the **Auto Preview** check box. This allows you to preview your changes inside the Edit window as you make them.
3. Select a specific command or a parent command from the list of commands in the **Command Text Colors** area. Click on the plus signs to expand the list to view additional sub commands. This allows you to set the principal colors (**Marked**, **Unmarked**, **Step Mode**, and **Error**) for specific commands or parent commands. To set the changes for *all* command text, select **Default** from the top of the list.
4. Simply click the **Edit** button for the text or background color in the **Command Text Colors** area. A **Color** selection box appears.
5. Select the new color, or customize a color by selecting the **Define Custom Colors** button.
6. Click the **OK** button. The **Color** selection box closes. If you selected a parent command, PC-DMIS presents you with an option to set all child commands under the parent command to use the same color as well. At this prompt, select either **Yes** or **No**.
7. When you finish defining your colors, click the **Apply** button to see the changes without closing the **Colors** dialog box.
8. Click the **OK** button to apply the changes and close the **Color Editor** dialog box.

PC-DMIS applies the color changes immediately.

To change the background colors used

1. Select **Edit | Preferences | Edit Window Colors** from the menu bar. The **Color Editor** dialog box opens.
2. Click the **Auto Preview** check box. This allows you to preview your changes inside the Edit window as you make them.
3. Simply click the **Edit** button for the background or highlight color from the **Background Color** area. A **Color** selection box appears.
4. Select the new color, or customize a color by selecting the **Define Custom Colors** button.

5. Click the **OK** button. The **Color** selection box closes.
6. When finished defining your colors, click the **Apply** button to see the changes without closing the **Colors** dialog box.
7. Click the **OK** button to apply the changes and close the **Color Editor** dialog box.

The color changes are immediately made.

To export a color scheme

1. Select **Edit | Preferences | Edit Window Colors** from the menu bar to open the **Color Editor** dialog box.
2. Make changes to your color scheme as needed.
3. Click the **Export** button. A **Save As** dialog box appears. This dialog box allows you to store your Edit window color scheme as a color file (a file with a *.clr* extension).

Save As Dialog Box

4. Navigate to where you want to store this file.
5. Type a name for the stored color file in the **File name** box.
6. Click **Save**.

To import and use a color scheme

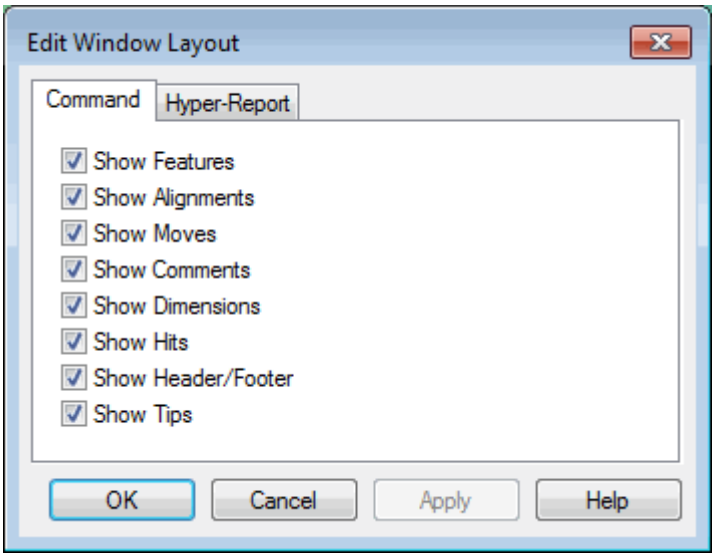
1. Select **Edit | Preferences | Edit Window Colors** from the menu bar. The **Color Editor** dialog box opens. Click the **Auto Preview** check box. This allows you to preview your changes inside the Edit window once you select the color scheme.
2. Click the **Import** button. An **Open** dialog box appears.
3. Navigate to the color scheme file (a file with a *.clr* extension).
4. Select the file and click **Open**. The **Open** dialog box closes.
5. Click **Apply** and then **OK** to use the color scheme you just imported.

Defining the Edit Window Layout

The **Edit | Preferences | Edit Window Layout** menu option displays the **Edit Window Layout** dialog box. The dialog box contains these tabs:

- **Command** tab - This tab lets you hide or show certain commands in the Edit window. For more information, see "Command Mode Display Options".
- **HyperReport** tab - This tab usually does not appear, but it might for some older measurement routines. It determines if PC-DMIS automatically loads a legacy HyperView report once a measurement routine finishes execution. For more information, see "Hyper-Report Configuration Options".

Command Mode Display Options



Edit Window Layout Dialog Box - Command Tab

The **Command** tab of the **Edit Window Layout** dialog box (**Edit | Preferences | Edit Window Layout**) allows you to choose which of the following display options are available for Command Mode.

Show Features
This option displays the features measured by the measurement routine.

Show Alignments
This option shows alignment changes as they occur during the measurement routine. It displays all alignment changes that occur in the dimension or feature lists.

Show Moves
This option shows any moves that have been added to the measurement routine.

Show Comments
This option shows any comments that have been added to the measurement routine.

(See "Inserting Programmer Comments" in the "Inserting Report Commands" chapter for additional information.)

Show Dimensions
This option displays the specified dimension for the features inspected by PC-DMIS. It is displayed in the selected format using the `FORMAT` command described in the "Dimension Format" topic in the "Using the Edit Window" chapter.

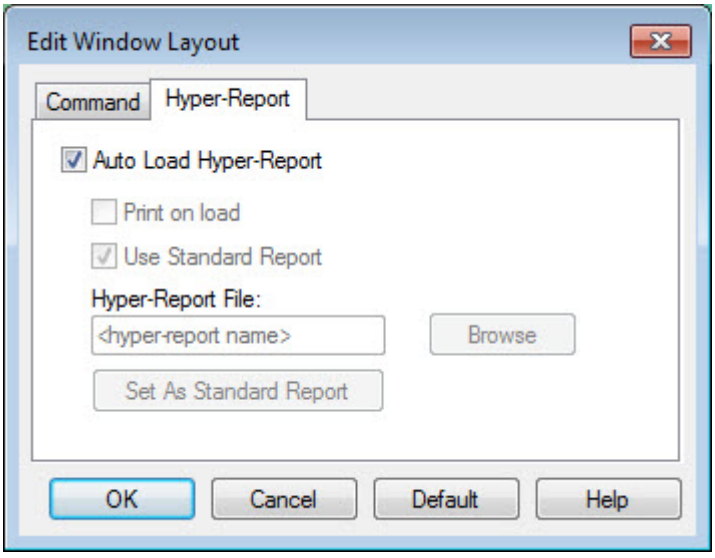
Show Hits
This option displays each hit.

Show Header / Footer
This option displays the header or footer from the LOGO.DAT, HEADER.DAT, and ELOGO.DAT files. (See "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window" chapter for information on altering these files.)

Show Tips
This option displays the tip file names used to inspect the part.

For information on working with Command Mode, see the "Working in Command Mode" topic in the "Using the Edit Window" chapter.

Hyper-Report Configuration Options



Edit Window Layout Dialog Box — Hyper-Report Tab

> This older tab remains to support legacy HyperView reports. It only appears if the **Auto Load Hyper-Report** check box is marked. Once you clear that check box, the tab never reappears for the current measurement routine.

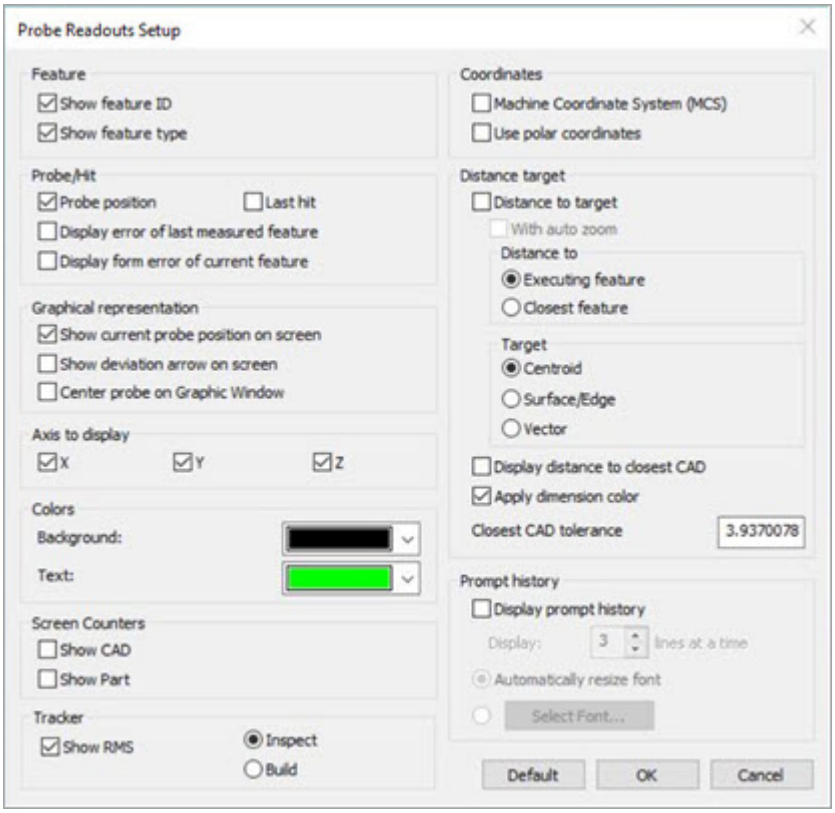
The **Auto Load Hyper-Report** check box tells PC-DMIS to load a specific legacy HyperView report (.rpt) as soon as execution finishes.

The current purpose of this tab is to let you clear the **Auto Load Hyper-Report** check box so that the legacy HyperView report no longer opens for the measurement routine.

If you need information on the other options on this tab or on HyperView reports, see "Working with Legacy (HyperView) Reports" in the "Reporting Measurement Results" chapter.

For information on current reporting practices, see the "Reporting Measurement Results" chapter.

Setting Up the Probe Readouts Window



Probe Readouts Setup Dialog Box

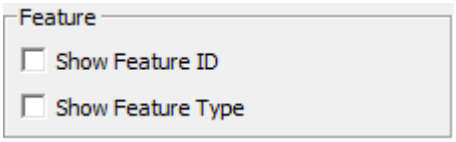
The **Probe Readouts Setup** dialog box (**Edit | Preferences | Probe Readouts Setup**) allows you to select the desired format of the Probe Readouts window. Select the check boxes that meet your probe readout needs. When you next access the Probe Readouts window, it will reflect the chosen format.

Example of a Probe Readouts window.

You can also right-click on the Probe Readouts window and click **Setup** to access the **Probe Readouts Setup** dialog box.

For information on how to use the Probe Readouts window, see "Using the Probe Readouts Window" in the "Using Other Windows, Editors, and Tools" chapter.

Feature Area



Feature Area

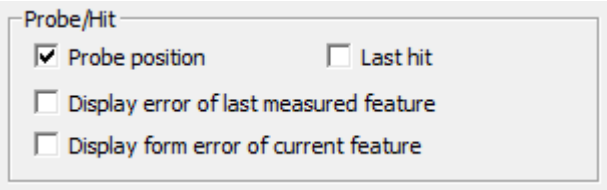
Show Feature ID check box
If in Learn Mode and there are CAD elements, this shows the text label of the nearest CAD element.

If in Execute Mode and PC-DMIS is executing sequentially through the measurement routine, this shows the ID of feature being executed. If PC-DMIS is executing in any order,

and the **Closest Feature** option is selected from the **Distance Target** area, it will show the ID of the closest feature.

Show Feature Type check box
If in Execute Mode and PC-DMIS is executing sequentially through the measurement routine, this displays the executing feature type. If PC-DMIS is executing in any order this displays the feature type of the closest feature.

Probe/Hit Area



Probe/Hit Area

Probe Position check box

When you select the **Probe Position** check box, PC-DMIS displays the current position of the probe. The Probe Readouts window displays the probe's position in the active coordinate system.

Last Hit check box

When you select the **Last Hit** check box, PC-DMIS displays the location of the last hit taken with the probe. If this option is not selected, PC-DMIS displays the current position of the probe.

Display error of last measured feature check box

When you select the **Display error of last measured feature** check box, PC-DMIS displays any deviations along the XYZ coordinates (and D for circular features) for the last measured feature in the Probe Readouts window. Even if the deviation is 0 then only 0 is displayed.

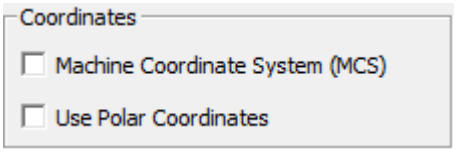
Display form error of current feature check box

When you select the **Display Form error of current feature check box**, PC-DMIS displays the form error for the feature being learned or executed.

If the feature has a valid form dimension (circularity, straightness, flatness, or cylindricity), then PC-DMIS displays the appropriate GD&T symbol near the value. Otherwise, PC-DMIS displays the Greek letter Sigma indicating standard deviation.

If there are not enough hits to calculate a form value, PC-DMIS displays a line of dashes for the value until sufficient hits are processed.

Coordinates Area



Coordinates Area

Machine Coordinate System (MCS) check box
When you select the **Machine Coordinate System (MCS)** check box, PC-DMIS displays the information in the machine's coordinate system, not the part coordinate system. This check box switches between these coordinate systems. Deselecting it will once again display the information in the part's coordinate system.

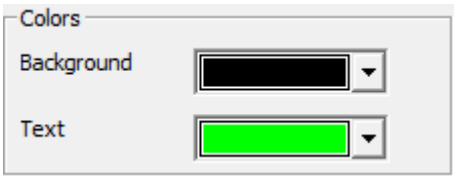
Use Polar Coordinates check box
When you select the **Use Polar Coordinates** check box, PC-DMIS switches between rectangular and polar coordinates. When polar coordinates are used, the normal direction of the workplane is also displayed.

Axis to Display Area

Axis to display Area

Check Box	Description
X	This hides or displays the X axis in the Probe Readouts window.
Y	This hides or displays the Y axis in the Probe Readouts window.
Z	This hides or displays the Z axis in the Probe Readouts window.

Colors Area

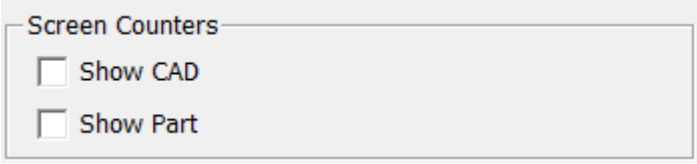


Colors Area

Background list - This sets the background color of the Probe Readouts window.

Text list - This sets the text color of the Probe Readouts window.

Screen Counters Area



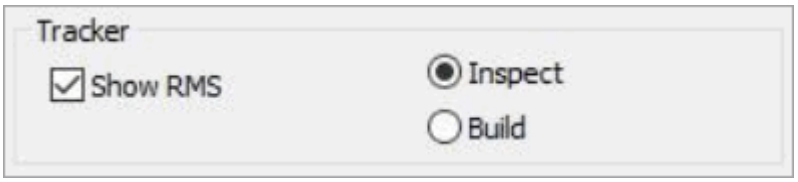
Screen Counters Area

Show CAD check box
This check box hides or displays the CAD X, Y, Z, I, J, K information for the CAD element at the mouse pointer's location in the Probe Readouts window. The origin is based off of the CAD model.

Show Part check box
This check box hides or displays the Part X, Y, Z, I, J, K information for the CAD element at the mouse pointer's location in the Probe Readouts window. The origin is based off of the current part alignment.

See "Displaying Mouse Coordinates in Screen Counter Text" in the "Editing the CAD Display" chapter.

Tracker Area



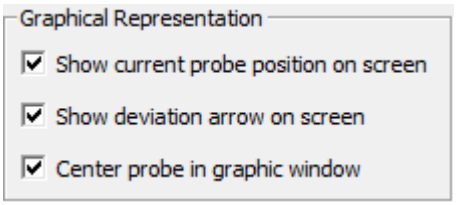
Tracker Area

Show RMS check box
This check box shows the **RMS** value in the Probe Readouts window if the measure machine is a portable tracker device.

Inspect or **Build** option
These options determine whether or not the information in the tracker is reported according to the **Inspect** mode or **Build** mode.

See the PC-DMIS Portable documentation for more information.

Graphical Representation Area



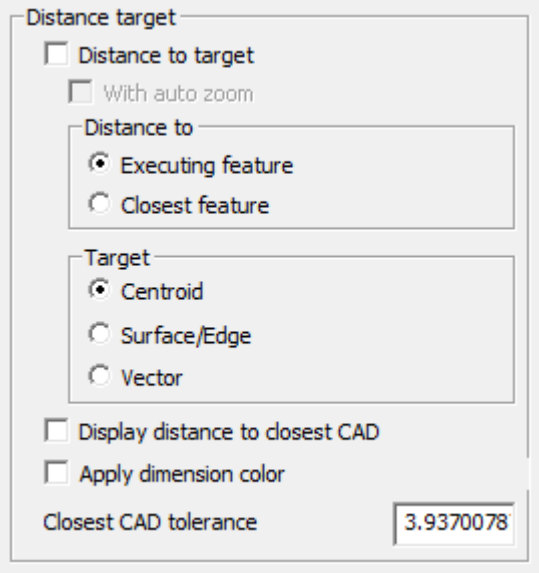
Graphical Representation Area

Show current probe position on screen - This check box shows a graphical representation of the probe on screen, relative to the machine when you move the probe's position using the jog box. This is helpful when you work in Online mode. When you use the jog box to move the probe, a graphical representation of the probe also moves on the PC-DMIS Graphic Display window. This only functions when you run PC-DMIS in Online mode connected to a physical machine and the measurement routine has an alignment.

Show deviation arrow on screen - This check box shows a 3D arrow in the Graphic Display window, which indicates the direction of deviation during execution. The **With Auto Zoom** check box in the **Distance Target** area must be marked.

Center probe in graphic window - This check box shifts the display in the Graphic Display window so that the animated probe always appears centered in the window whenever PC-DMIS executes motion commands or takes hits on the part. This is useful when you want to keep your view zoomed in close to your part model during execution yet visually follow the probe's progress. This functions only if PC-DMIS is not in Program Mode.

Distance Target Area

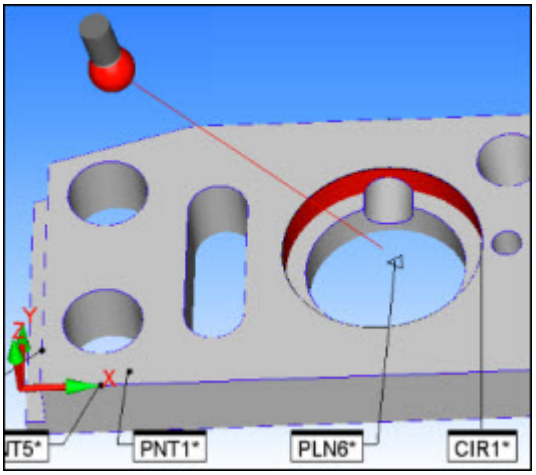


Distance Target Area

Distance to target check box
When you select the **Distance to target** check box, and the **Display distance to closest CAD** check box is not checked, PC-DMIS displays the distance of the probe to the target point depending on the **Distance to** setting. Otherwise, PC-DMIS always displays the distance to the closest CAD.

The probe's position is displayed in the active coordinate system. Manually drive the probe to the keyed in location. When the target point is reached, the Probe Readouts window will display 0,0,0.

A red line also appears on the Graphic Display window from the probe tip to the target, indicating the next feature to measure.



Example showing the red line pointing to the center of the PLN6 feature.

The target is based on the combination of options specified in the **Distance to** and **Target** areas of the dialog box. See a description of these options below and in the "Description of Target for T Value" topic.

With Auto Zoom check box
When you select both the **Distance to Target** check box and **With Auto Zoom** check box, PC-DMIS also displays the distance of the probe to the target point. The probe's position is displayed in the active coordinate system. As the probe is manually driven to the keyed in location, PC-DMIS makes the target point the center of the screen and zooms in on the point in the Graphic Display window.

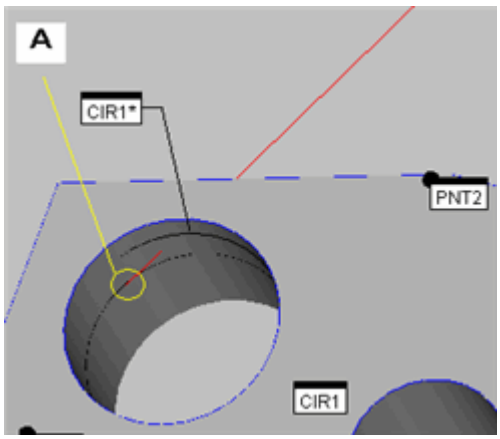
Distance area
This area contains these two option buttons to further define the target:

- **Executing feature** defines the target as the next executing feature.
- **Closest feature** defines the target as the feature closest to the probe.

You can determine the exact location on the target feature (either the centroid or the closest surface/edge point) by selecting the desired option from the **Target** area.

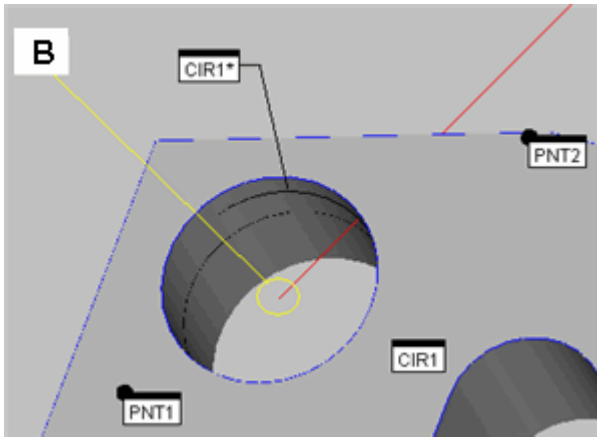
Target area
This area contains two option buttons to determine the exact target location to use:

- **Surface/Edge** defines the target as either the closest point on the feature or the next expected point based on what you selected from the **Distance** area:
  - If you selected **Closest feature**, the target is the closest point on the feature.
  - If you selected **Executing feature**, the target is the next expected point to take on the feature.



As shown in the above image, for example, while measuring the circle (CIR1), the **Surface/Edge** option causes the Probe Readouts window to show the distance to the actual target point at location A.

- **Centroid** calculates the distance to the feature's centroid.



As shown in the above image, for example, while measuring the circle (CIR1), the **Centroid** option causes the Probe Readouts window to show the distance to the feature's centroid at location B, instead of the actual target point.

- **Vector** calculates the distance from the probe's center to the nearest point on the feature's vector.

Display distance to closest CAD check box
Marking this check box has several effects. When marked, the target becomes the closest point on the nearest CAD surface. The red line connects the probe tip to this target.

Also, PC-DMIS displays a ‘T’ (CAD) value, or total deviation value, in the Probe Readouts window. See the "Description of Target for T Value" topic for information on exactly how the distance is used for each feature's T value.

This setting overrules the **Distance to Target** setting during execution.

Apply dimension color check box
This check box changes the colors of the deviation values (Distance to Target values) to match the out of tolerance dimension colors.

Closest CAD tolerance field
This is the field to enter the tolerance (in current units), which is used to determine the maximum distance the software tries to locate the surface to compare the current probe position to. Beyond this distance, no distance to CAD will be returned.

Description of Target for T Value

The T Value is always the magnitude (or length) of the vector from the probe tip to a target point. The target point is determined by the **Centroid**, **Surface/Edge**, or **Vector** option buttons located in the **Target** area of the **Probe Readout Setup** dialog box (**Edit | Preferences | Probe Readout Setup**).

- If you selected **Centroid** or **Vector**, PC-DMIS uses the distance from the probe's center point.
- If you selected **Surface Edge**, PC-DMIS uses the probe-compensated distance.

During Execution with "Closest Feature" and "Centroid" Selected and Closest Distance to CAD Not Selected

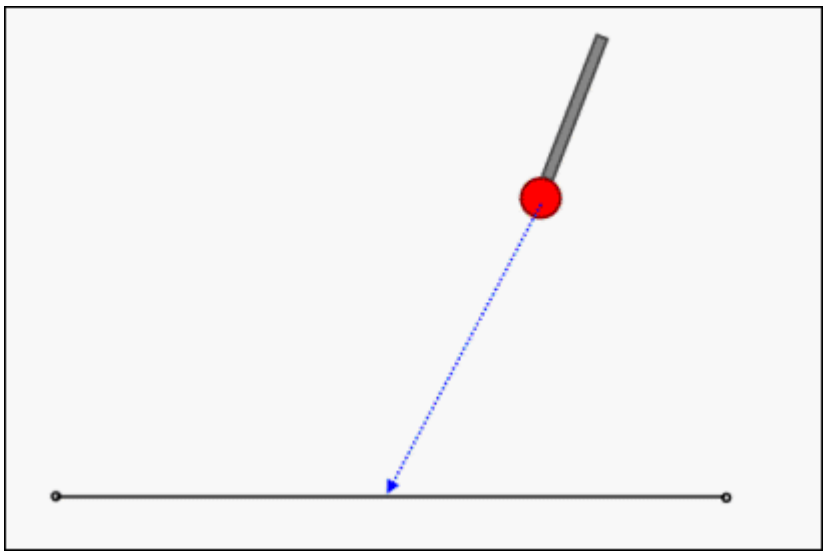
The Centroid is the feature's center. While this makes more sense for some features than others, all features have a center. For the nine basic feature types, the center for each is defined below with illustrations where necessary.

- Point

  The T value is the distance to the point itself.

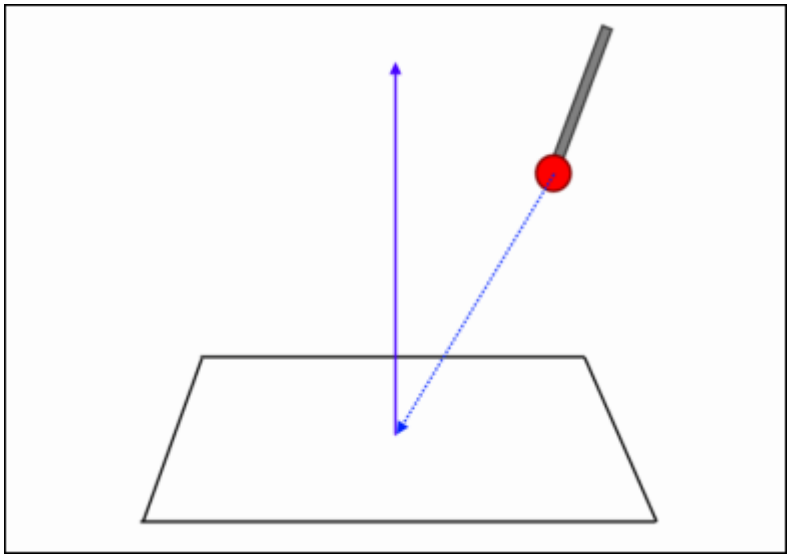
- Line

  The T value is the distance to the midpoint between the two outermost hits on the line segment.



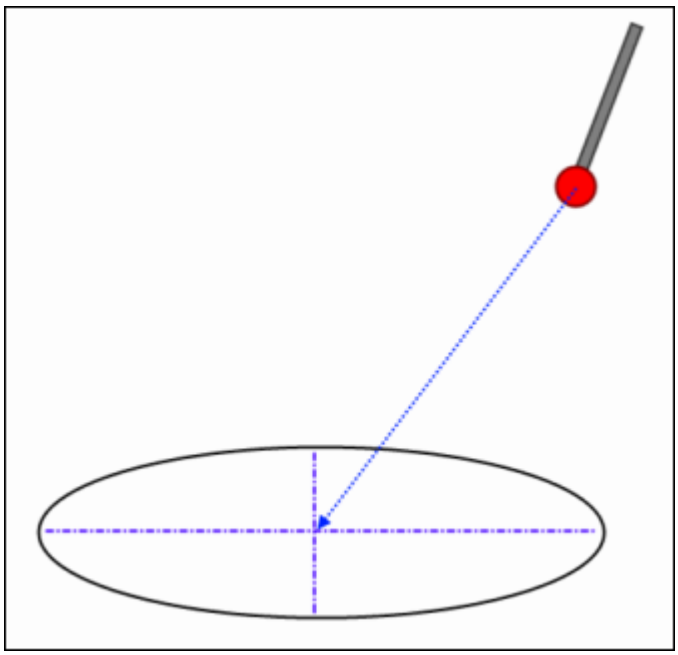
- Plane

  The T value is the distance to the center of the plane. The center is the midpoint of the polygon defined by the bounding hit points defining the plane.



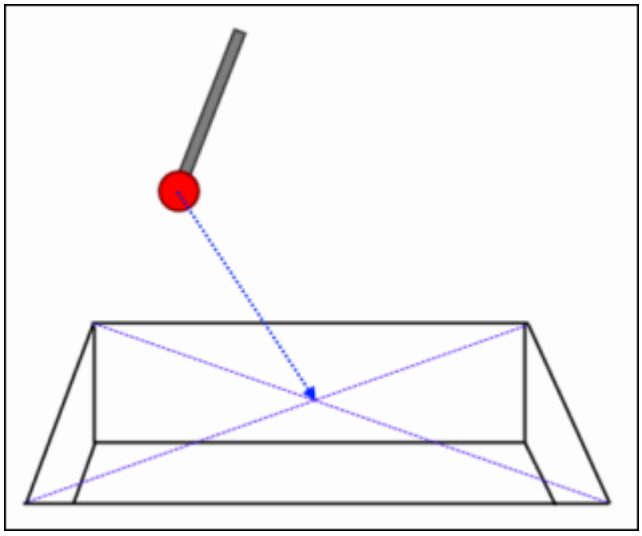
- Circle

  The T value is the distance to the circle's centroid.



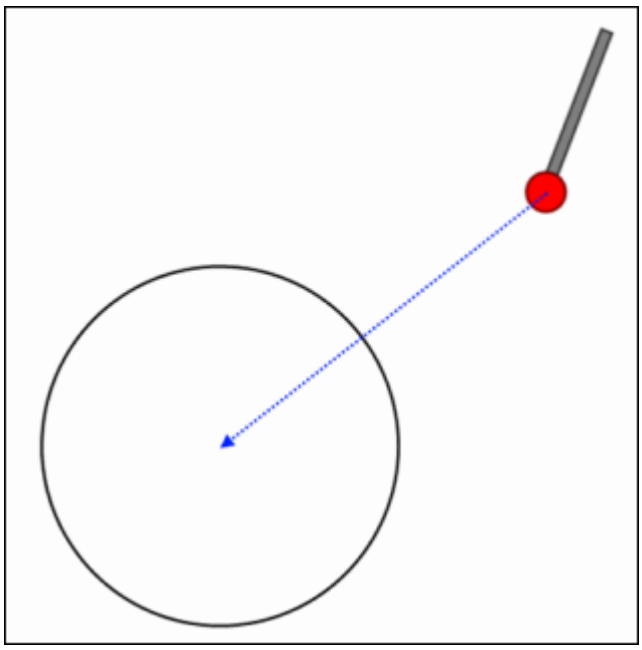
- Round and Square Slots

  The T value is the distance to the midpoint of the slot's plane.



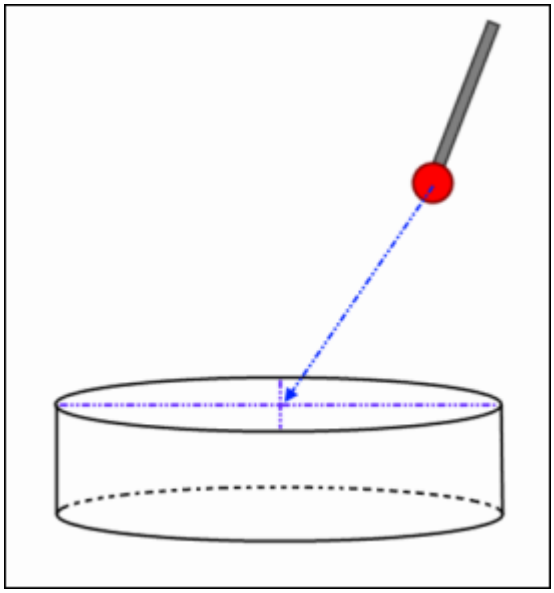
- Sphere

  The T value is the distance to the center of the sphere.



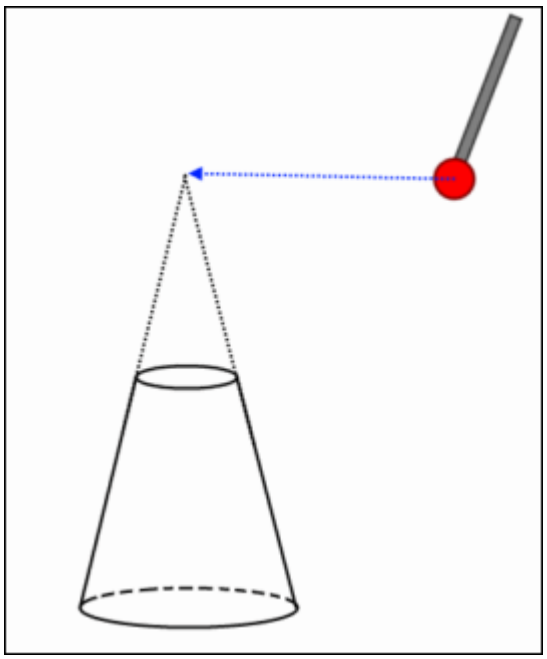
- Cylinder

  The T value is the distance to the center of the top of the cylinder:



- Cone

  The T value is the distance to the cone's vertex.



During Execution with "Closest Feature" and "Surface/Edge" Selected and Closest Distance to CAD Not Selected

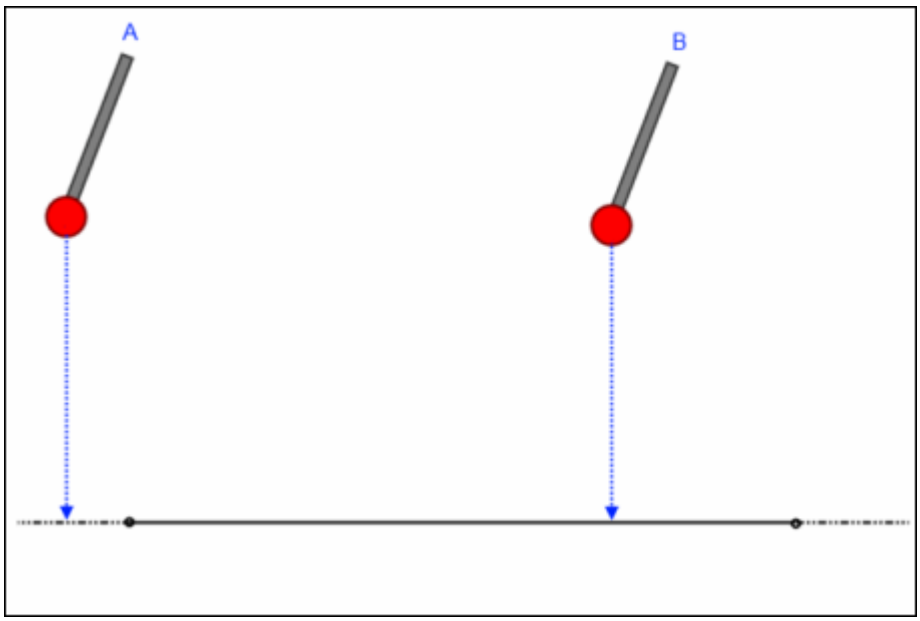
When locating the surface/edge, the target for the T value is the point on the feature that is closest to the probe. For 3D features, the closest point is on the surface of the feature; for 2D features, it is on the edge. For the nine basic feature types, the center for each is defined below with illustrations where necessary.

- Point

  The T value is the distance to the point itself.

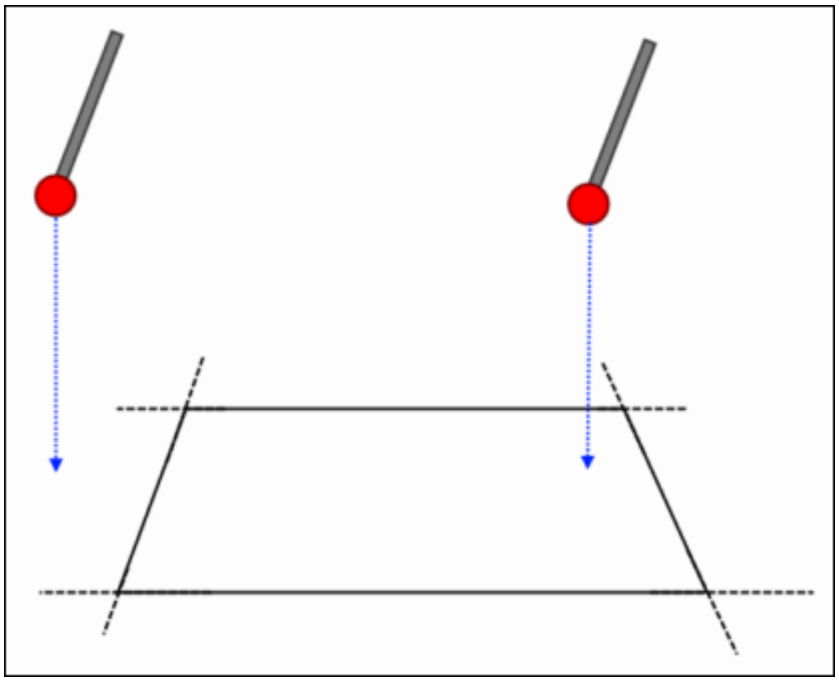
- Line

  The T value is the distance to the closest point on the line defined by two or more hits. This is not a line segment but a true geometric line. The example below shows two different cases.



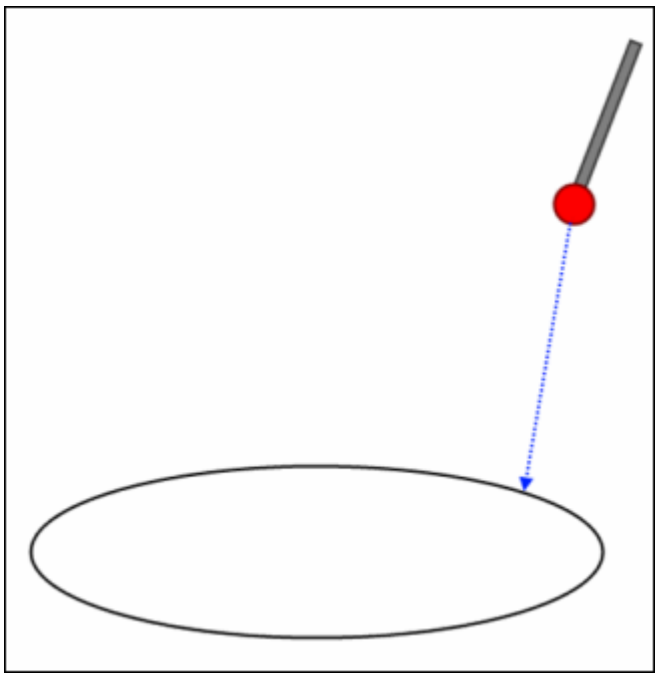
- Plane

  The T value is the distance to the closest point on the geometric plane defined by the three or more hits.



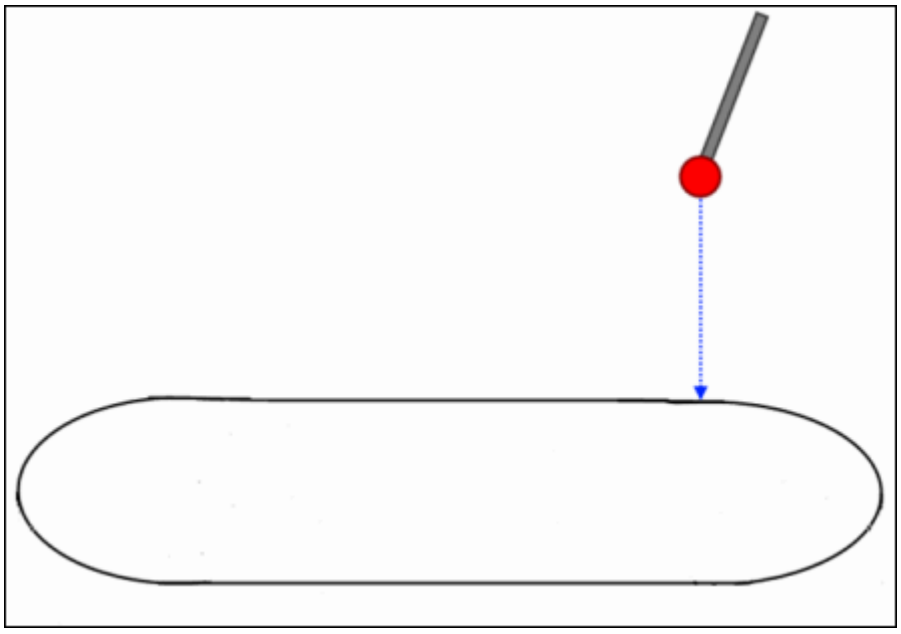
- Circle

  The T value is the distance to the closest point on the circle's edge.



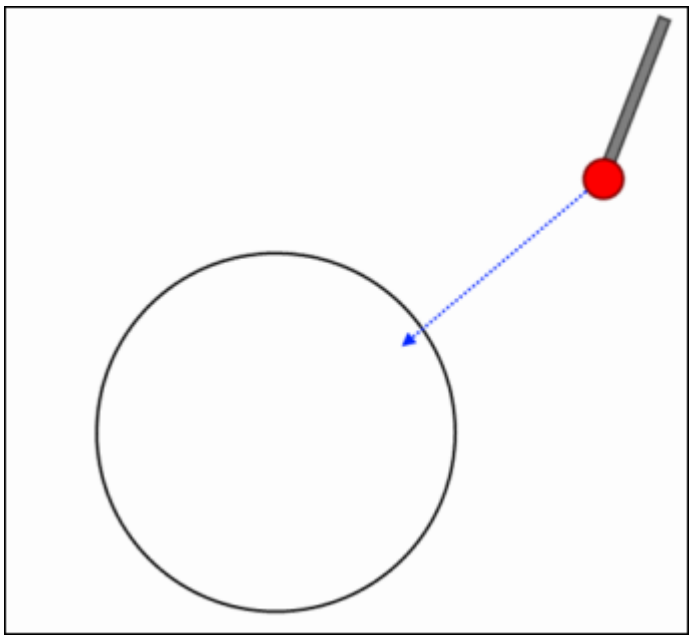
- Round and Square Slots

  The T value is the distance to the closest point on the slot's edge.



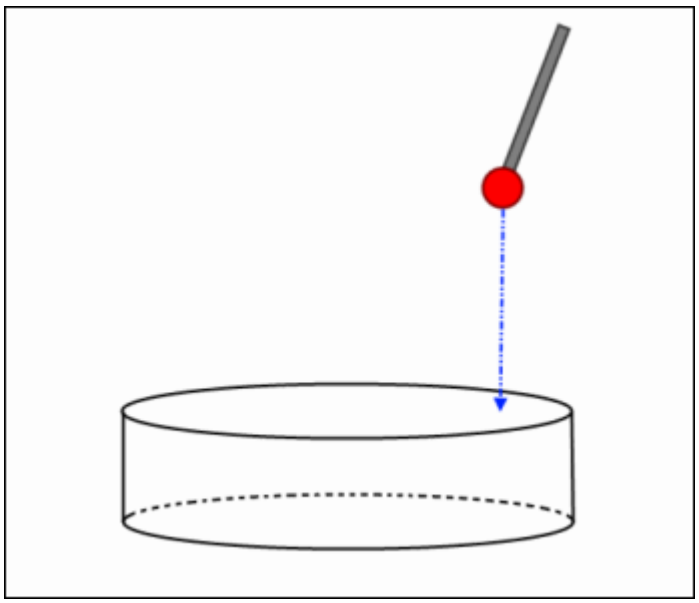
- Sphere

  The T value is the distance to the closest point on the sphere's surface. The illustration below shows the point on the surface of an outer feature, but an inner feature works the same and uses an inner surface.



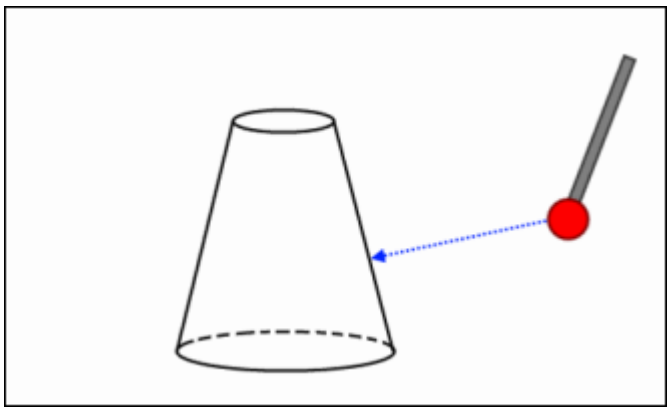
- Cylinder

  The T value is the distance to the closest point on the surface of the cylinder. The illustration below shows the point on the surface of an outer feature, but an inner feature works the same and uses an inner surface.



- Cone

  The T value is the distance to the closest point on the surface of the cone (not the geometric cone). The illustration below shows the point on the surface of an outer feature, but an inner feature works the same and uses an inner surface.



During Execution with "Closest Feature" and "Vector" Selected and Closest Distance to CAD Not Selected

When locating the vector, the target for the T value is the point on the feature vector closest to the probe. For the nine basic feature types, the center for each is defined below with illustrations where necessary.

- Point

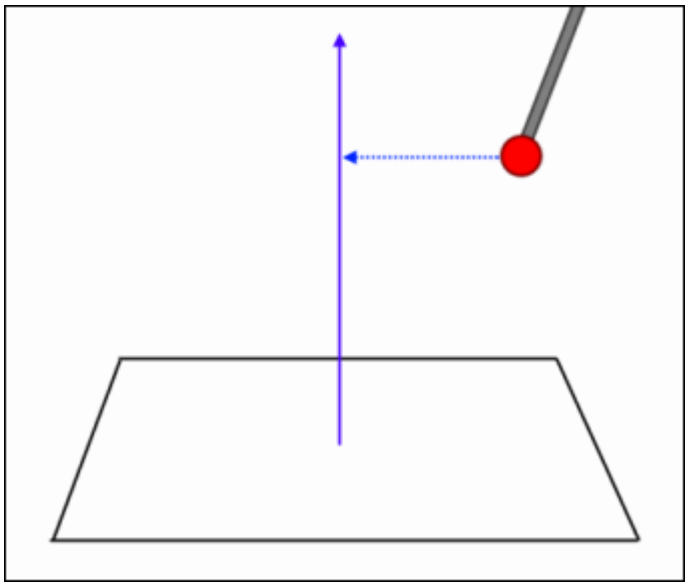
  The T value is the distance to the closest point on the point's hit vector.

- Line

  The T value is the distance to the closest point along the line's vector.

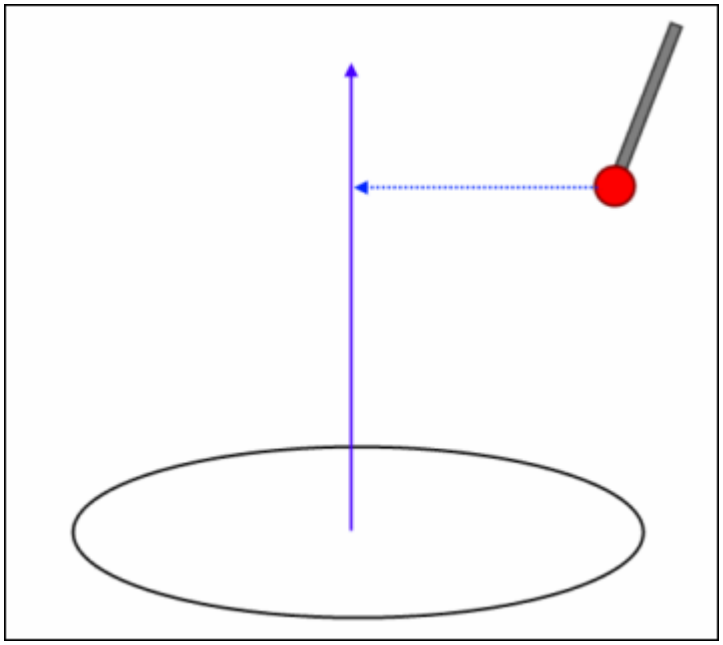
- Plane

  The T value is the distance to the closest point on the plane's normal vector that is anchored to the plane's center (see the Plane Centroid example above for a description of the plane's center).



- Circle

The T value is the distance to the closest point on the circle's normal vector that is anchored to the circle's center point.

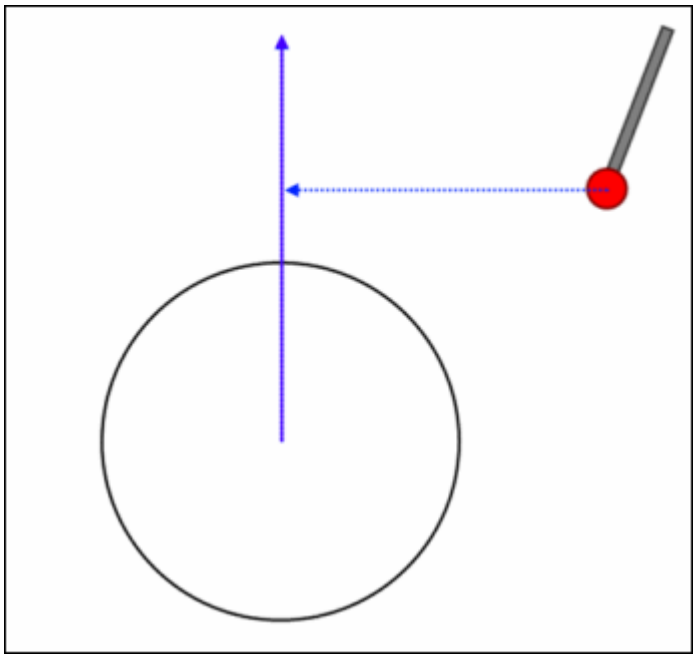


- Round and Square Slots

  The T value is the distance to the closest point on the normal vector of the slot's plane. The plane is defined by the slot's hit points. See the Plane Vector example above. As with that example, the normal vector is anchored to the center point of the slot's plane.

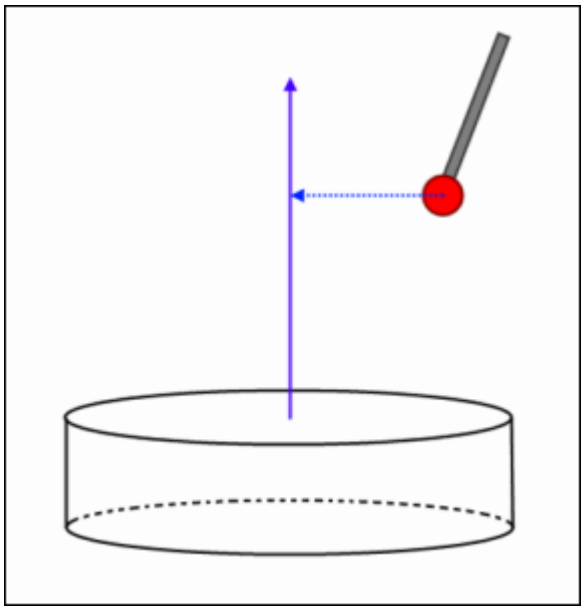
- Sphere

  The T value is the distance to the closest point on the sphere's normal vector. The vector is the same as the defined workplane or reference plane.



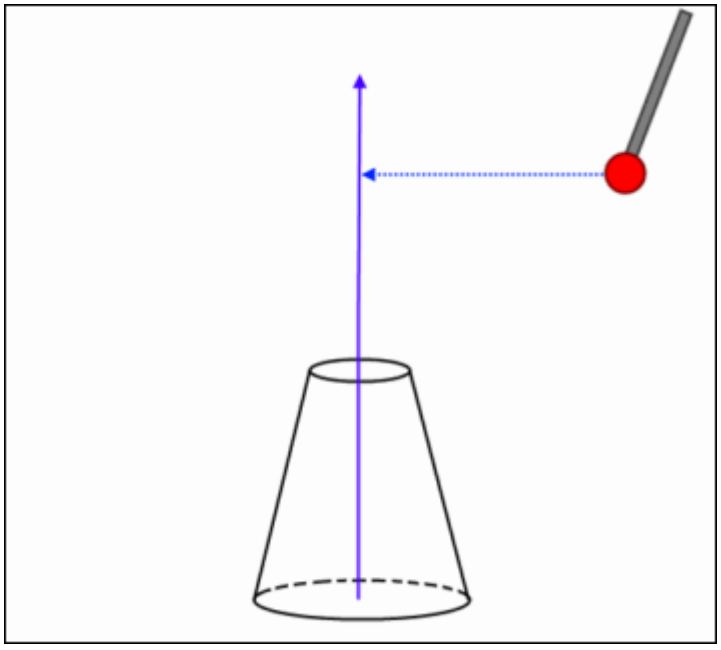
- Cylinder

  The T value is the distance to the closest point on the cylinder's central axis vector:



- Cone

  The T value is the distance to the closest point on the cone's central axis vector:



No Execution or Execution with "Closest CAD" Selected and a CAD File Loaded

The T value appears as T (CAD) and refers to the closest point on the CAD. The DX, DY, and DZ are the vector components of the T (CAD) value. There must be surface CAD data, and you must set your view to Surface Mode for this to work. If no CAD is loaded, the T,

DX, DY and DZ values refer to the **Distance to Target** setting, which is only active during execution.

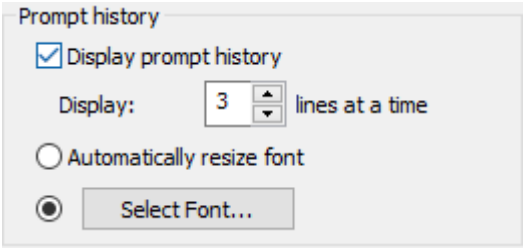
During Execution with "Executing Feature" and "Surface/Edge" Selected

This mode is for backwards compatibility and is the original functionality. In this state, the target point is the distance to the next hit on the feature.

During Execution with "Executing Feature" and "Centroid" or "Vector" Selected

If you selected **Centroid** or **Vector**, these options behave as the "Closest Feature" examples above, but they show distances to the next executing feature.

Prompt History Area



When you select the **Display prompt history** check box, PC-DMIS displays the new Probe Readouts comment type inside the Probe Readouts window.

- Using the **Display lines at a time** box, you can specify how many lines PC-DMIS should set apart for these comments on the Probe Readouts window.
- When the **Automatically resize font** button is marked, PC-DMIS automatically resizes the font size for the Operators Comments in Report Commands based on the value for the `ReadoutDisplayFont_AutoHistoryFontSizeRatio` entry in the PC-DMIS Settings Editor. For details, see "ReadoutDisplayFont_AutoHistoryFontSizeRatio" in the PC-DMIS Settings Editor documentation.
- Using the **Select Font** button, you can also specify the font type and size for these comments on the Probe Readouts window.

For information on inserting comments, see "Inserting Programmer Comments" in the "Inserting Report Commands" chapter.

For information on displaying the Probe Readouts window, see "Using the Probe Readouts window" in the "Using Other Windows, Editors, and Tools" chapter.

Always Track FOV Center

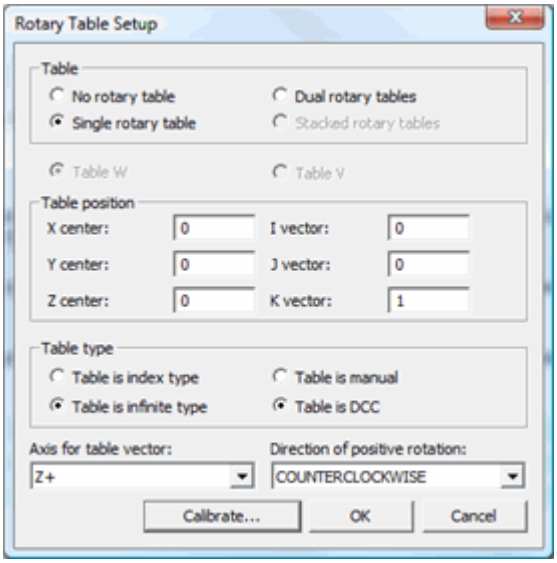
If this item is selected, PC-DMIS displays the FOV center of a non-contact video probe. This item only appears if you have a non-contact video probe defined.

Setting Up Multiple Arms

See the "Using Multiple Arm Mode" chapter for information regarding setup procedures for multiple arms.

Defining the Rotary Table

Select the **Edit | Preferences | Rotary Table Setup** menu option to open the **Rotary Table Setup** dialog box.



Rotary Table Setup Dialog Box

PC-DMIS disables rotary table motion when ClearanceCube motion is active.

This dialog box allows you to define the rotary table. You can only accept one option per category.

1. Select the type of table being used. If you select **Dual Rotary Table** or **Stacked Rotary Tables**, indicate the active table (Table W or Table V). If you select **Dual Rotary Tables** or **Stacked Rotary Tables**, PC-DMIS also displays the **Active Rotary Table** toolbar once you click the **OK** button. This toolbar contains two icons that allow you to select which rotary table is active.

2. Select either **Table is Index Type** or **Table is Infinite Type** option to define if the table is an indexable or infinite type.

> Index type rotary tables have a finite number of specific angles to which they can be positioned. Generally, they have a fixed increment (in degrees of rotation) from a given position to the next available position. The incremental amount varies based on your specific table. Please consult your rotary table documentation if necessary. Infinite type rotary tables accept any position (in degrees).

3. Select either the **Table is Manual** or **Table is DCC** option to define if the table is a manual or DCC table type.
4. From the **Axis for Table** list, select the machine axis closest to the axis of rotation for the table.
5. From the **Direction of Positive** list, select whether or not the table has a positive rotation about the chosen axis in either the CLOCKWISE direction or the COUNTERCLOCKWISE direction. The reference point of view for the direction is the perspective of looking down the axis for the table towards the origin.
6. If known, type the XYZ and IJK values. For PC-DMIS NC, the approximate location of the table center (within the probe's prehit distance) must be included because the entire calibration is done under DCC control.
7. Click **Calibrate** to begin the calibration process.

> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

> The **Rotary Table Setup** menu option is available only if your PC-DMIS license is configured to accept rotary tables.

ROTABSPEED Command

When you define a `Move/Rotab` command, the command uses the speed value set in the **Rotary Table** tab of the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**). You can override this value to slow down or speed up the rotary table with the `ROTABSPEED` command.

For example:

```
STARTUP=ALIGNMENT/START,RECALIBRATE:USE_PART_SETUP,LIST=YES

  ALIGNMENT/END

  MODE/DCC

  MOVESPEED/ 96

  FLY/ON

  FORMAT/
TEXT,OPTIONS,,HEADINGS,SYMBOLS,;NOM,TOL,MEAS,DEV,OUTTOL,,

  LOADPROBE/X5HD

  TIP/X5HD, SHANKIJK=0,0,1, ANGLE=90

  MOVE/ROTAB,30,SHORTEST,

  .

  .

  .

  ROTABSPEED/ 72

  .

  .

  .

  MOVE/ROTAB,60,SHORTEST,

  .

  .

  .

  ROTABSPEED/ 20

  .

  .

  .

  MOVE/ROTAB,77,SHORTEST,
```

```
.
.
.
```

In this example, the last five lines show the insertion of the `ROTABSPEED` command. The first instance speeds up the rotary table to 72 degrees per second, possibly to speed up the measurement process for a small part. The second `ROTABSPEED` command slows the rotary table down to 20 degrees per second, possibly for a larger part.

If you use the default value of 0 (zero) for the `ROTABSPEED` command, the rotary table's speed is proportional to the value set for the `MOVESPEED` command. This is necessary for backward compatibility prior to the implementation of the `ROTABSPEED` command.

For systems with stacked rotary tables, you need to account for both tables in the ROTABSPEED command, `ROTABSPEED <wspeed>[,<vspeed>]` where `<wspeed>` is table one and the optional `<vspeed>` is table two.

```
ROTABSPEED/ 50,40
```

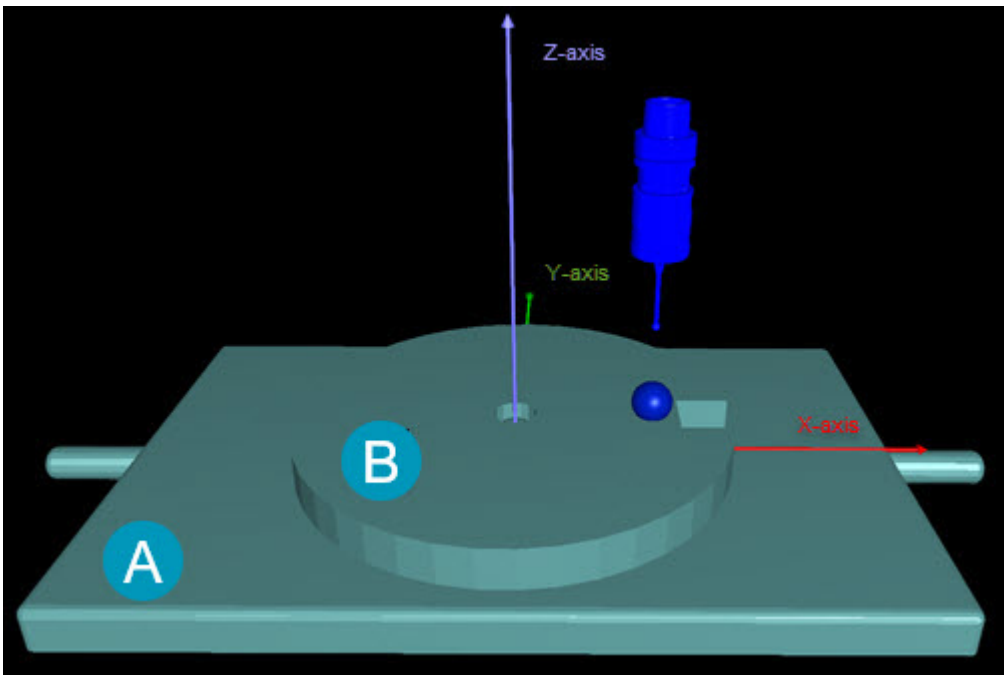
An example of a system with a stacked rotary table is the Optive CMM for vision systems.

Difference between a Stacked and Dual Rotary Table

Stacked Rotary Table

Stacked rotary tables provide the ability of rotating a part or calibration artifact beyond the standard XYZ axes.

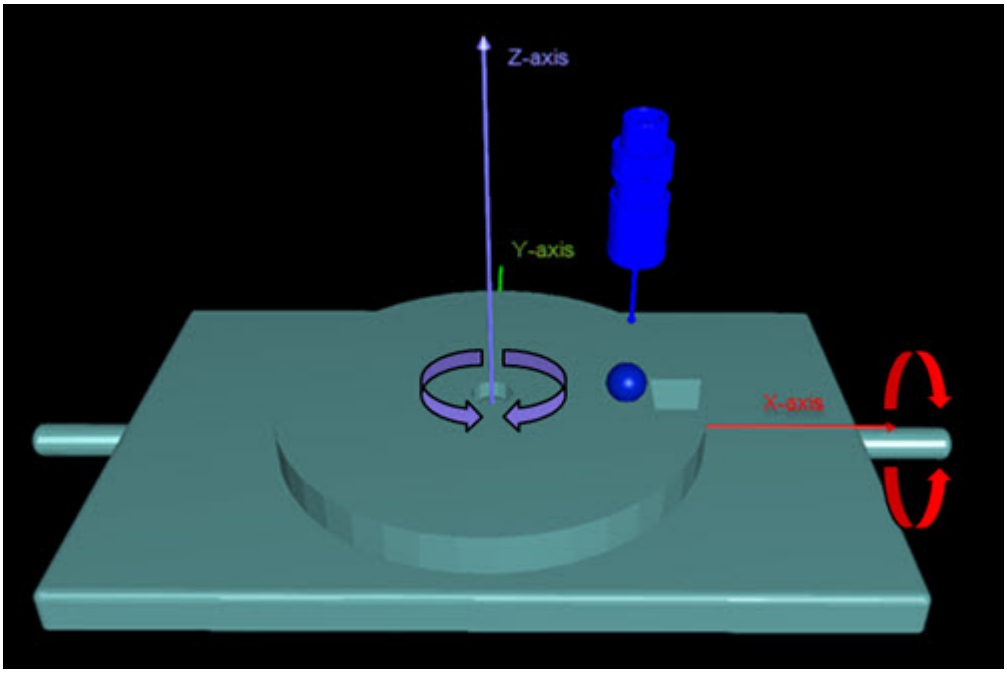
A stacked table system has two tables stacked on top of each other.



A. *Bottom table*

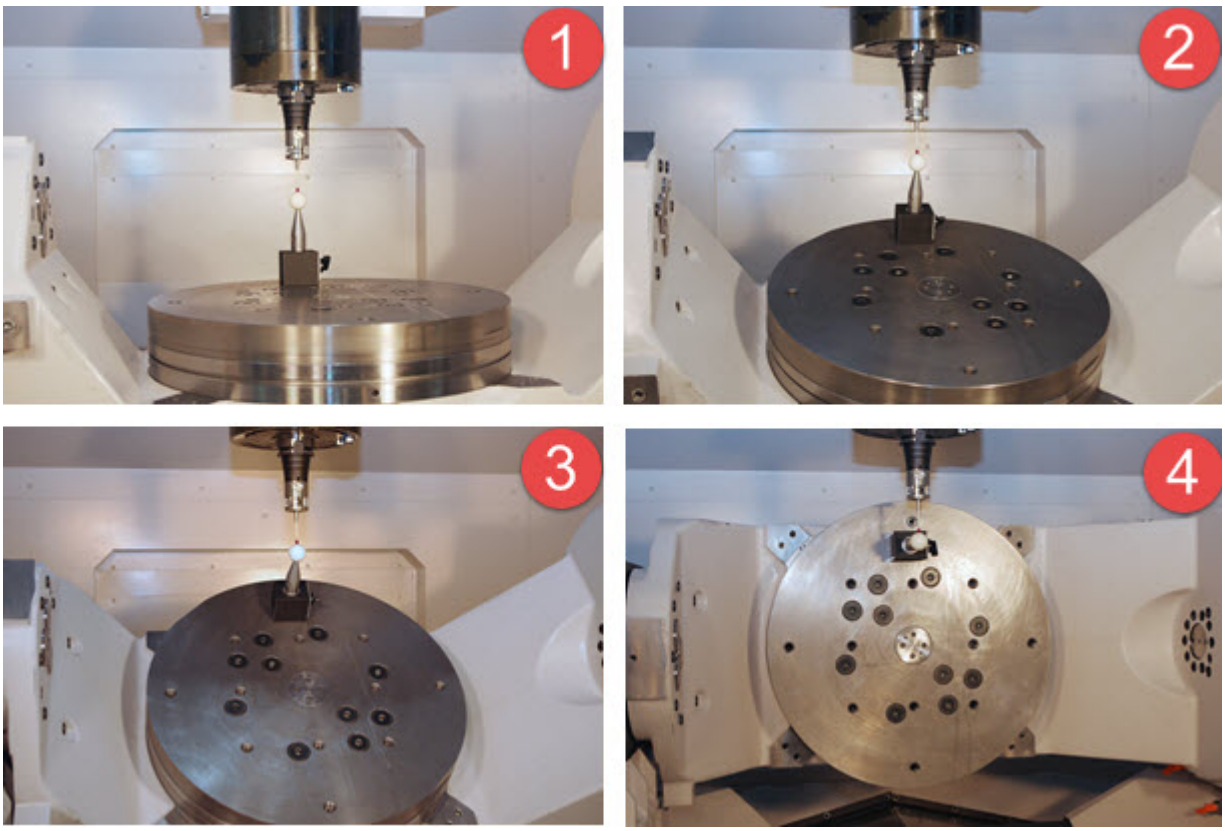
B. *Top table*

The top table rotates about the Z axis and the bottom table rotates about the X axis.



Example showing the table rotation directions.

The following four images show a real-world example of the A table on a stacked rotary table as it rotates 90 degrees.

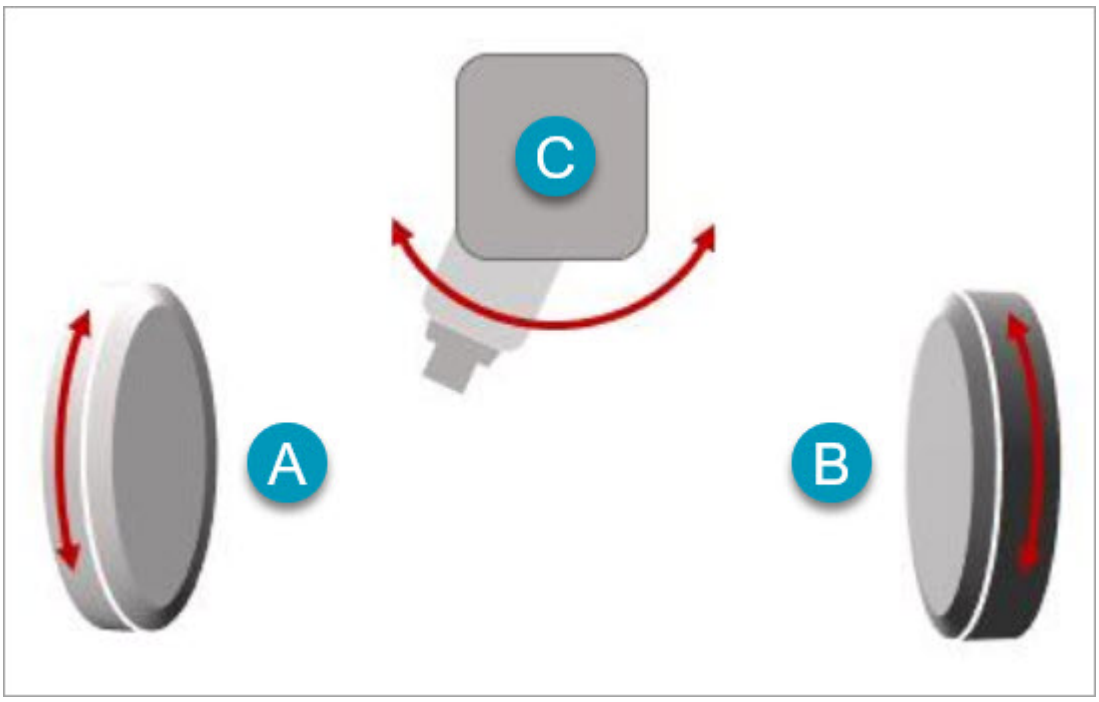


Example showing a stacked rotary table rotating 90 degrees.

Dual Rotary Table

The configuration of a Dual Rotary Table system has these properties:

- It is comprised of two independent rotary tables as typically seen on multitasking machines.
- Each table represents a lathe spindle. You define one spindle as the main spindle, and the other as the secondary spindle.
- The tables within the machine must be far enough apart from each other to provide the necessary operating space.



Example of a typical multitasking machine setup with two independent rotary tables (A and B) and a single-axis rotary head (C).

On a multitasking machine, both rotary tables (A and B) are lathe spindles and are powered separately. The single axis rotary head (C) is a positional milling spindle. The spindles turn at very high speeds and are used to cut (or mill) stationary parts. With two lathe spindles and a positional milling spindle, this is what makes a multitasking machine special. You can use it to completely manufacture all sides of a part, which allows you to have two parts in various stages of completion on the machine. You can cut on both lathe spindles independently and at the same time. In most cases you can also transfer a part clamped in one lathe spindle to the other automatically. For example, you can drive the B table to the A table until the jaws on the B table can grab the part from the A table.

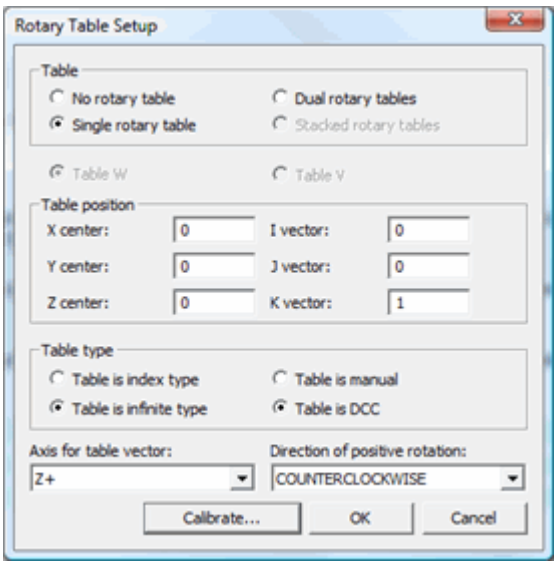
> **!** If you want to control two independent rotary tables, you should use two separate measurement routines, one for each table.

On non-multitasking machines, you can only have one table active at a time. You can select the active table from the Active Rotary Table toolbar. Once you activate a table, it remains active for the entire length of the measurement program. The other table stays inactive.

You can calibrate and use the active table just as you would a single table system.

Calibrate Rotary Table

The **Edit | Preferences | Rotary Table Setup** menu item accesses the **Rotary Table Setup** dialog box.

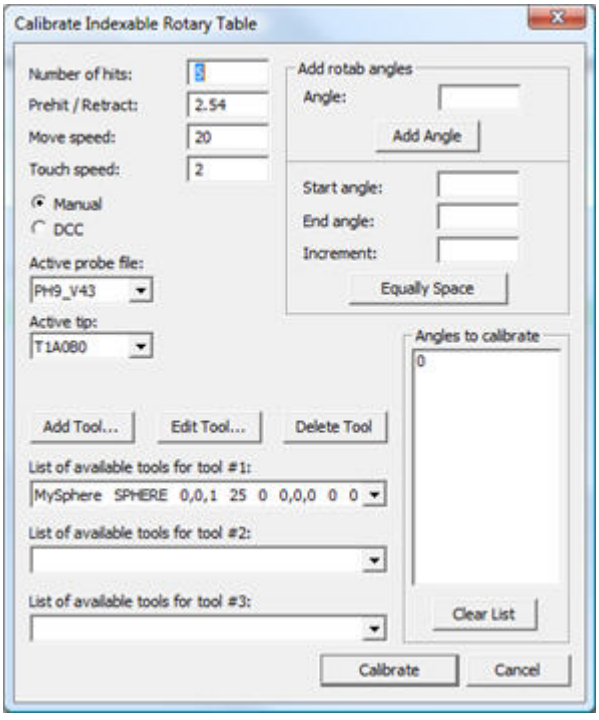


Rotary Table Setup Dialog Box

Depending on the options selected **Table Type** area of the **Rotary Table Setup** dialog box (see "Defining the Rotary Table"), one of two dialog boxes opens when you click the **Calibrate** button.

- Selecting the **Table is Infinite Type** option and clicking the **Calibrate** button opens the **Calibrate Infinite Rotary Table** dialog box.
- Selecting the **Table is Index Type** option and clicking the **Calibrate** button opens the **Calibrate Indexable Rotary Table** dialog box.

Calibrate Indexable Rotary Table



Calibrate Indexable Rotary Table Dialog Box

Using the **Calibrate Indexable Rotary Table** dialog box, you can select the options PC-DMIS uses to calibrate the selected table. Once the options within the dialog box are satisfactory, click the **Calibrate** button to start the rotary table calibration process.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Usage Requirements: The indexable rotary table calibration must include the 0 position. Additionally, all the angles used in a measurement routine must also be calibrated. This calibration procedure calculates and stores a transformation for all other angles relative to the 0 position.

Calibration Requirements: The indexable rotary table calibration procedure requires a valid XYZ, IJK for the table in order to run properly. This can be accomplished in two ways:

1) If the values are known, they can be manually typed into the **Rotary Table Setup** dialog box. This is usually *not* the case.

2) Initially select the **Table is Infinite Type** option and complete an infinite rotary table calibration, which calculates and saves the XYZ, IJK. Then select the **Table is Indexable Type** and perform the indexable rotary table calibration. This is normally only an issue during the initial software installation/setup or if the rotary table has been moved or if something has happened to significantly alter the origin of the machine coordinate system. Once the XYZ, IJK is determined closely enough that the indexable rotary table calibration procedure can run successfully it is not necessary to go back and re-run the infinite rotary table calibration in order to re-run the indexable rotary table calibration.

Add Rotab Angles

The **Add Rotab Angles** area allows you to define the list of table angles that are included in the calibration. You can define one angle at a time, or you can define an incremented range of angles. Defined angles are then placed in the **Angles to Calibrate** list. When you click **Calibrate**, PC-DMIS then calibrate the rotary table using the defined angles.

If you want to calibrate all the angles between 5 and 95 degrees with a 10 degree increment between each angle, you would fill in the **Start Angle**, **End Angle**, and **Increment** boxes with 5, 95, and 10 respectively, and then click the **Equally Space** button.

Angles to Calibrate List

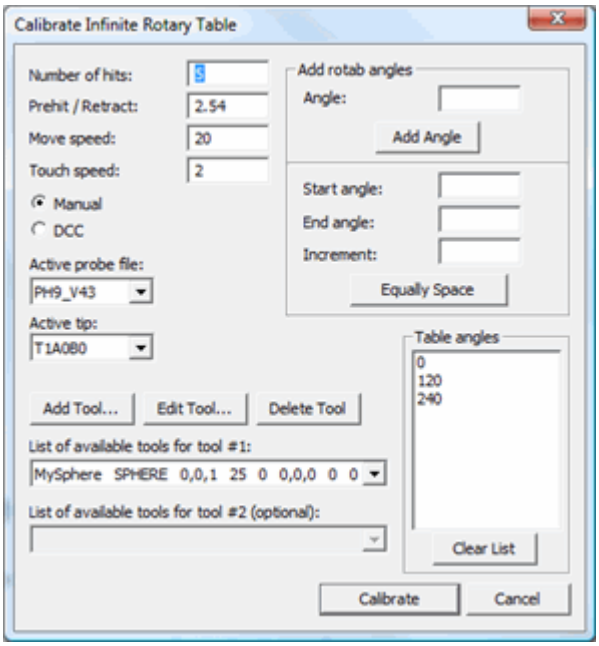
This list contains all the table angles for the calibration. You can add angles to this list from the **Add Rotab Angles** area. The **Angles to Calibrate** list must use the zero angle.

Other Dialog Box Parameters

This dialog box also contains many of the same options described in the "Defining Hardware" chapter.

- For information on **Number of Hits**, **Prehit / Retract**, **Move Speed**, **Touch Speed**, **Manual / DCC**, **List of Available Tools**, **Add Tool**, and **Delete Tool**, see the "Measure" topic in the "Defining Hardware" chapter.
- For information on **Active Probe File** and **Active Tip**, see the "Defining Probes" topic in the "Defining Hardware" chapter.

Calibrate Infinite Rotary Table



Calibrate Infinite Rotary Table Dialog Box

For information on many of the options on this dialog box, see the "Calibrate Indexable Rotary Table" topic in the PC-DMIS Core documentation.

The **Calibrate Infinite Rotary Table** dialog box differs from the **Calibrate Indexable Rotary Table** dialog box in these areas:

- Instead of the **Angles to Calibrate** list, the dialog displays the **Table Angles** list.
- You don’t have to use the 0.0 angle in the **Table Angles** list.
- Instead of three calibration tools, the dialog box only displays one calibration tool.

Once the options within the dialog box are satisfactory, click the **Calibrate** button to start the rotary table calibration process.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Setting Up Probe Changer Options

Use the **Probe Changer** dialog box to set up various options that you can use on a probe changer. To display the dialog box, select **Edit | Preferences | Probe Changer**.

For complete information, see "Defining Probe Changers" in the "Defining Hardware" chapter.

Managing Multiple Probe Changers

The topics below discuss the more popular probing systems used today (the TP2, ACR1, TP20 and TP200, and the SP600). The "Configuring Multiple Racks" and "SP25 Probe/Stylus Change system" topics then provide detailed examples of how to work with multiple probe changers.

Background on TP2

When Renishaw developed the small touch trigger probe (TP2), it attached to the CMM ram via a probe head with a threaded M8 connection in line with the probe body itself. However, this design required a probe recalibration whenever it was removed or reattached.

To alleviate this need to recalibrate, Renishaw developed a quick-connect adapter (termed QuickConnect) which used a ¼ turn of a key to lock and unlock the adapter from the CMM. The TP2 screwed into this adapter. This adapter could be detached and re-attached very quickly and repeatedly without the necessity of always recalibrating.

Background on ACR1

The ACR1 was the first probe changer introduced by Renishaw. It held up to eight of the quick-connect extensions, each with a separate TP2 probe body. Once the probes were qualified and placed into the rack, they could be automatically dropped off and picked up by the CMM with simple move commands and coordinated with the locking/unlocking mechanisms of the rack. A software module was eventually developed which controlled the rack operations.

Background on TP20 and TP200

Over time, design evolution and electronic developments paved the way for alternatives to the TP2 probe body. In addition, for some customers, purchasing up to eight probe bodies proved prohibitive. New probe body designs were developed, which allowed for removal and attachment of new stylus adapters. These took the place of the quick-connect joints. Individual stylus assemblies could then be removed and re-attached repeatedly and inexpensively.

Two of the better-known probe bodies of this design from Renishaw include the TP20 and the TP200. The TP20 uses a magnetic coupling, which allows the exchange of multiple probe modules. You can store the individual modules in the TP20 probe rack, which is actually called the MCR20 (where "MCR" is Module Change Rack).

Each probe body is roughly equivalent to the TP2 in size and shape, but there are two main differences:

- Due to their enhanced electronics, they can hold more weight and still produce more accurate and repeatable results.
- They are designed with a magnetic "split" between the upper probe body and the lower stylus holder. This allows them to be used with their own stylus change rack systems: the MCR20 and the SCR200, respectively.

Background on the SP600 Analog Probe

Another improvement is the popular SP600 analog probe. This probe can perform analog scans as well as touch trigger measurements. While its body design is much larger than the TP series of probes, a magnetic stylus holder can separate from the body. Accordingly, it also has its own rack system, the SCR600 tip changer.

Configuring Multiple Racks

While there are other probing systems on the market, the four most popular probing systems, as discussed, are the TP2, TP20, TP200, and SP600 systems. Each has its own rack system which can function on its own as a single rack. Alternately, you can use more than one rack with a CMM. The PC-DMIS software has the ability to move between the racks to drop off and pick up probes and stylus holders as needed.

Important Notes:

- Each rack has its own calibration method in PC-DMIS. While they share a lot of commonality, for calibration purposes, they are completely independent of each other.

- The clearance move point for each rack is only related to that rack, except that some consideration must be given to where to will travel next. For this reason, the clearance move should be at such a location to allow free travel to any other rack which may be involved. Even when using a single rack, this clearance move must be sufficient to travel from and return to the part inspection operations.
- Defining the ports' contents for multiple racks is the most critical part of configuring multiple racks for simultaneous use. Since each port may contain multiple probe references (the contents of the port can be used with multiple probes) each port must identify all of the probes it will potentially use.

Example of Multiple Probe References

Suppose you have these three probe configurations:

PROBE_01	PROBE_02	PROBE_03
QuickConnect AutoJoint	QuickConnect AutoJoint	QuickConnect AutoJoint
TP2	TP20	TP20
3 mm x 10 mm stylus	2 mm x 10 mm stylus	4 mm x 20 mm stylus

The ACR1 will switch between the TP2 and the TP20 probes. The MCR20 will switch between the styli associated with the TP20 probe system(s).

A typical port definition might look like this:

ACR1	MCR20
Port 1	Port 2
Probe_01	Probe_02
	Probe_03

In operation, suppose the system is using PROBE_01 and it needs to switch to PROBE_02. The system would:

- Pause the measurement routine.
- Travel to the clearance location for the ACR1.
- Return the currently loaded probe to port #1 of the ACR1.
- Move to port #2 of the ACR1 and pick up the TP20 body for PROBE_02.
- Use the respective Safety/Clearance points and move to the MCR20.
- Move into port #1 of the MCR20 to attach the stylus adapter with the desired stylus.
- Return to the MCR20’s Clearance point.
- Continue with the measurement routine.

Suppose after measuring some features, the system then needs to use PROBE_03. The system would:

- Pause the measurement process.
- Travel to the clearance location for the MCR20.
- Move into port #1 to drop off PROBE_02’s stylus.
- Travel out, over, and into port #3 to pick up the stylus required for PROBE_03.
- Travel to the clearance location.
- Continue with the measurement routine.

Now, suppose the system needs to switch from PROBE_03 (the TP20 with the 4 mm x 20 mm stylus) back to Probe_01 (the TP2 with the 3 mm x 10 mm stylus). The system would:

- Pause the measurement routine.
- Move to the clearance point for the MCR20.
- Go into port #2 and drop off the stylus assembly.
- Return to the MCR20’s clearance location
- Move to the ACR1’s clearance location.
- Go into port #2 to drop off the TP20 assembly.
- Move out and return to port #1 to pick up the TP2 assembly (the TP2 probe already has the stylus attached).
- Move to the clearance point for the ACR1.
- Continue the measurement routine.

Note that in this example, you only need one TP20 probe body. You use the MCR20 to switch between two different probe stylus assemblies for varied measurement requirements.

SP25 Probe/Stylus Change system

The SP25 rack is an extension of the same procedures used in other racks. This discussion covers the ARC1 and TP20 racks and then branches out to describe the SP25 rack system.

Understanding the ACR1 Rack

When using the ACR1, the system uses the Quickened joint between the probe head and the probe body. All of the ports in the ACR1 are identical, so if you had an SP600 in port 1, it would connect directly to the probe head. If you wanted to put a TP2 into port 2, you would need to mount an adapter to the TP2 (as with TP20 and TP200 probing systems).

With this configuration in mind, you only need to define one probe name for each port. Technically, the ACR1 is a probe changer, and each probe would have a stylus already attached to it when it sits in the rack.

Suppose you want to assign these probes to the ACR1 rack: "SP600", "TP2", "TP20", and "TP200". The port definitions in the ACR1 rack would be:

PORT 1	PORT 2	PORT 3	PORT
SP600	TP2	TP20	TP200

Understanding the ACR1 with a TP20 Stylus Changer

Now, if you want to combine the ACR1 with a TP20 rack (a stylus changer), things get slightly more complicated. The ACR1 will switch between the SP600 and the TP20 probe bodies. Once the system picks up the TP20, it will go to the MCR20 rack to pick up an appropriate stylus for it.

Suppose you have 3 separate styli that you wanted to use with the TP20 and that the SP600 is used as a single probe (no stylus changing).

When you define the ports, you might call the SP600 probe "SP600_1", and the TP20 probes with the varied styli attached, you might call "TP20_1", "TP20_2", and "TP20_3".

The ports in the ACR1 probe changer would be defined as:

PORT 1	PORT 2	PORT 3	PORT 4
SP600	TP2	TP20_1	TP200
		TP20_2	
		TP20_3	

The ports in the TP20 stylus changer would be defined as:

PORT 1	PORT	PORT 3	PORT 4
TP20_1	TP20_2	TP20_3	empty

Understanding the SP25 Rack System

You can extend the logic from the above topics to the SP25 rack system. This rack can hold two different types of probe components. For one type the port is calibrated empty, and for the other type a port is calibrated with an insert.

This discussion will abandon the previous references to SP600, TP2, TP20, and TP200 since the SP25 rack system uses these components instead:

- SP25M
- SM25-x
- SH25-x

The SP25M system can adapt to any of five different SM25 modules for varied styli lengths and weights, and a sixth module for use with the TP20. They all use the same SP25M probe body.

- SM25-1 module - This module accepts only the SH25-1 stylus holder. It is used for styli lengths between 20 mm and 50 mm.
- SM25-2 module - This module accepts only the SH25-2 stylus holder. It is used for styli lengths between 50 mm and 105 mm.
- SM25-3 module - This module accepts only the SH25-3 stylus holder. It is used for styli lengths between 120 mm and 200 mm.

- SM25-4 module - This module accepts only the SH25-4 stylus holder. It is used for styli lengths between 220 mm and 400 mm.
- SM25-5 module - This module accepts only the SH25-5 stylus holder. It supports straight styli lengths of approximately the same range as the SM25-2. However, it is specially designed for improved handling of perpendicularly attached styli (that is, crank styli configurations). The maximum length of a crank stylus depends on how far down from the stylus holder it is attached. The maximum crank stylus length forms an approximately conical shape varying from 105 mm at 15 mm down to 20 mm at 80 mm down.
- TM25-20 module - This module can only accept TP20 touch trigger probes and therefore does not support scanning.

Any of the SM25-x modules above are capable of individual touch probing as well as scanning.

Example of SP25 with Multiple Probes

Suppose your SP25 rack has these six probe assemblies that are always attached to the probe head:

P1	P2	P3	P4	P5	P6
SP25M	SP25M	SP25M	SP25M	SP25M	SP25M
SM25-1	SM25-1	SM25-2	SM25-3	TM25-20	TM25-20
SH25-1	SH25-1	SH25-2	SH25-3	TP20	TP20
2 mm x 20 mm stylus	4 mm x 30 mm stylus	6 mm x 80 mm stylus	8 mm x 100 mm stylus	2 mm x 20 mm stylus	4 mm x 20 mm stylus

You would define the ports in the FCR25 rack to hold components making up the various probe configurations. This first table shows the ports that hold components for the above probe configurations.

PORT 1	PORT 2	PORT 3	PORT4	PORT5	PORT6
P1	P1	P2	P3	P4	P5*
P2					

In detail, the ports would hold these components:

PORT 1	PORT 2	PORT 3	PORT4	PORT5	PORT6
SM25-1	SH25-1	SH25-1	SH25-2	SM35-3	TM25-20
	2 mm x 20 mm stylus	4 mm x 20 mm stylus	6 mm x 80 mm stylus	SH25-3	TP20*
				8 mm x 100 mm stylus	2 mm x 20 mm stylus*

- Slot 1 contains the SM25-1 module only. It uses no insert for this. This component attaches directly to the SP25 and requires the addition of the SH25-1 with the 2 mm stylus found in Slot 2 or the SH25-1 with the 4 mm stylus found in port 3.
- Slot 2 contains the SH25-1 stylus holder with an attached 2 mm x 20 mm stylus. This component requires an insert in the port to adapt the physical characteristics of the port for this component. This component requires an SM25-1 module (found in port 1). Once it picks up the SM25-1, the probe assembly is complete.
- Slot 3 contains the SH25-1 stylus holder with an attached 4 mm x 30 mm stylus. This component requires an insert in the port to adapt the physical characteristics of the port for this component. This component requires an SM25-1 module (found in port 1). Once it picks up the SM25-1, the probe assembly is complete.
- Slot 4 contains the SH25-2 stylus holder and an attached 6 mm x 80 mm stylus. It uses no insert for this. Once this is picked up, the probe assembly is complete.

- Slot 5 contains the SM25-3 with the SH25-3 stylus holder and an attached 8 mm x 100 mm stylus. It uses no insert for this. Once this is picked up, the probe assembly is complete.
- Slot 6 contains the TM25-20 module only. It uses no insert for this. This component attaches directly to the SP25M.

  * When used with a single stylus, the TP20 probe module and stylus can be attached to the TM25-20 module while in the port, and requires no additional rack system. However, when used with multiple styli (as in this example), the TM25-20 module has no other components attached while it sits in the port, but requires the use of additional FCR25 ports with adapters specifically for the purpose of holding the TP20 module/stylus combinations. For this example, the added three ports would look like this:

PORT 7	PORT 8	PORT 9
P5	P6	empty

PORT 7	PORT 8	PORT 9
TP20	TP20	empty
2 mm x 20 mm stylus	4 mm x 20 mm stylus	

- **When using P1**, the CMM would drop off any existing components. Then it would pick up the SM25-1 found in port 1 of the FCR25 and proceed to pick up the SH25-1 found in port 2 of the same rack.
- **When using P2**, the CMM would drop off any existing components. Then it would pick up the SM25-1 found in port 1 of the FCR25 and proceed to pick up the SH25-1 found in port 3 of the same rack.
- **When using P3**, the CMM would drop off any existing components. Then it would pick up the SM25-2 and SH25-2 combination found in port 4. With this, the probe assembly is complete.
- **When using P4**, the CMM would drop off any existing components. Then it would pick up the SM25-3 and SH25-3 combination found in port 5. With this, the probe assembly is complete.

- **When using P5**, the CMM would drop off any existing components. It would then pick up the TM25-20 body in port 6 of the FCR25 and proceed to pick up the TP20 module/stylus combination in port 7 of the FCR25.
- **When using P6**, the CMM would drop off any existing components. It would then pick up the TM25-20 body in port 6 of the FCR25 and proceed to pick up the TP20 module/stylus combination in port 8 of the FCR25.

As with other tool changers and probe assemblies, components are dropped off in reverse order from how they are picked up.

Loading the Active Probe

The **Operation | Load Active Probe** menu option loads the active probe required by the measurement routine. While in Learn mode, use the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**) to change the loaded probe file or files.

Use the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**) to define the appropriate probe configurations for each port that is used. You can then use the **Load Active Probe** option to tell the machine to exchange the necessary probe configuration or configurations.

Examples of Working with Single and Multiple Probe Changers

A probe changer is a mechanical rack system that changes components at the kinematic joint. This topic shows examples of working with single and multiple probe changers in the same measurement routine after calibration.

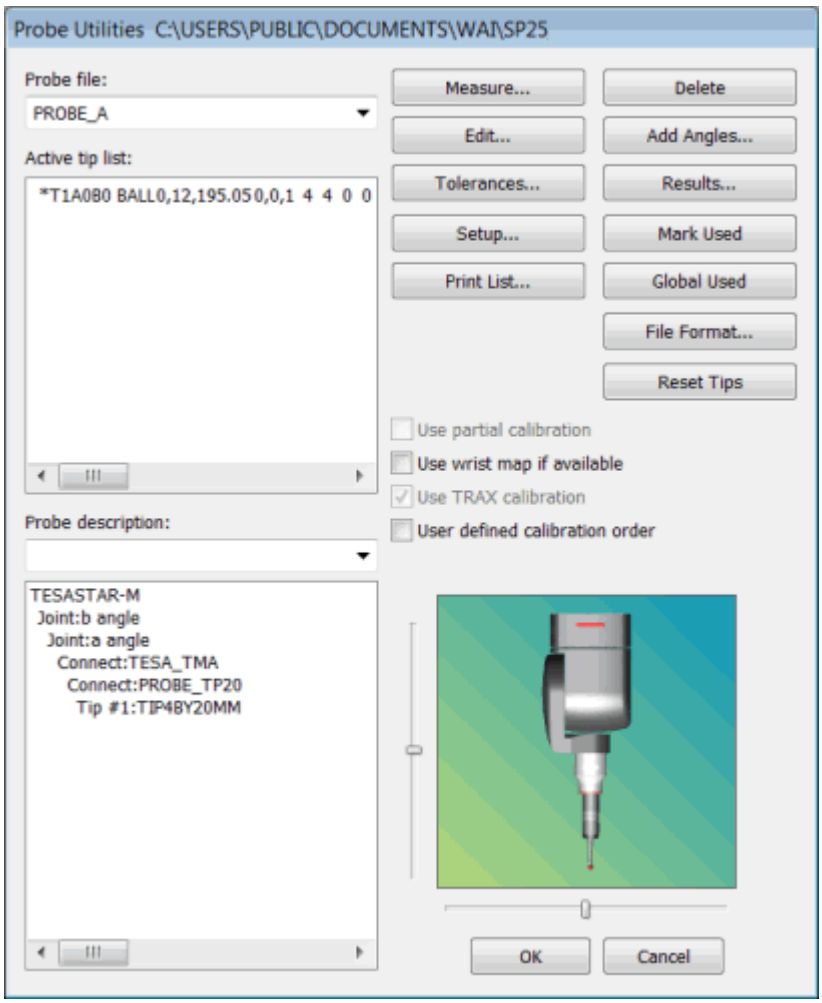
The "Defining Probe Changers" topic in the "Defining Hardware" chapter provides you with a step-by-step description of how to set up and calibrate a probe changer. It also describes how to display an existing probe changer inside the Graphic Display window.

The examples use the TP20, which is a single probe, and the LSPX1. Also, in these examples, the kinematic joint is the Autojoint. The Autojoint is the connection where a key is used to engage the male and female halves of this coupling. For systems with no probe racks, the operator manually uses a key to engage and disengage the autojoint. For rack systems, this is done with a rack and pinion style device to turn the key.

Example - Working with a TP20 Probe

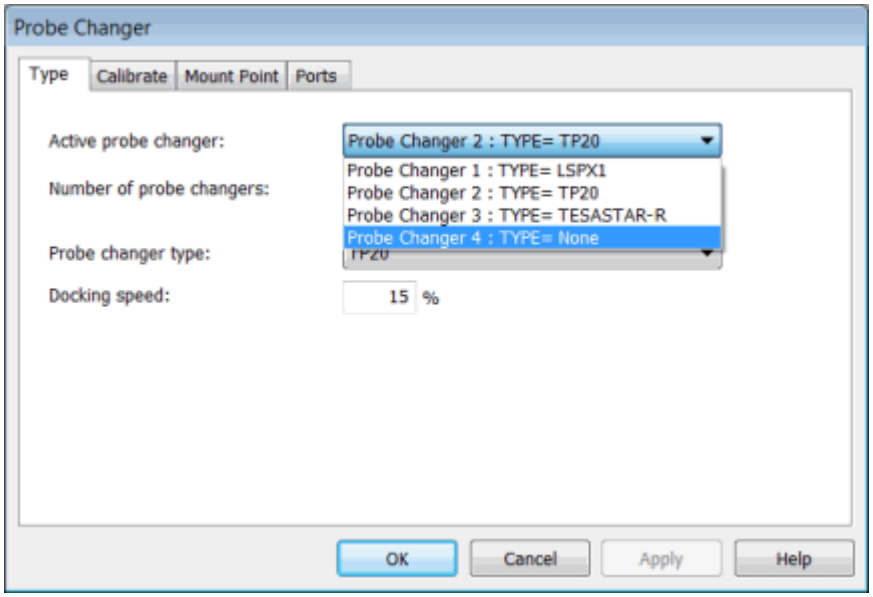
In this example, you would do the following:

1. Set up the TP20 in the **Probe Utilities** dialog box. For example:



TP20 setup in Probe Utilities Dialog Box

2. Select **Edit | Preferences | Probe Changer**. The **Type** tab in the **Probe Changer** dialog box appears:

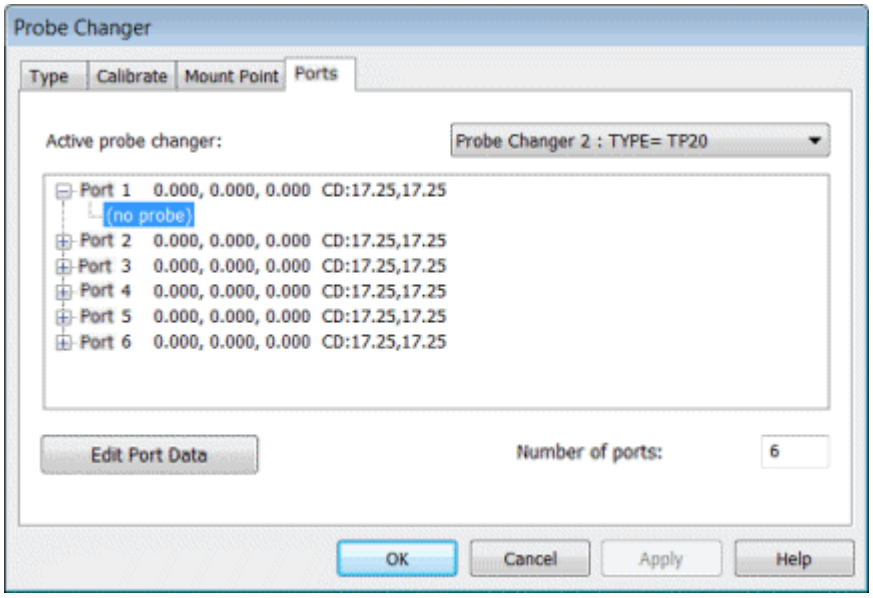


Probe Changer Dialog Box - Type Tab

To use different modules for the TP20 probe and load them automatically, you would use a probe changer. In this case, it is referred to as the "TP20" probe changer in the **Active probe changer** list. The example above shows that the TP20 probe changer (and others) has already been configured.

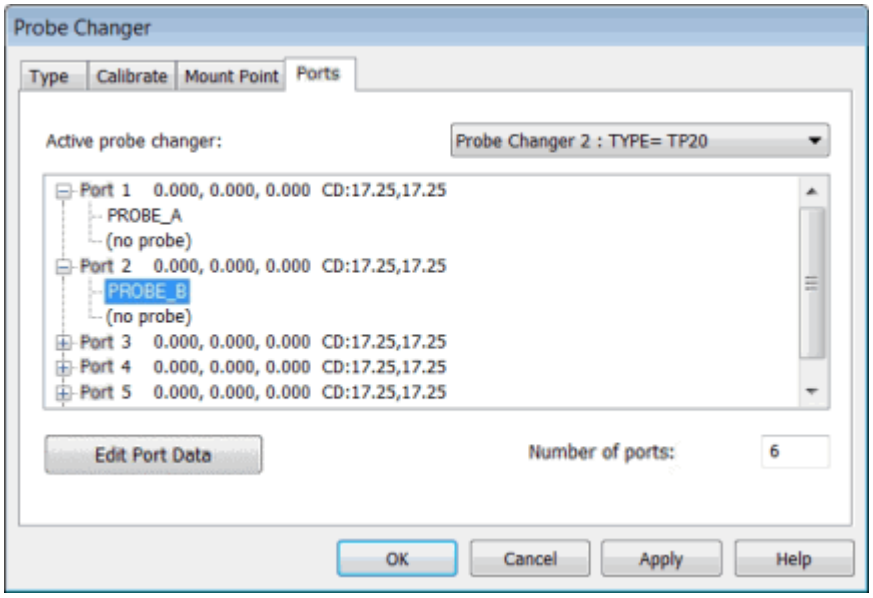
3. Select **TYPE= TP20**.

4. Select the **Ports** tab:



Ports Tab

5. To define the port contents for the TP20 probe changer, click the plus sign (+) to the left of the port number and then assign a probe file to each port from the list that appears. (For help with assigning probe files, see "To Define the Probe Configuration for Each Port".) Repeat this procedure for each port, until you have defined all of the ports that you need to use. For example:



Example showing assigned probe files for the TP20 probe.

You should assign each probe file to only one port. If you are using a single probe changer, and the name of one of the probe files appears in multiple ports, PC-DMIS may not perform as you would expect.

6. Click **Apply**, and then click **OK** to close the dialog box.

7. Use the probes in your measurement routine. To do this, you only need to include the `LOADPROBE` command in your measurement routine where you want to use that probe. When PC-DMIS encounters this command, it moves the machine.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

The machine does the following:

- It drives from the current position on the table to the probe changer's mount point.
- It proceeds to drop off the current probe and then to pick up the new one.
- It returns to the mount point for that probe changer.

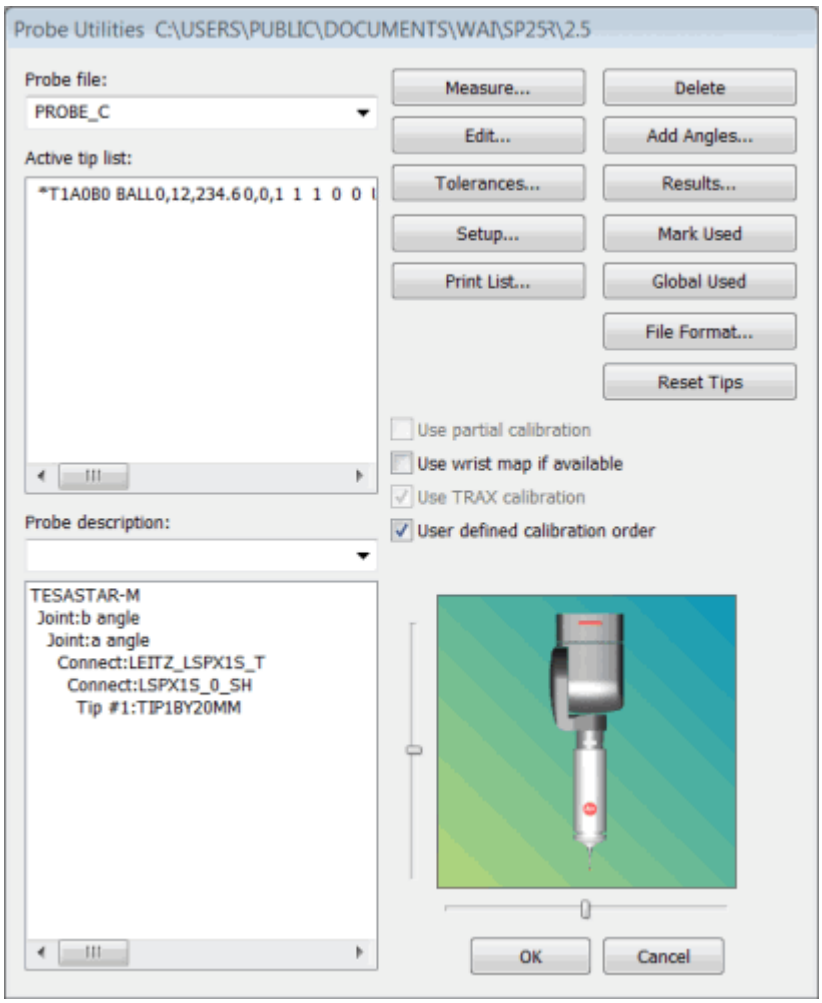
Your measurement routine must contain the necessary Move Point and Clearance Plane commands to ensure safe travel between the last position, the mount point, and the next position in the measurement routine. For more information about these commands, see "Inserting Move Commands".

Example - Working with Two Different Probes

Suppose you have two different probes, and you want to use them in the same measurement routine. This example uses the TP20 and LSPX1. The LSPX1 also has its own probe changer.

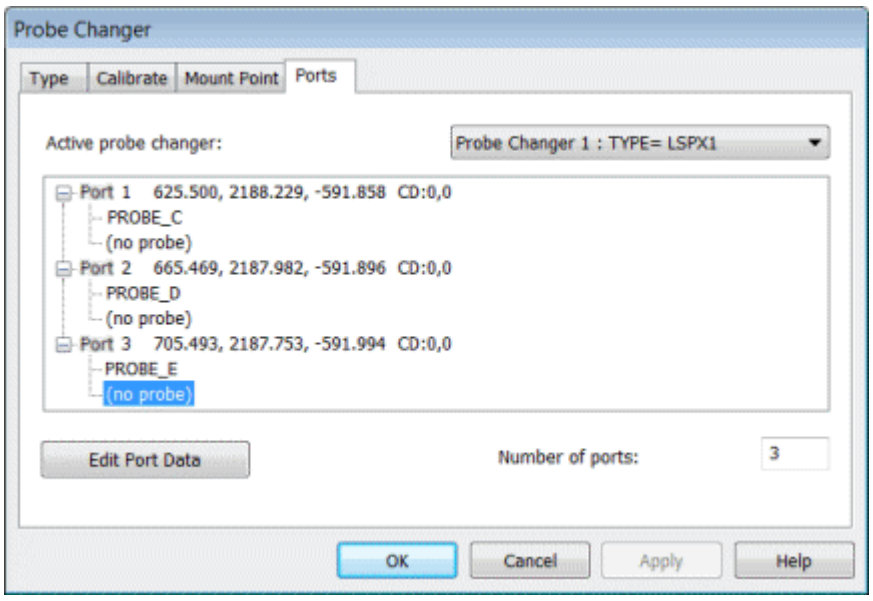
In this example, you would do the following:

1. Set up the LSPX1 in the **Probe Utilities** dialog box. For example:



Example of a LSPX1 probe setup in Probe Utilities dialog box.

2. Select **Edit | Preferences | Probe Changer**. The **Type** tab in the **Probe Changer** dialog box appears.
3. Select **TYPE= LSPX1**.
4. Select the **Ports** tab to define the port contents for the LSPX1 probe changer. For example:



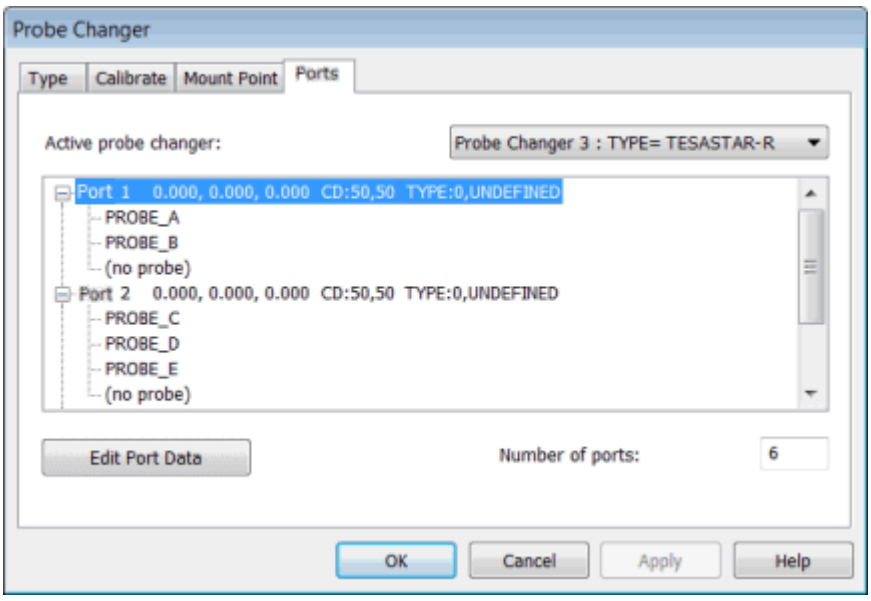
Example showing assigned probe files for the LSPX1 probe.

At this point, you could have a measurement routine that uses each probe from the LSPX1 rack and picks them up automatically. However, suppose that you want a measurement routine that uses a combination of the TP20 and LSPX1 probes. When the changeover occurred, the measurement routine would stop and prompt you to remove one probe and attach the other one. It would then continue to drop off and pick up modules specific to that probe.

To automate this process, you could use a third, intermediate rack. In this example, the third rack is the TESASTAR-R / HR-R. The challenging task is to define the port contents for the TESASTAR-R / HR-R probe changer. In the previous example, the TP20 probe changer has two defined probes: PROBE_A and PROBE_B. In this example, the LSPX1 probe changer has three defined probes: PROBE_C, PROBE_D, and PROBE_E.

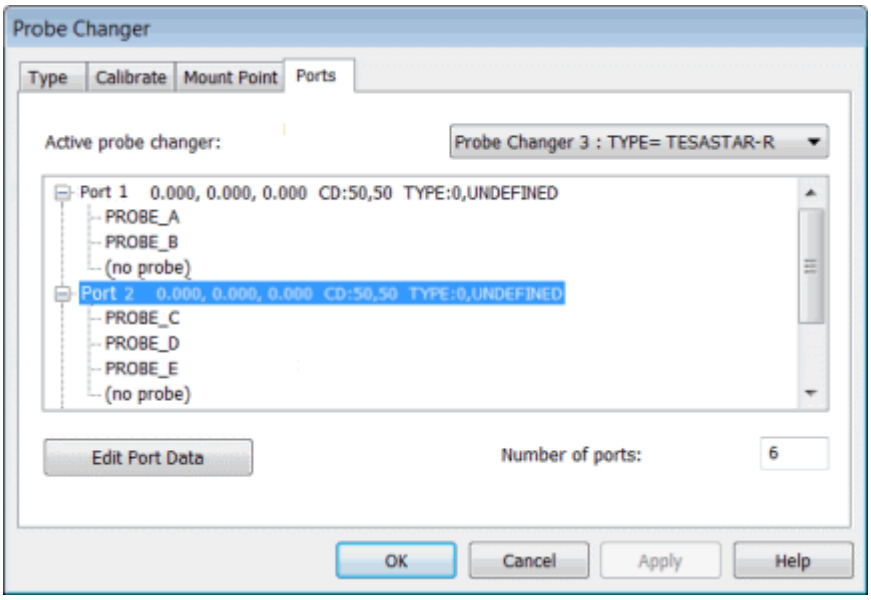
The TESASTAR-R / HR-R does not hold any of the individual modules for the other two probes; it holds only the probe bodies. The body is the upper part of the probe from the kinematic joint down to the magnetic probe coupling. The probe body for the TP20 is associated with two different modules (PROBE_A and PROBE_B). The probe body for the LSPX1 is associated with three other modules (PROBE_C, PROBE_D, and PROBE_E) for that probe.

Therefore, you would define two of the probes in one port and the remaining three probes in the other port, as shown in the example below:



Example showing the Port 1 assigned for the TP20 probe body.

In this case, you have selected the first port of the TESASTAR-R / HR-R probe changer to hold the probe body for the TP20. You have selected the second port to hold the probe body for the LSPX1:



Example showing the Port 2 assigned for the LSPX1 probe body.

Physically, there is only one part in the port. Logically, that one part is associated with multiple probe assemblies. In this example, the TP20 body is used with two other modules and the LSPX1 probe body is used with three other probes. You can have more entries in the port than this example shows; the example is just a guide to explain how the multiple racks interact with each other.

One way to think about this is the following: If any piece of the probe assembly is in that port, you must select the probe file name (of the entire probe assembly) for that port. It is feasible to find the same probe file name in two or more ports because a piece of the probe assembly can be found in that port.

If you had PROBE_B loaded (TP20) and you want to use PROBE_D (LSPX1) when you execute the measurement routine, and PC-DMIS encounters the LOADPROBE command, this is what would happen next:

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

1. The CMM moves to the mount point for the TP20 rack, drops off the PROBE_B module, and then returns to the mount point for this same rack.
2. The CMM then moves to the mount point for the TESASTAR-R / HR-R rack and drops off the TP20 body.
3. From there, the CMM picks up the LSPX1 probe body and returns to the mount point for this rack.
4. The CMM moves to the mount point for the LSPX1 rack, picks up the PROBE_D assembly, and then returns to the mount point for this rack.
5. The CMM continues with the part inspection from there.

If the probes and ports have been properly set up, PC-DMIS drops off the current probe, loads the new probe, and continues executing the measurement routine. This process does not require any operator intervention.

Configuration Error

If PC-DMIS stops DCC execution with a prompt to either unload the current probe or load the probe that you expected to be loaded automatically, a configuration error may have occurred. Possible causes are:

- The same probe is defined in too many ports, and PC-DMIS may not know how to deal with it.
- The probe in question has not been assigned to any port.
- You are using a probe that is incompatible with the probe rack where it is defined.

How to Protect Probes from Probe Changer Collisions

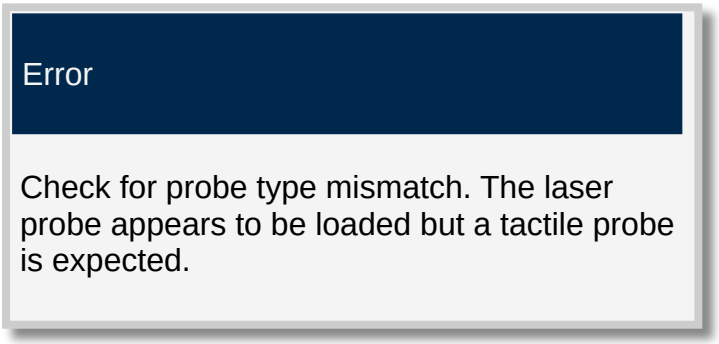
Depending on a variety of circumstances, including user errors, there’s the possibility that a physical crash happens during a probe change cycle. PC-DMIS provides multiple levels of protection against that event.

To avoid this type of situation, a safety mechanism that you can enable is the probe changer collision safety check. For details, see the topic "Protecting Probes from Probe Changer Collisions" in the PC-DMIS Core documentation.

Check to Verify the Location of the Laser Sensor in the Probe Changer

If you mistakenly inform PC-DMIS that you have a tactile probe mounted on the probe assembly head when you actually have a laser sensor mounted, when you attempt a probe change, PC-DMIS performs a proximity check on the laser slot to ensure it doesn't try to drop off the laser sensor in the slot that has the tactile probe.

If this is the case, PC-DMIS shows a warning message similar to this:



You must enable the Settings Editor entry "`CW43LTTest3AxisSlotAlwaysTC`" to use this feature. For details, see the "CW43LTTest3AxisSlotAlwaysTC" topic in the PC-DMIS Settings Editor documentation.

Setting Up the Machine Interface

The **Edit | Preferences | Machine Interface Setup** menu option opens the **Machine Options** dialog box for your machine interface. The **Machine Interface Setup** option is only available when you work in Online mode.

In most cases, *you should not change any of the values* in the **Machine Options** dialog box. Some items in this dialog box, such as the items in the **Mechanical Offsets** area, permanently overwrite values stored for your machine on the controller's hard drive. For questions on how and when to use the **Machine Options** dialog box, please contact your local service representative.

For information on this dialog box, please refer to the "Machine Options Dialog Box" topic in the Machine Interface Installation Manual (MIIM). The MIIM is typically used only by Hexagon service engineers.

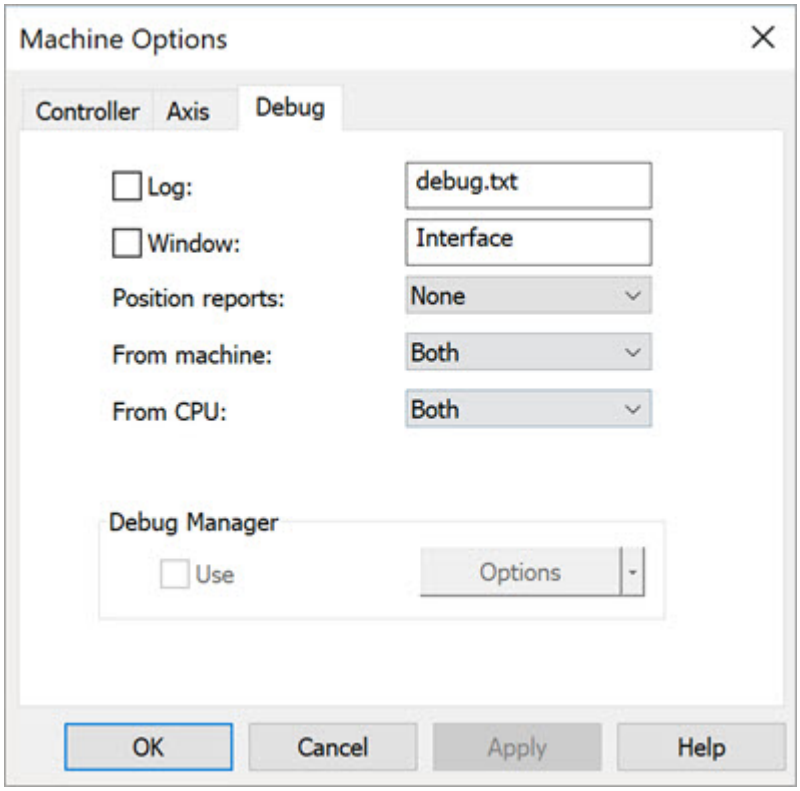
Generating a Debug File

You can tell PC-DMIS to generate a debug file. This special text file contains all of the communication data between PC-DMIS and the CMM during measurement routine execution. A debug file can help Hexagon Technical Support resolve certain problems that involve your CMM.

The debug file lists all of the commands that PC-DMIS sends to the CMM, the responses it receives, and the error messages that the controller generates. If you encounter repeatable problems that involve the movement of your CMM in Online mode, you may want to submit that to Hexagon Technical Support.

To generate a debug file, follow this procedure:

1. From the **Machine Options** dialog box (**Edit | Preferences | Machine Interface Setup**), select the **Debug** tab.



Machine Options Dialog Box - Debug tab when connected to non-vision machines

> Be sure to modify only the **Debug** tab. The other tabs on the **Machine Options** dialog box are used by service personnel to set up your machine.

2. Select the **Log** check box.

3. Type a name for the debug file in the box next to the **Log** check box. The default file name is debug.txt. You can also precede the file name with a full path to specify the drive and directory where PC-DMIS should send the debug file.

4. Click **Apply** and then **OK** to close the dialog box.
5. Execute your measurement routine. When you encounter the error, immediately exit PC-DMIS.
6. Navigate to the directory that contains your debug file.
7. Rename the debug file. If you don't rename the debug file, the next time you start PC-DMIS, it automatically overwrites all existing data in the debug file. This results in a loss of data that may be necessary for debugging purposes.
8. Send the debug file, measurement routine file (.prg), probe file (.prb), and any other necessary files to Hexagon Technical Support.

By default, PC-DMIS sends the debug file to the ProgramData directory. This is typically "C:\ProgramData\Hexagon\PC-DMIS\<version>", where <version> is the version of PC-DMIS you're running.

For details on the default locations for PC-DMIS files, see "Understanding File Locations".

Starting a Fresh Debug File

You can use the `DebugLogReset` entry in the Settings Editor application to specify whether to overwrite all existing data in the debug file or append new data to the file each time you execute a measurement routine. For details, refer to "DebugLogReset" in the "Option" section in the Settings Editor documentation.

Setting Additional Debug Options

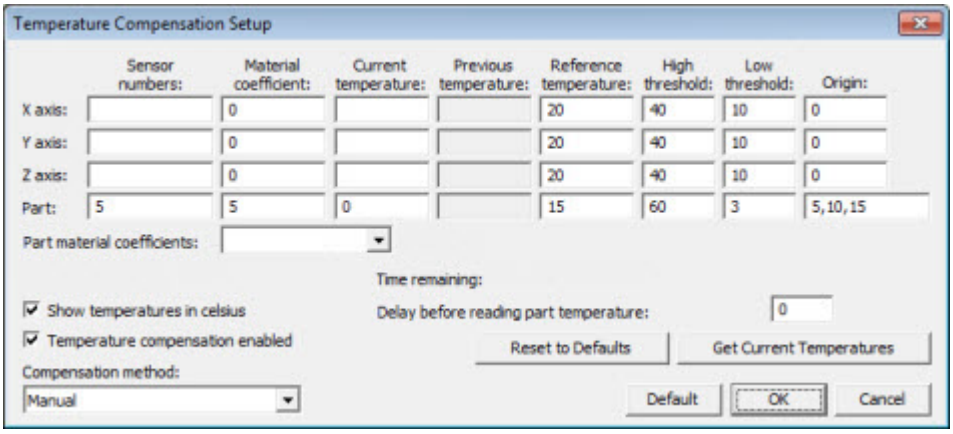
You can control the type of debug information that PC-DMIS logs and where it is sent.

- **Window** check box: To display debug information in a window for review, select this check box and enter a name for the window in the box.
- **Position reports** list: To log position reports from PC-DMIS, select **None**, **Both**, **Log**, or **Window**.
- **From machine** list: To log debug information that your machine sends to your computer, select **None**, **Both**, **Log**, or **Window**. Some machine interfaces do not support this option.
- **From CPU** list: To log debug information that your computer sends to your machine, select **None**, **Both**, **Log**, or **Window**. Some machine interfaces do not support this option.

Vision Machine Focus Debug Option

When connected to a vision machine, the **Focus** mode option is available. For details, see "Machine Options: Debug Tab" in the PC-DMIS Vision documentation.

Compensating for Temperature



Temperature Compensation Setup Dialog Box

To open the **Temperature Compensation Setup** dialog box, select **Edit | Preferences | Temperature Compensation Setup**.

> To set up temperature compensation for your system as shown in this legacy section, you need to start PC-DMIS as an administrator. In addition, the value for the `UseTemperatureCompensationV2` entry must be False.

This dialog box enables you to compensate for room and part temperature, which increases the accuracy of the inspection process.

To compensate for temperature during multiple arm calibration, see "Using Temperature Compensation with Multiple Arm Calibration" in the "Using Multiple Arm Mode" chapter.

To use advanced options for temperature compensation, see "Using Advanced Temperature Compensation" in the Toolkit Modules documentation. Advanced options enable you to use the continuous temperature compensation ability of the Firmware Distributed Controller (FDC) and Hexagon B3C controller.

Using STP Files

Each CMM uses machine-specific parameters for temperature compensation for each axis, such as thermal coefficients and sensor assignments. These parameters, along with embedded code that tells PC-DMIS if the file is a structural or linear compensation file, are stored inside special STP files (named for the .stp extension) created by your CMM vendor.

PC-DMIS requires these STP files to compensate for temperature.

Prior to compensating for temperature, ensure that these STP files reside in the appropriate locations on your hard drive:

- The DEA Structural Thermal Compensation method expects to find the Serv1.stp file in the *C:\Thermal_OCX* directory.
- The DEA Linear Thermal Compensation method expects to find the Serv1.stp file in the *C:\Program Files\Thermal_OCX* directory.

Linear and Structural Temperature Compensation

Linear compensation = (Thermal Coefficient of Expansion) X (Displacement + Change in Temperature of Each Axis and the Part). If multiple temperature sensors exist on an axis, PC-DMIS averages the readouts to find the temperature change.

Structural compensation recognizes that a CMM's various material components may have different temperatures (a single axis of the machine, for example, may have several different temperatures, which cause bending, bowing or skewing of the machine in some way). Structural compensation, then, applies the temperature corrections for specific areas of the CMM. When you select **Edit | Preferences | Temperature Compensation Setup**, the Structural Thermal_OCX gets called, and PC-DMIS calculates a new temporary volumetric compensation map.

Available Input Parameters

The following parameters are in the **Temperature Compensation Setup** dialog box (**Edit | Preferences | Temperature Compensation Setup**):
Temperature Compensation Setup Dialog Box

Sensor numbers boxes

The **Sensor numbers** boxes contain a list of one or more sensor numbers to be used for the given axis or part. These values are very important when reading temperatures from the controller because they *must* correspond to how the sensors are actually configured.

- Each sensor is a number in the range of 1 to 32.
- Items in the list can be either a single number or a range from first to last.
- Items are separated by commas or left blank.
- The input allows up to 32 values for any axis or the part.

For "manual" mode, these numbers are relatively meaningless, but at least one sensor number must be assigned for each axis and the part.

Material coefficient boxes

The **Material coefficient** boxes contain numbers that reflect the material property. They are the fractional change in length per unit change in temperature.

- The values vary depending on what type of material was used to make the scales on the machine axes and on the part's material.
- The units are meter per degree C.

A scale with a coefficient of 11.5 microns is written as 0.0000115 Meters/Degree/C.

Acceptable Part Values on Leitz Machines

If you are using a Leitz machine, PC-DMIS forces you to use an acceptable material coefficient value in the **Part** box. An acceptable value lies within the range of -0.001 through 0.001 Meters/Degrees/Celsius.

- If you enter a value outside of this acceptable range in the **Temperature Compensation Setup** dialog box, a warning message appears, and the value in the **Part** box is reset to 0.0.
- If you open a measurement routine created from an older version of PC-DMIS that permitted unacceptable values, a message box now informs you of the unexpected material coefficient value, and PC-DMIS resets the value to 0.0.
- If you attempt to manually edit the `TEMPCOMP/ORIGIN` command to use an unacceptable value, PC-DMIS displays an error message in the **Execution** dialog box during execution. The message informs you that the part material coefficient value is out of range. The only option is to click **Cancel**, and modify the command to contain an acceptable value.

Part material coefficients list

The **Part material coefficients** list contains a list of standard material types. Selecting one of these materials automatically puts its associated coefficient in the **Material coefficient** box.

Part materials and coefficients are stored in the MaterialCoefficients.xml file. You can use a text editor or the Material Coefficients Editor to modify this file. For information, see "Editing Part Materials and Coefficients".

Current temperature boxes

The **Current temperature** boxes contain the current temperatures in the appropriate units. You can either type them or read them in from the controller, depending on the type of machine available and the selected options.

Previous temperature boxes

The **Previous temperature** boxes always contain the previously-read temperatures. If no temperatures were previously read in, these values are either zero or left blank.

Reference temperature boxes

The **Reference temperature** boxes contain the reference temperature from which temperature compensation adjustments need to be applied.

- The amount of correction to be applied is based on multiplying the material coefficient by the difference between the current and reference temperatures.

  Amount of Correction = Material coefficient x (Current temperature – Reference temperature)

- If the current temperature is the same as the reference temperature, the net effect is that no thermal compensation adjustment is applied.
- The value in these boxes is almost always 20 degrees C, or the Fahrenheit equivalent.

High threshold boxes

The **High threshold** boxes contain an upper limit (in the appropriate units) on the current temperature above which no further thermal compensation will be applied. PC-DMIS doesn't display any warning or error message.

With a reference temperature of 20 degrees C, a current temperature of 35 degrees C, and a high threshold of 30 degrees C, the amount of correction actually applied would be based on a difference of (30 – 20) instead of (35 – 20) because the current temperature exceeded the upper limit.

Low threshold boxes

The **Low threshold** boxes is conceptually like the high threshold except it provides a lower limit on the current temperature below which no further thermal compensation will be applied.

Origin boxes

The **Origin** boxes in the **Temperature Compensation Setup** dialog box (**Edit | Preferences | Temperature Compensation Setup**) determine the length of the item to which thermal compensation is being applied.

Length Value = Current Position Value – Origin Value

The X, Y, and Z values in the **Origin** boxes are zero most of the time. However, some types of machines do not use zero for the origin of their scales.

Typically, the **Part** value is also zero unless there is some special type of fixture constraint. The part origin coordinates can be inserted into the coordinate system for a machine or an active alignment. The type of coordinate system depends on where the temperature compensation command (`TEMPCOMP`) is inserted into the measurement routine:

- If you insert a `TEMPCOMP` command before an alignment, the part origin coordinates are expressed in the machine's coordinate system. For example:

```
          TEMPCOMP/ORIGIN=376.627,293.461,-489.749 Material Coeff=0.0000113,Reference Temp=20
            ,Hi Threshold=40,Lo Threshold=10,Sensor num=8
            ,X Axis Temp=,Y Axis Temp=,Z Axis Temp=,Part Temp=26.797
          MOVE/POINT,NORMAL,<292.876,360.313,-394.495>
A1        =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
            ALIGNMENT/LEVEL,ZPLUS,
            ALIGNMENT/ROTATE,XPLUS,TO,,ABOUT,ZPLUS
            ALIGNMENT/TRANS,ZAXIS,
            ALIGNMENT/TRANS_OFFSET,ZAXIS,-9
            ALIGNMENT/TRANS,YAXIS,
            ALIGNMENT/TRANS,XAXIS,
          ALIGNMENT/END
```

- If you insert a `TEMPCOMP` command after an alignment, the part origin coordinates are expressed in the alignment's coordinate system. For example:

```
A1        =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
            ALIGNMENT/LEVEL,ZPLUS,PLN1
            ALIGNMENT/ROTATE,XPLUS,TO,LIN1,ABOUT,ZPLUS
            ALIGNMENT/TRANS,ZAXIS,PLN1
            ALIGNMENT/TRANS_OFFSET,ZAXIS,-9
            ALIGNMENT/TRANS,YAXIS,LIN1
            ALIGNMENT/TRANS,XAXIS,PNT1
          ALIGNMENT/END
          MODE/DCC
          MOVE/POINT,NORMAL,<0,0,50>
          TEMPCOMP/ORIGIN=100.008,17.576,4.502 Material Coeff=0.0000113,Reference Temp=20
            ,Hi Threshold=40,Lo Threshold=10,Sensor num=9
            ,X Axis Temp=,Y Axis Temp=,Z Axis Temp=,Part Temp=26.784
```

Show temperatures in celsius check box

The **Show temperatures in celsius** check box affects both the temperatures and the material coefficient.

- If you select this check box, PC-DMIS uses degrees Celsius.
- If you clear this check box, PC-DMIS uses degrees Fahrenheit.

Temperature compensation enabled check box

The **Temperature compensation enabled** check box tells PC-DMIS to use temperature compensation.

- If cleared, PC-DMIS does not perform any temperature compensation, and the `TEMPCOMP` command (if present in the measurement routine) has no effect.
- If marked, PC-DMIS behaves according to the input parameters.

If you mark this check box and choose one of the compensation methods where PC-DMIS does the part compensation rather than the machine controller, the diameter for probes on non-portable machines in the **Edit Probe Data** dialog box may vary with the current part temperature. See "Edit Probe Data" in the "Defining Hardware" chapter.

Compensation method list

Following are the compensation methods and their respective processes in PC-DMIS.

For Sheffield controllers, you *must* define the **Material coefficient** and **Reference temperature** boxes and then click **Default**, regardless of the compensation method you are using.

Manual

- Compensation is manual (controlled by your input) with no involvement from the machine controller.
- PC-DMIS performs all compensation calculations.
- During measurement routine execution, the **Temperature Compensation Setup** dialog box opens. You can edit the current settings before you proceed with the rest of the measurement routine.

Read Temperatures from Controller

- When you use a machine that supports this option, PC-DMIS reads the current temperatures from the controller automatically rather than you supplying the data.
- PC-DMIS performs all compensation calculations. The controller only provides the current temperatures.
- During measurement routine execution, the **Temperature Compensation Setup** dialog box does not open.
- The measurement routine does not pause for your confirmation.

To retrieve the CTE (Coefficient of Thermal Expansion) values for the axes for Sheffield controllers, click the **Get Current Temperatures** button.

Controller Compensates Axes Only

- The controller performs compensation of the machine axes itself.
- The inputs for the axes are not used.
- Part input parameters apply since PC-DMIS still performs compensation for the part.
- During measurement routine execution, the **Temperature Compensation Setup** dialog box does not open.
- The measurement routine does not pause for your confirmation.

Controller Comps Axes and Part

- The controller performs compensation of both the machine axes and the part.
- The inputs for the axes are not used.
- PC-DMIS does not perform any compensation calculations.
- The input for the part for the material coefficient, reference temperature, and origin must still be supplied because PC-DMIS must pass that information down to the controller.
- The input for the part channel is optional. If it is provided, PC-DMIS sends the temperature reading of this channel (or the average in case of multiple sensors) to the controller instead of the temperature reading associated with the part sensor defined in the controller configuration.
- During measurement routine execution, the **Temperature Compensation Setup** dialog box does not open.
- The measurement routine does not pause for your confirmation.

For Sheffield controllers, you do not need to enter the CTE values for the axes.

Time remaining display

The **Time remaining** display shows the time remaining before the temperature reading takes place. This only displays if you set up a delay period for execution. See "Delay Before Reading Part".

Delay before reading part temperature box

The **Delay before reading part temperature** box enables you to specify a period that PC-DMIS will wait during measurement routine execution before reading the sensors to obtain the current temperatures. If you enter zero, then there is no pause.

Reset to Defaults button

The **Resets to Defaults** button updates any previously modified values with the previously saved values. If this is on a DEA machine and a serv1.stp is available, then PC-DMIS reads that file.

Get Current Temperatures button

If you select the **Read Temperatures from Controller** method in the **Compensation method** list, and if you use a machine that supports this option, the **Get Current Temperatures** button causes PC-DMIS to read the current temperatures from the controller and display them in the **Temperature Compensation Setup** dialog box.

TEMPCOMP/ORIGIN Command in the Edit Window

When you click the **OK** button to accept the inputs in the **Temperature Compensation Setup**, PC-DMIS inserts a `TEMPCOMP/ORIGIN` command into the measurement routine.

```
TEMPCOMP/ORIGIN=0,0,0,CTE=0.000012778,Reference Temp=73
  ,Hi Threshold=32,Lo Threshold=32
  ,PART SENSOR NUM=4
  ,X AXIS TEMP=68,Y AXIS TEMP=68,Z AXIS TEMP=68,PART TEMP=68
```

Command Mode

Normally, a measurement routine only uses one `TEMPCOMP` command. The `TEMPCOMP` command should be placed near the top of the measurement routine prior to any measurements. When you execute the measurement routine, it executes according to the various input parameters.

Controller Support

Not all compensation methods are supported by all controllers. The following are the supported controllers for the different compensation methods. For detailed information about compensation methods, see "Compensation method list".

Compensation Method	Supported Controllers
Manual	All since there is no controller involvement with this method.
Read Temperatures from Controller	DEA (only with DEAC family controllers), and Sharpe32z with the Leitz protocol.
Controller Compensates Axes Only	Sharpe32z with the Leitz protocol, and dual-arm configurations with an FDC interface.
Controller Comps Axes and Part	Sharpe32z using the Leitz protocol, Sheffield, stand-alone configurations with an FDC interface, and single-arm configurations with an FDC interface.

Local Temperature Settings

When it opens a measurement routine containing a TEMPCOMP command, PC-DMIS verifies the part sensor number with the local settings.

- If the values differ, PC-DMIS automatically updates the command to reflect the current settings, and a comment is inserted into the measurement routine reflecting the old and new values.
- If no local setting is available for the part sensor, PC-DMIS marks the TEMPCOMP command RED in the Edit window.

Using Simplified Temperature Compensation

Simplified temperature compensation is available for machines that are connected to Hexagon controllers. Currently, only stationary products, Leica Trackers and Romer Portable Arms are supported.

You can access Simplified Temperature Compensation through the menu (**Edit | Preferences | Temperature Compensation Setup**).

The following product lines and machine types are not supported. They must use the original temperature compensation command.

- PC-DMIS NC
- Non-Hexagon controllers and equipment
- Controllers that do not support temperature sensors
- Controllers with the Sheffield interface
- DEA controllers

> ⚠ If you use a controller with the Sheffield interface, you must set the `UseTemperatureCompensationV2` entry in the **Option** section of the PC-DMIS Settings Editor application to **FALSE**. If you set it to **TRUE**, that entry may affect your measurement routine's move points in unexpected ways.

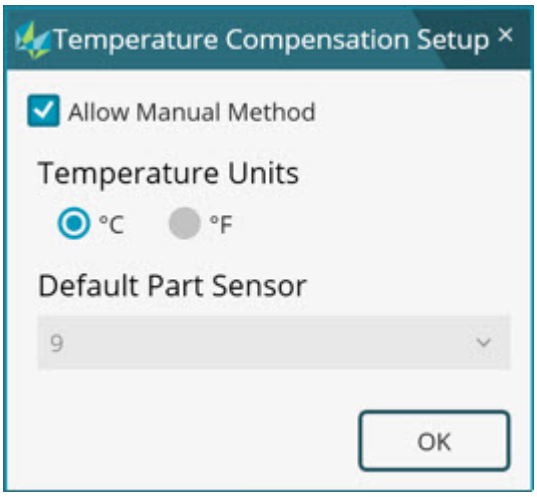
Temperature Compensation Setup

To set up your system, select **Edit | Preferences | Temperature Compensation Setup**.

> ❗ To set up temperature compensation for your system as shown in this topic, you need to start PC-DMIS as an administrator. In addition, the value for the `UseTemperatureCompensationV2` entry must be True.

Hexagon Systems

For a machine that is connected to a Hexagon controller, the following options appear in the **Temperature Compensation Setup** dialog box:



Hexagon system

Allow Manual Method check box:

- In special cases, if you would like to manually enter the part temperature, select this check box. Both the automatic and manual methods of temperature compensation become available.
- If you clear this check box, only the automatic method of temperature compensation becomes available in the **Temperature Compensation** dialog box. By default, the check box is cleared. Hexagon recommends use of only the automatic method where available.

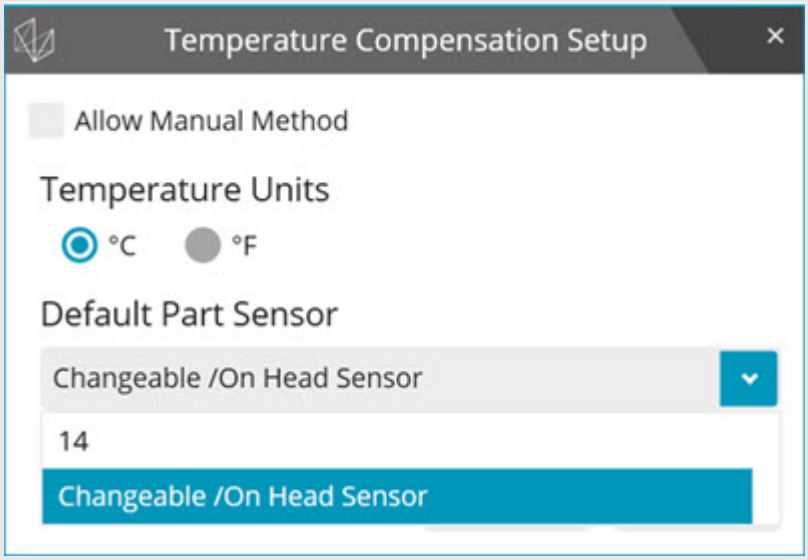
Temperature Units - Select degrees Celsius or degrees Fahrenheit.

Default Part Sensor list - Most machines are supplied with one part temperature sensor. This list displays the sensor number with which it is connected to the controller when the machine is online.

Some machines have multiple part temperature sensors. If your system is configured with multiple sensors, multiple sensor numbers appear in this list.

In PC-DMIS version 2023.1 and earlier, when the software detected the "Changeable Part Sensor" setup in the CMM configuration, PC-DMIS did not consider any of the standard sensors. This made it so the "Changeable Part Sensor" was always set as the default sensor.

Starting with PC-DMIS version 2023.2, for Leitz CMM's (Bx family of controllers), you can now configure the setup to default to "Changeable /On Head Sensors" or a "Standard" sensor.



This is only for V2 Temperature Compensation and only if the sensors are present in the CMM configuration.

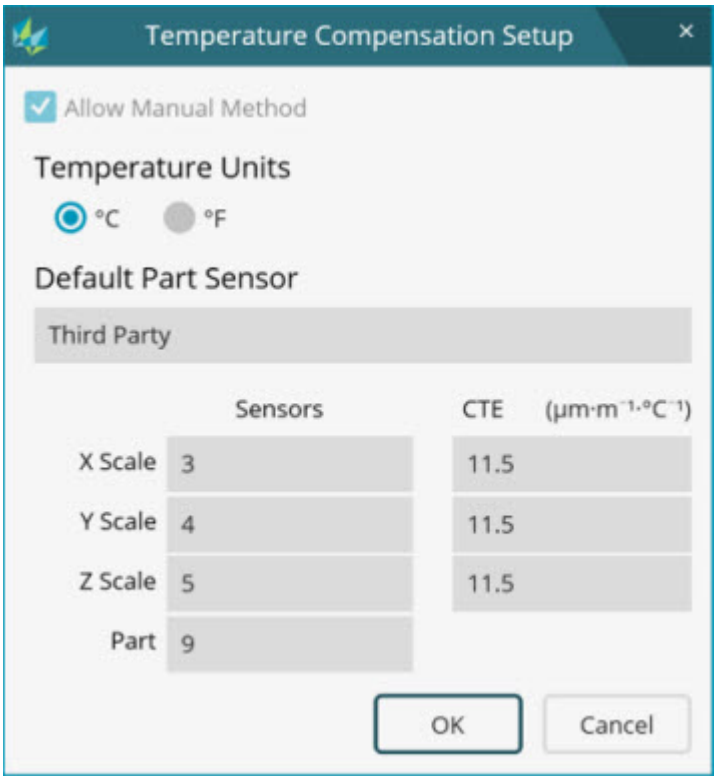
Select a sensor number that you want to designate as the default number. The advantage of using a default sensor number is portability of the measurement routine to another system with a different default sensor number.

If PC-DMIS is connected to a Leica Tracker device, the default part sensor displays as **Tracker**.

OK button - To save your settings, click **OK**.

Non-Hexagon Systems

For a machine that is connected to a non-Hexagon controller, the following options appear in the **Temperature Compensation Setup** dialog box:



Non-Hexagon system

Allow Manual Method check box - It may or may not be possible for PC-DMIS to get the temperature of axes and parts from the controller. When PC-DMIS cannot automatically read the temperatures, the only temperature compensation option is the manual entry of temperatures. This check box is selected by default. You must enter the CTE values for the scale of each axis to compensate for temperature of scales.

You need to enter the temperature of the X scale, Y scale, Z scale, and part at the time of measurement routine execution.

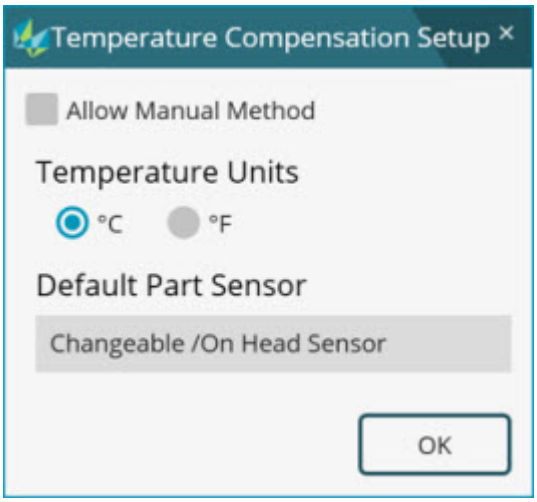
In some cases, it may be possible for PC-DMIS to read the temperature from the controller. In these cases, enter the sensor numbers and CTE values for the scale of each axis and part.

Default Part Sensor list - This list, and the **Part Sensor** area in the **Temperature Compensation** dialog box, display **Third Party**.

OK button - To save your settings, click **OK**.

Changeable or On-Head Part Temperature Sensor

If PC-DMIS determines that a changeable or on-head temperature sensor is mounted on the probe head, it displays that information in the **Default Part Sensor** area of the dialog box:



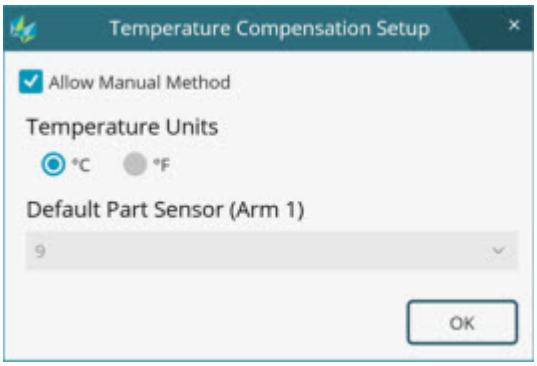
Changeable or on-head sensor

For more information on changeable or on-head sensors, see "Temperature Compensation".

> PC-DMIS does not support a changeable or on-head part temperature sensor and a magnetic sensor on the same machine.

Temperature Compensation with a Multiple Arm CMM

If PC-DMIS is connected to a multiple arm CMM, the default part sensor is one of the part sensors attached to Arm 1. Multiple Arm Mode allows the use of part sensors attached to Arm 1 only.



Multiple arm

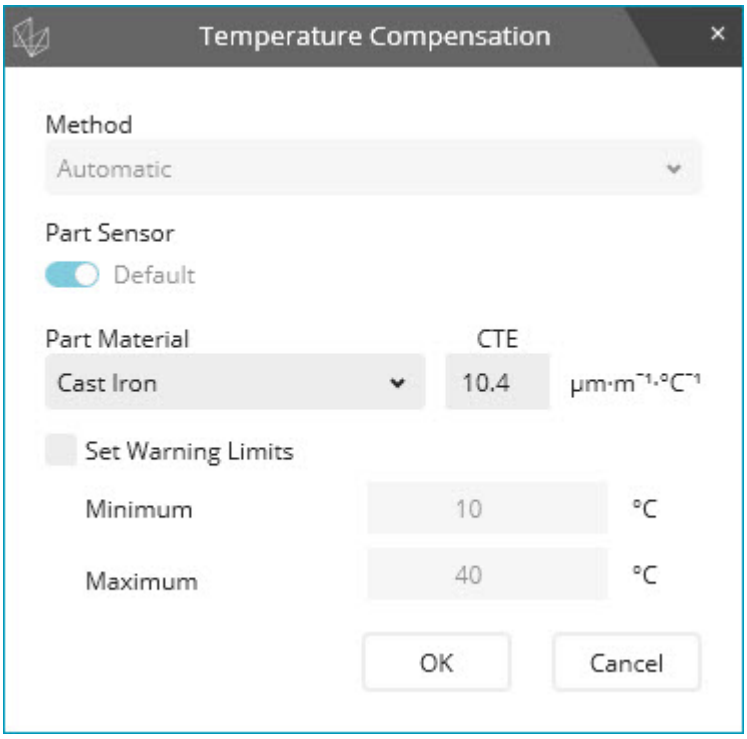
For a multiple arm CMM, note the following:

- Part sensors attached to Arm 1 can only be used to measure the temperature of the part.
- Each arm compensates for its own axis. PC-DMIS compensates for the part.
- You must ensure that the arms are synchronized before and after temperature compensation. To synchronize the arms, insert `MOVE/SYNC` commands before and after the `TEMPCOMP` command.

For more information about multiple arms, see the "Using Multiple Arm Mode" chapter.

Temperature Compensation

To compensate the measurement for a temperature other than 20 °C as required by ISO-1, select **Insert | Parameter Change | Temperature Compensation**. The following dialog box appears:



Temperature Compensation Dialog Box

Method list - From the list, select the temperature compensation method:

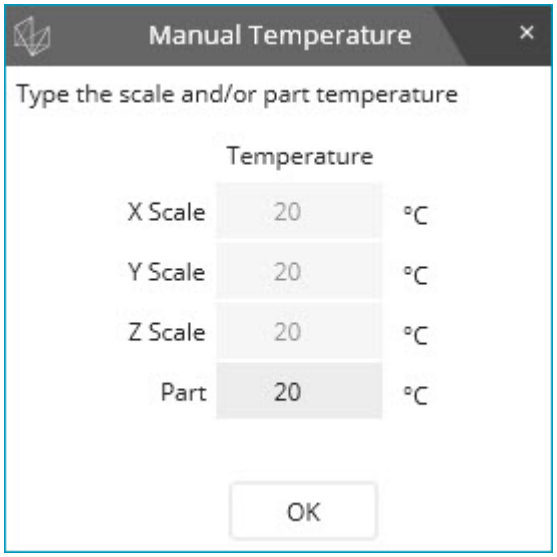
- **Automatic** - The automatic method is the default method. It is available for most machines that are connected to a Hexagon controller. These controllers can measure the temperature using a sensor that is mounted on scales and has one or more sensors that can be attached to a part.

  If you select this method, PC-DMIS reads the temperatures from the controller. You do not need to enter any temperatures.

- **Manual** - Third-party equipment (non-Hexagon controllers) may not support the automatic method of temperature measurement. For these controllers, only the manual method is available. For this method, you measure and enter the temperature at the time of measurement routine execution.

  The manual method is available if the **Allow Manual method** check box is selected in the **Temperature compensation setup** dialog box.

  If you select this method, the **Manual Temperature** dialog box appears during measurement routine execution:



Manual Temperature Dialog Box

  - For a Hexagon CMM, PC-DMIS reads and enters the X scale, Y scale, and Z scale temperatures from the controller. You cannot change the temperatures.
  - You must use a custom sensor to measure the part temperature. Enter the temperature in the **Part** box.
  - If the controller does not support a temperature sensor, you must enter the X scale, Y scale, Z scale, and part temperatures.

  To continue with measurement routine execution, click **OK**.

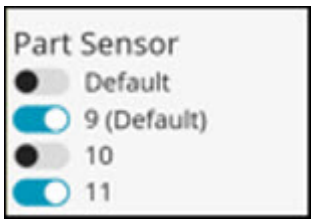
If you are using a Leica Tracker device, select the appropriate temperature compensation method:

- If your Tracker device is equipped with a part temperature sensor that can read the part temperature, you can select either the automatic or manual compensation method. If you select the automatic method, PC-DMIS reads the temperature as measured by the Tracker's part temperature sensor. Ensure that the part temperature sensor is connected to your Tracker and is in contact with the part during measurement.

- If your Tracker device is not equipped with a part temperature sensor, only the manual compensation method is possible. During measurement routine execution, you will need to enter the current part temperature in the **Part** box in the **Manual Temperature** dialog box. You can use any external device to measure the part temperature.

Part Sensor - The option that appears in this area depends on the type of part sensor:

- One part temperature sensor - Usually, a controller is equipped with only one part sensor. This sensor appears in the dialog box as **Default**. You cannot cancel the selection of this sensor.
- Multiple part temperature sensors - Some controllers support multiple part sensors. If your controller is equipped with multiple part sensors, each part sensor number appears in this area. For example:



Part Sensor area showing multiple part sensors.

Select the sensor or sensors that are connected to the part being measured. Ensure that you select the correct sensor number. If you select multiple sensors, PC-DMIS uses the average temperature for thermal compensation.

- External temperature sensor - You may need to use an external sensor to measure the part temperature. In this case, you can use a variable to determine the temperature of a part. This sensor appears in this area as follows:

For more information, see "Variable for Part Temperature in Temperature Compensation Command".

- Changeable or on-head part temperature sensor - The Hexagon FDC and Leitz controllers on fixed-head CMMs can be equipped with a changeable or on-head part sensor that is mounted on the probe head. The controller can also be equipped with a probe-like sensor that can measure the part temperature by probing a point on the part. These sensors appear in this area as follows:

Part Sensor area showing the Changeable / On Head Sensor option.

PC-DMIS can determine whether the part sensor on the Leitz controller is a changeable or an on-head part sensor. For the FDC controller, the `UseChangeableTemperatureProbes` entry in the PC-DMIS Settings Editor must be set to True.

For information about creating and using a changeable temperature probe file, see "Working with Temperature Sensors" in the PC-DMIS CMM documentation.

- For a machine that is connected to a non-Hexagon controller, the sensor appears as **Third Party**. For more information, see "Temperature Compensation Setup".

Part Material list - From the list, select the material of the part. The material's coefficient of thermal expansion (CTE) appears in the **CTE** box.

Part materials and coefficients are stored in the MaterialCoefficients.xml file. You can use a text editor or the Material Coefficients Editor to modify this file. For complete information, see "Editing Part Materials and Coefficients".

CTE (Coefficient of Thermal Expansion) - This is a unique value. When you select a part material, the coefficient value appears in the **CTE** box. You cannot modify this value in the dialog box, but you can change it in the Edit window. When you modify the CTE value in the Edit window and press Tab, PC-DMIS attempts to find a matching material. It searches through the available materials in the MaterialCoefficients.xml file. It then displays the material that corresponds to the typed value. If it cannot find a corresponding material, PC-DMIS sets the MATERIAL field in the Edit window to "Custom Material". When you next open the **Temperature Compensation** dialog box, you will see "Custom Material" in the **Part Material** list. You can select any other material from the list.

The CTE field in the Edit window also supports variables. If you use a variable in this field, PC-DMIS uses the current variable value during execution. If you type a variable name for the CTE field in the Edit window, the material appears as "Variable". If you press F9 to edit this command, the material appears as "Variable", and the CTE is the current value of the variable. You cannot select another material in the **Part Material** list. To change the material, you need to remove the variable name from the CTE field in the Edit window.

The CTE value's units are in micron per meter per degree C (or µm·m-1·°C-1). Before version 2017 R2, this was in meter per degree C. This means, for example, that a previous value of 0.0000115 now appears as 11.5.

The CTE value must be a unique value. If you have two or more materials that have the same CTE value, you must enter them on the same row in the Material Coefficients Editor separated by a forward slash.

For example, if Material1 has the same CTE value as Material2, you must enter "Material1/Material2" in the **Materials** column. In the corresponding **Coefficients** column, you would enter the CTE value.

For information on how to use the Material Coefficients Editor, see the "Editing Part Materials and Coefficients" topic.

Based on the CTE value, PC-DMIS will search for the corresponding material in the MaterialCoefficients.xml file.

Set Warning Limits check box - The `TEMPCOMP` command displays a warning if the temperature of the X scale, Y scale, Z scale, or part is outside of the limit.

The `TEMPCOMP` command also supports the "Temperature Outside Limits" error type.

You can use the options in the **On Error** dialog box to control the measurement only if the specified temperature conditions are met. For detailed information about setting up the options in the dialog box, see "Branching on an Error".

Minimum and **Maximum** boxes - These boxes display temperatures as follows:

- If you select the **Set Warning Limits** check box, PC-DMIS sets the suggested minimum temperature to 18 °C and the maximum temperature to 22 °C. You can modify the values.
- If you clear the **Set Warning Limits** check box, the minimum temperature is set to 10 °C and the maximum temperature is set to 40 °C. You cannot modify the values.

You can input temperature values between 50°F (10°C) and 104°F (40°C). You can also use variables in the Edit window for the MINIMUM and MAXIMUM fields. Ensure that the **Minimum** value is less than the **Maximum** value.

TEMPCOMP/METHOD Command in the Edit Window

When you click **OK**, PC-DMIS inserts a `TEMPCOMP/METHOD` command into the measurement routine.

```
TEMPCOMP/METHOD = AUTOMATIC,MATERIAL = Aluminium,CTE=23
  ,SET WARNING LIMIT = TRUE,MINIMUM = 18,MAXIMUM = 22
  ,PART SENSOR NUM=PORTABLE ,X SCALE= 20,Y SCALE= 20,Z SCALE= 20,PART TEMP=20
```

Command Mode

Normally, a measurement routine only uses one `TEMPCOMP` command. The `TEMPCOMP` command should be placed near the top of the measurement routine prior to any measurements. When you execute the measurement routine, it executes according to the various input parameters.

Variable for Part Temperature in Temperature Compensation Command

You can use a variable to determine the temperature of a part.

You cannot use variables for scale temperatures.

If it is not feasible to attach a sensor to the part, you may want to use an external sensor to measure the part temperature. You can update the variable with a measured part temperature value and use the variable to execute the temperature compensation command. For example, this is useful when you place a part on the CMM, and the temperature of the part is measured through an external method.

To use a variable for the temperature of the part:

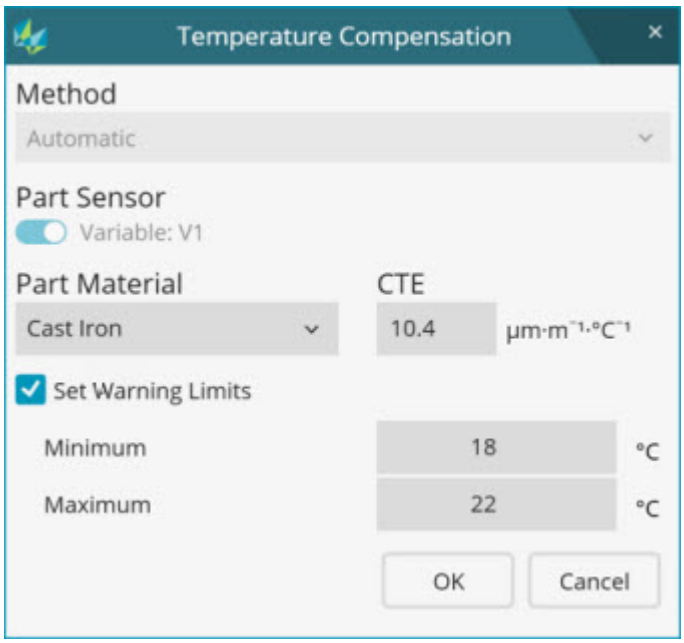
1. Create the variable in the measurement routine.
2. Update the `Part Temp` parameter in the Edit window with the name of the variable.

   In this code, V1 contains the temperature and is assigned to `Part Temp` :

   ```
   TEMPCOMP/CTE=10.4,Lo Threshold=18,Hi Threshold=22,PART SENSOR
   NUM=V1,X Axis Temp=,Y Axis Temp=,Z Axis Temp=,Part Temp=V1
   ```

3. Press F9 on the `TEMPCOMP` command.

   The sensor appears as a variable in the **Part Sensor** area in the **Temperature Compensation** dialog box. For example:



Temperature Compensation Dialog Box

Automatic Insertion of Temperature Compensation Command

For many types of machines, PC-DMIS automatically inserts the `TEMPCOMP` temperature compensation command when you create a measurement routine. This provides you with the opportunity to set up a measurement routine for temperature compensation. The **Temperature Compensation** dialog box appears, and you need to select the material of the part.

If you want PC-DMIS to automatically insert the temperature compensation command into a new measurement routine, select the **Auto Insert Temperature Compensation in New Routine** check box on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**).

If you do not want PC-DMIS to automatically insert the temperature compensation command into a new measurement routine, clear this check box.

PC-DMIS does not automatically insert the temperature compensation command into a new measurement routine for the following items:

- PC-DMIS NC software application
- Dual-arm CMMs
- Portable equipment
- Non-Hexagon controllers
- Machines with controllers that do not support temperature sensors

Temperature Compensation Dimensioning

The Temperature Compensation command can report the temperature of the scale and part that you are using for compensation. The Temperature Gage command dimensions the Temperature Compensation command when you place the Temperature Gage command immediately after the Temperature Compensation command. There should be no other command between the Temperature Compensation command and the Temperature Gage command.

For more information about the Temperature Gage command, see "Temperature Gage".

Thermal Expansion Origin Command

The default setting for the origin of the thermal expansion is the machine zero point. In some cases, you may want to set the origin at the location on the part. For example, you may want to set the origin when the part is clamped down on the center location and the part will not expand at that point.

To set the origin of the thermal expansion:

1. Select **Insert | Parameter Change | Thermal Expansion Origin**. The **Thermal Expansion Origin** dialog box appears.
2. Enter the **X**, **Y**, and **Z** values in the current coordinate system.
3. To insert the `THERMALEXPANSIONORIGIN` command into the Edit window, click **OK**. To cancel your changes and close the dialog box, click **Cancel**.

If you want to set the thermal expansion origin at a location other than the machine origin, then you must insert the thermal expansion origin command above the temperature compensation command or above the advanced temperature compensation command.

- For more information about the temperature compensation command, see "Temperature Compensation".
- For more information about the advanced temperature compensation command, see "Creating an Advanced Temperature Compensation Command".

Temperature Gage

You can create a Temperature Gage command that reads the temperature of the X scale, Y scale, Z scale, and part without starting temperature compensation.

Temperature Gage Command

The Temperature Gage command reads the temperature of the X scale, Y scale, Z scale, and part.

To create a Temperature Gage command:

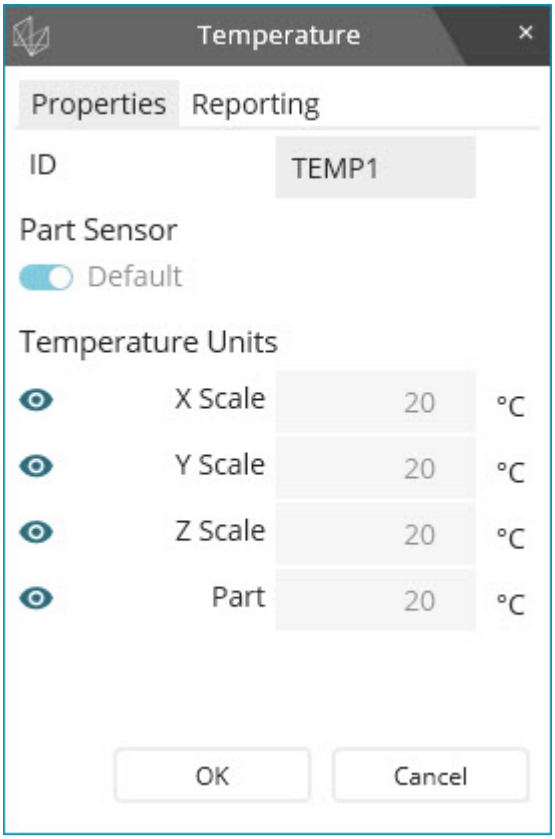
1. Select **Insert | Gage | Temperature** to open the **Temperature** dialog box.

> You can also access the **Temperature** dialog box from the **QuickMeasure** toolbar. Click the **Gage** drop-down arrow, and then click the **Temperature** button.

2. Complete the **Properties** tab and **Reporting** tab.

Properties Tab

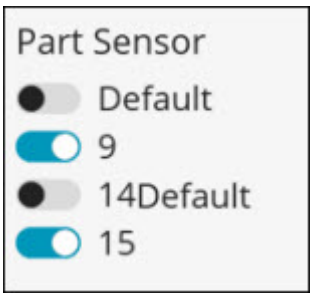
Use the **Properties** tab to set the ID, sensor numbers, and other parameters.



Temperature Dialog Box - Properties Tab

ID box - Type the ID of the Temperature Gage command.

Part Sensor option - Usually, a machine is equipped with only one part temperature sensor. This sensor appears as **Default**. You cannot cancel the selection of this sensor. If your machine is equipped with multiple part sensors, they appear in this area. For example:



Part Sensor Area

Select the sensor or sensors that are connected to the part being measured. Ensure that you select the correct sensor number. If you select multiple sensors, PC-DMIS uses the average of the measured values.

Temperature Units area - The **X Scale**, **Y Scale**, **Z Scale**, and **Part** boxes display the current temperatures as measured. You cannot change the values. The temperature units and the default sensor number are as defined in the **Edit | Preferences | Temperature Compensation Setup** menu option.

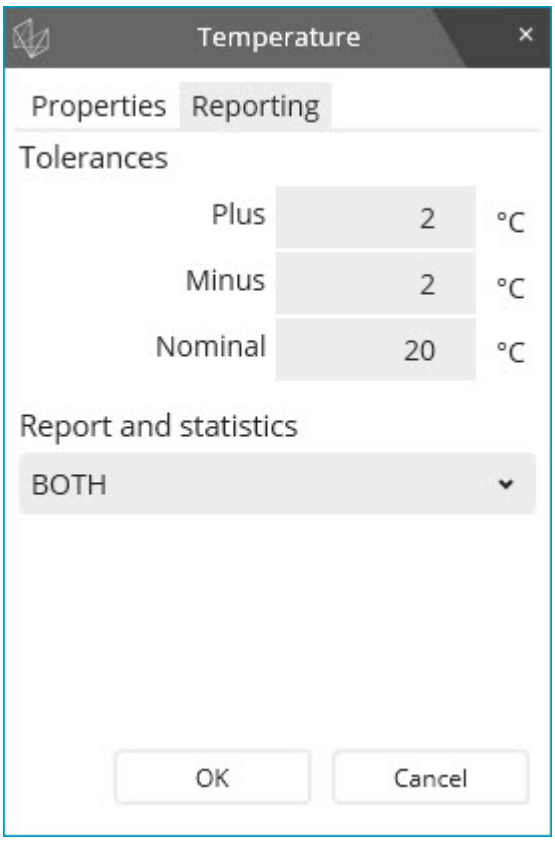
> **Show in Dimension/Hide in Dimension** - This button shows ( ) or hides ( ) the temperature units in the command. You need to show at least one temperature unit or axis.

OK button - Click this button to save your settings and insert the Temperature Gage command into the Edit window.

Cancel button - Click this button to close the dialog box without applying any changes.

Reporting Tab

Use the **Reporting** tab to set tolerances and the type of report output.



Temperature Dialog Box - Reporting Tab

Tolerances area:

- **Plus** box - Type the plus tolerance. You can change the plus tolerance for each axis and/or part.
- **Minus** box - Type the minus tolerance.
- **Nominal** box - Type the nominal value. The default nominal temperature is 20°C.

The same nominal and tolerance values apply to scale and part temperatures.

If necessary, you can change the nominal temperature and tolerances for each axes and/or part in the Edit window.

Report and Statistics area - In the list, choose how to send the output of the command when it is executed:

- **STATS** - Sends the output to a statistical file.
- **REPORT** - Sends the output to an inspection report.
- **BOTH** - Sends the output to both an inspection report and statistical files.
- **NONE** - Does not send the output anywhere.

Edit Window

The Edit window displays the command as a dimension.

It appears in Summary Mode as shown below:

Summary Mode

It appears in Command Mode as shown below:

```
DIM TEMP1= TEMPERATURE  OUTPUT=STATS,PART SENSOR NUM=DEFAULT,UNITS=C
AX       MEAS        DEV     NOMINAL       +TOL       -TOL     OUTTOL
X       19.890     -0.110      20.000      2.000      2.000      0.000 ----#----
Y       20.460      0.460      20.000      2.000      2.000      0.000 -----#---
Z       20.995      0.995      20.000      2.000      2.000      0.000 ------#--
M       21.870      1.870      20.000      2.000      2.000      0.000 --------#
END OF DIMENSION TEMP1
```

Command Mode

The command appears according to the format statement in your measurement routine.

If the controller is equipped with only one part temperature sensor, the `PART SENSOR NUM` field appears in the Edit window. If your controller is equipped with multiple part sensors, and you selected a part sensor number in the **Part Sensor** area in the **Temperature** dialog box, the number appears in the Edit window.

Expressions

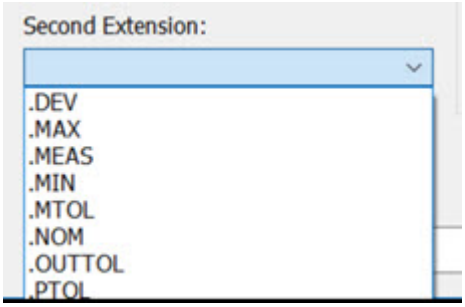
The Temperature Gage command supports expressions, such as:

```
ASSIGN/V1=READTEMP1.M.Meas
```

To create an expression:

1. Press F2 on the `ASSIGN` statement in the Edit window to open the **Expression Builder** dialog box.
2. From the **Expression Element Type** list, select "Temperature". All of the Temperature Gage commands that are in the measurement routine appear in the **ID** list.
3. From the **Extension** list, select the temperature of the X, Y, Z, or M (part). Select them as required to build an appropriate expression.

4. From the **Second Extension** list, select the type of value you want to store, such as deviation, maximum, measured, and so forth.



For details about expressions, see "Using Expressions and Variables". For details about the **Expression Builder** dialog box, see "Creating Expressions with the Expression Builder".

Temperature Compensation Dimensioning

The Temperature Compensation command can report the temperature of the scale and part that you are using for compensation. For details, see "Temperature Compensation Dimensioning".

Offline Mode and Non-Hexagon CMM

Temperatures cannot be read in Offline Mode. Therefore, in Offline mode, the measured value shows as 20 °C.

Similarly, the temperature shows as 20 °C if the controller of the connected machine does not support the ability to define and read temperature sensors.

Editing Part Materials and Coefficients

A material coefficient (also called a coefficient of thermal expansion, or CTE) is a number that describes how much a material expands or contracts as its temperature changes. Different materials — such as steel, aluminum, or titanium — expand at different rates. PC-DMIS uses these values during temperature compensation to adjust measurements and account for the fact that a part may be a different size at shop floor temperature than it would be at the standard reference temperature of 20 °C (68 °F).

PC-DMIS ships with coefficients for common materials, but you can use the editor discussed below to add materials or refine values for specific grades.

About the XML Files

Part materials and coefficients for temperature compensation are stored in a MaterialCoefficients.xml file. You can find this file in your system's hidden data path (where your computer stores program data files). When you modify materials or coefficients, PC-DMIS saves your changes to a separate file named usrMaterialCoefficients.xml in the same location. If this user file is present, PC-DMIS loads it instead of the default

MaterialCoefficients.xml. For more information about file locations, see "Understanding File Locations" in the "Setting Your Preferences" chapter.

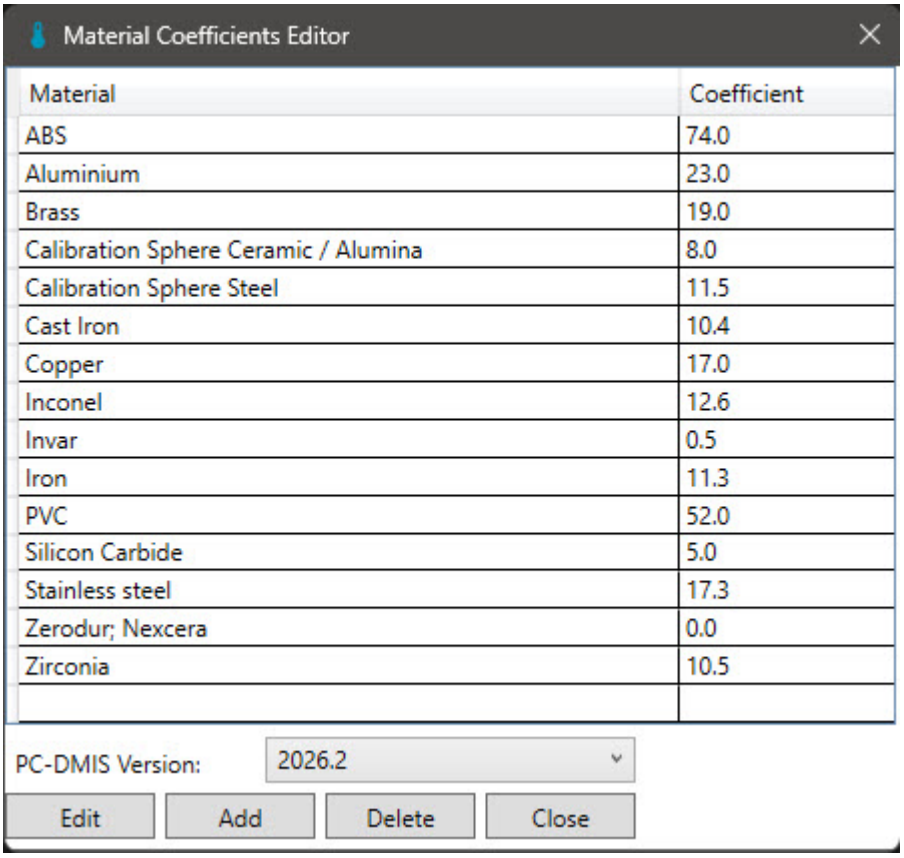
Starting with PC-DMIS 2026.2, your custom material coefficients (usrMaterialCoefficients.xml) are automatically migrated when you install a new version of PC-DMIS. You no longer need to manually redefine your custom materials after an upgrade.

About the Material Coefficients Editor

You can use the Material Coefficients Editor to add or modify coefficient values. When you do, PC-DMIS modifies the usrMaterialCoefficients.xml file. After you make any changes, you must reopen the **Temperature Compensation Setup** dialog box or **Advanced Temperature Compensation** dialog box to see any changes that you made to the file.

All coefficient values must include a decimal point. The Material Coefficients Editor displays a message about a wrong input if you don't include a decimal.

To use the Material Coefficients Editor, run the **MaterialCoefficientsEditor.exe** utility. This utility is located in the folder where you installed PC-DMIS.



Material Coefficients Editor

This editor lists the materials and their corresponding coefficients. It also provides buttons for you to edit the materials and values.

For the **Material Coefficients Editor** dialog box, the entered values are evaluated to **n X 10^{-6}**.

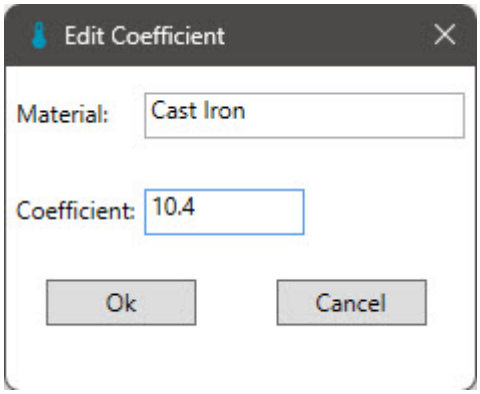
For example, iron has a coefficient of 11.3 in the example above. The software would evaluate this to 0.0000113.

For the **NewCoefficient** dialog box (see the **Edit** option below), you must enter temperature values in Celsius units.

This is very important when you edit an existing material coefficient value or add a new one.

PC-DMIS Version - This list defines the PC-DMIS version that is affected by the changes you make; it lists the versions of PC-DMIS that you have installed. Select a version from this list to pull in that version's materials and coefficients.

Edit - This button enables you to edit a selected material. It displays the **NewCoefficient** dialog box, where you can edit the material name or its coefficient value.

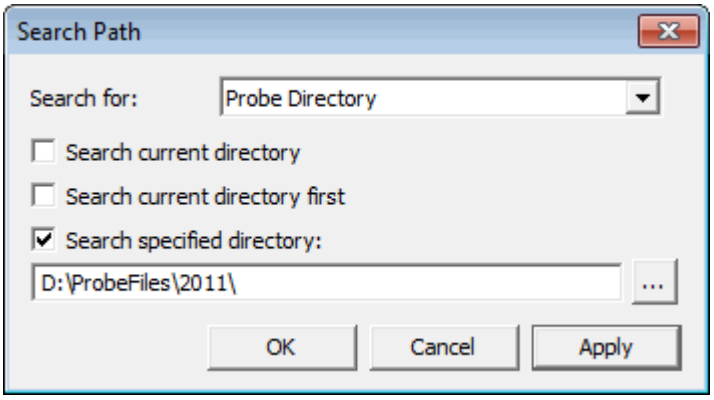


Add - This button enables you to add a new material and coefficient to the list. It displays the **Add Coefficient** dialog box, which is similar to the **NewCoefficient** dialog box discussed for the **Edit** button.

Delete - This button removes the selected row from the list of materials.

Close - This button closes the editor and displays a message that asks if you want to save your changes. To make the change to the usrMaterialCoefficients.xml file, click **Yes**. To close it without saving any changes, click **No**, or select the red X at the top right of the editor to close the editor.

Specifying Search Paths



Search Path Dialog Box

The **Edit | Preferences | Set Search Path** menu option displays the **Search Path** dialog box. You can use this dialog box to define the directories PC-DMIS uses when:

- Exporting CAD data or measurement routine data
- Importing CAD data or measurement routine data
- Loading and saving measurement routine files (.prg)
- Loading probe files (.prb)
- Recalling alignments (.aln)
- Calling subroutines

See the "Understanding File Locations" topic for information on where PC-DMIS stores specific settings and files.

Available Dialog Box Options

Search for - This list contains all the different items that you can associate directories with. The rest of the dialog box changes based on your selection. When PC-DMIS needs to find one of these file types or perform an action, it uses the directory associated with the selected item. These are the items in this list:

Default Export Directory - PC-DMIS exports CAD data or measurement routine data into the directory defined here.

Default Import Directory - PC-DMIS imports external CAD or measurement routine data into PC-DMIS from the directory defined here.

Default Measurement Routine Directory - PC-DMIS stores and loads measurement routine files from the directory defined here.

Probe Directory - PC-DMIS locates and stores probe files from the directory defined here.

Recall Directory - PC-DMIS recalls saved alignment files from the directory defined here.

Subroutine Directory - PC-DMIS loads measurement routine files and the subroutines stored within them from the directory tied to this item.

These check boxes may become available for selection based on what you choose above:

Search current directory - When marked, the software searches within the same directory where your current measurement routine resides.

Search current directory first - When marked and if both the **Search current directory** and the **Search specified directory** check boxes are selected, the software searches in both the current measurement routine's directory and the user specified directory. The order of the search depends on whether or not this check box is selected:

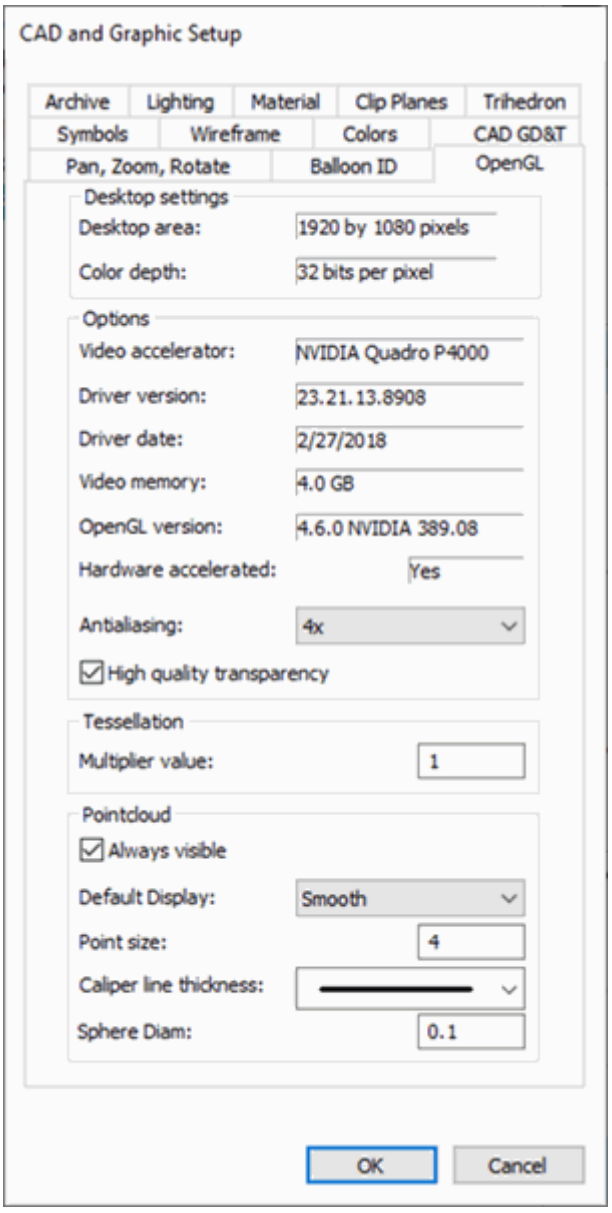
- If marked, PC-DMIS first accesses the same directory where your current measurement routine resides and then it accesses the user specified directory.
- If cleared, the search order is reversed, accessing first the user specified directory and then second the directory where your current measurement routine resides.

Search specified directory - When marked, this check box searches in a directory you specify. The pathway for the directory you specify appears in the box just below the check box. If no pathway is defined, you can type a full system pathway, or you can use the **...** button to choose a directory from your system's directory structure

To specify a new default directory to use:

1. Choose **Edit | Preferences | Set Search Path** to access the **Search Path** dialog box.
2. From the **Search for** list choose an item for which you want to define a pathway.
3. Mark any check boxes as needed.
4. Type the directory pathway in the box (or use the **...** button to browse to and select a directory).
5. Click **Apply**.
6. As needed, repeat the above steps to set additional search paths for other items.
7. Click **OK**.

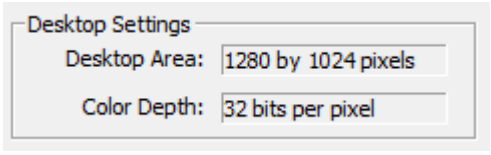
Changing OpenGL Options



CAD and Graphic Setup Dialog Box - OpenGL Tab

The **Edit | Graphic Display Window | OpenGL** menu option displays the **OpenGL** tab of the **CAD and Graphic Setup** dialog box. This allows you to change the OpenGL options that affect the display of the model in solid view mode. To change the part to solid view, see "Setting Up the Screen View" in the "Editing the CAD Display" chapter.

Desktop Settings area

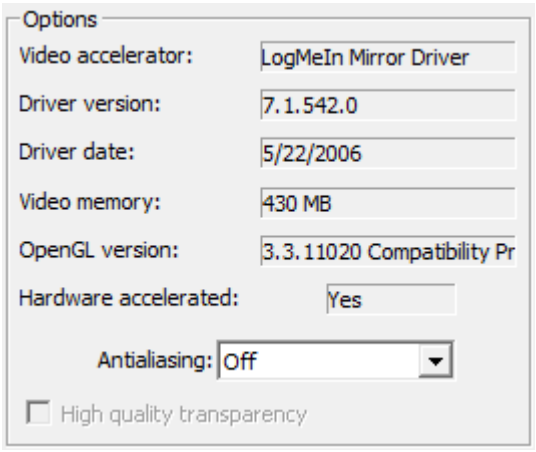


There can be different OpenGL options for each desktop display setting. The **Desktop Settings** area shows what the current desktop settings are.

Resolution Ratios of Different Monitor Sizes

Wide screen monitors need a 1.6 ratio instead of a 1.3333 ratio used by normal monitors. For example, a resolution of 1200x1600 has a 1.3333 ratio (1600/1200) and works well for a normal-size monitor, while a resolution of 1680x1050 has a 1.6 ratio, good for a wide screen monitor. If you use a wide screen monitor and your screen appears stretched (perhaps your circle features appear as ellipses in the Graphic Display window), use a 1.6 resolution ratio to resolve this problem.

Options area



The **Options** area displays information about the system's video card:

- **Video Accelerator** - Video card description
- **Driver version** - Video driver version
- **Driver date** - Video driver date of release
- **Video memory** - Amount of video card RAM
- **OpenGL version** - Version of OpenGL supported by the video driver
- **Hardware accelerated** - Displays Yes or No depending on whether graphics rendering is hardware accelerated. Hardware acceleration is much faster than software acceleration.

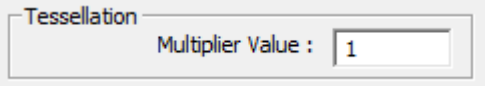
Antialiasing - The **Antialiasing** list allows the level of antialiasing by specifying the number of multisamples. 2x antialiasing samples each pixel twice. 4x antialiasing samples each pixel four times, and so on. When antialiasing is enabled, each pixel is sampled multiple times at slightly different locations within the pixel. An average color is computed from these samples to determine the final pixel color. This effectively reduces the model's jagged edges in the Graphic Display window. Higher antialiasing settings produce better visual results at the expense of slower system performance.

The capabilities of your video card determine the antialiasing options. Some graphics accelerators may support 64x antialiasing while others may only support up to 16x, or they may not support antialiasing at all. If your graphics accelerator supports antialiasing, the

amount of RAM it possesses and the resolution of your screen determine the default value (with a maximum default value of 4x). If your video card does not support antialiasing, PC-DMIS sets the default to **Off**.

The **High quality transparency** check box controls the `HighQualityTransparency` entry. By default, PC-DMIS clears this check box. It only functions if your graphics driver supports OpenGL 4.2 and your video adapter has at least 1 GB of memory.

Tessellation area



The **Tessellation** area controls the drawn image by setting a tessellation multiplier in the **Multiplier Value** box. PC-DMIS multiplies the **Multiplier Value** by the tessellation value for the given CAD system. These values are then used in the generation of the shaded image.

The tessellation value is the default value used to break up surfaces into patches for shading.

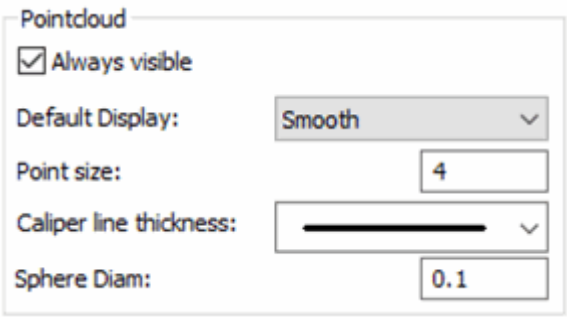
The tessellation multiplier updates the Graphic Display window immediately once you click outside of the **Multiplier Value** box or press Tab to move to a different item on the dialog box.

Modifying this tessellation value affects perimeter scans because PC-DMIS computes the distance around a curved surface edge by adding the segment lengths of a polyline that represents that edge. The tessellation multiplier changes the length of each polyline segment (lower tolerances result in smaller segments). While the actual perimeter points lie exactly on the edge curve, different tessellation tolerances result in small differences in where each point is located along the edge curve.

The size of the CAD data file and the tessellation multiplier value used affects the amount of memory needed. These both affect the number of tessellated facets needed to display the model. The smaller the tessellation multiplier value used, the more memory needed for the facets. For large CAD models, this could cause an "Out Of Memory" error. If this occurs, the current PC-DMIS session is left in an unstable state and should be terminated.

The default tessellation multiplier value is 1.0. Setting a tessellation multiplier of 0.1 results in a 10 to 20 percent increase in the memory required over the default value of 1.0. Decreasing the tessellation multiplier further to 0.01 results in an additional 50 to 65 percent increase of memory required.

Pointcloud area



The **Pointcloud** section determines how PC-DMIS draws a pointcloud (COP) in the Graphic Display window. A COP typically is generated from laser probes that can quickly collect large numbers of points. For more information on pointclouds, see the "PC-DMIS Laser" documentation.

Always visible - This setting applies to Pointcloud Point Colormaps when the Dots, Needles, and/or Text options are enabled.

- When you enable this option, the Dots, Needles, and/or Text for the Point Colormap are visible even if they are lower than the CAD model (negative relative to the CAD face).
- When this option is not enabled, PC-DMIS only displays the Point Colormap points and text if they appear in the current view.

Default Display - Sets the default graphical representation of the pointcloud in the Graphic Display window.

The valid options are:

- Smooth (default)
- Flat
- Two sided

- Normals

The software uses the display setting each time you create a pointcloud feature. For details, see "Pointcloud Graphical Representation" in the PC-DMIS Laser documentation.

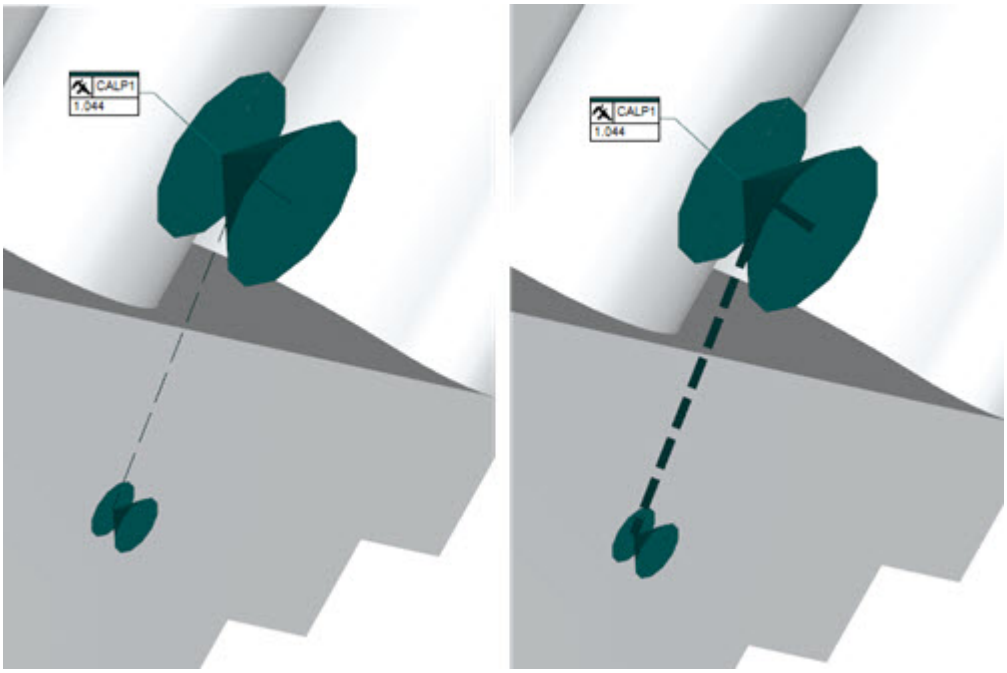
Point size - Specifies the pixel size of the points in a pointcloud.

Caliper line thickness - Specifies the thickness of the Caliper gage line, and the connecting line for thickness colormap annotations when the opposing sides are shown. The line thickness options are shown below with the default option selected:

Caliper line thickness options

The line thickness options correspond to (from thinnest to thickest) 1, 3, 5, and 7 pixels.

For details on the Caliper feature, see the "Caliper Overview" topic in the PC-DMIS Laser documentation.

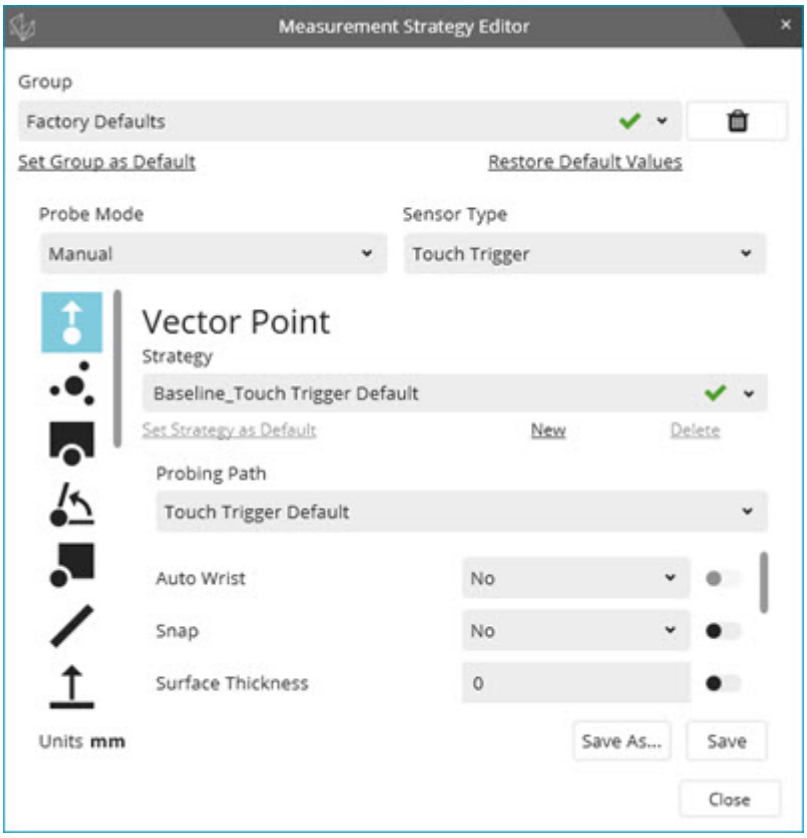


Examples that show the Caliper line thickness option set to minimum (left) and maximum (right).

Sphere Diam - Specifies the size of the thickness annotation points when you select the **Show opposing annotation points** check box on the **Pointcloud Operator** or **Mesh Operator** dialog box when you select the Pointcloud or Mesh Thickness Colormap operator. For details, see "Mesh Thickness Colormap" in the PC-DMIS Laser documentation.

Using the Measurement Strategy Editor (MSE)

You can use the Measurement Strategy Editor (MSE) to modify the settings for all the Auto features. You can store the modified settings as strategies and as custom groups.



Measurement Strategy Editor Dialog Box

To access the MSE, select **Edit | Preferences | Measurement Strategy Editor**. By default, PC-DMIS shows the MSE for CMM configurations. For portable configurations, it hides the MSE.

If the MSE is not available, you can enable it with the **Use Measurement Strategy Editor** check box from the **Setup Options** dialog box, **General** tab. For more information, see "Use Measurement Strategy Editor" in this chapter.

Related Topics

For information on the different settings in the MSE, see "The Auto Feature Dialog Box" in the "Creating Auto Features" chapter and "Using the Probe Toolbox" in the "PC-DMIS CMM" documentation.

For information on the measurement strategy widget, see "Using the Measurement Strategy Widget" in the "Creating Auto Features" chapter.

For information on measurement strategies in general, see "Working with Measurement Strategies" in the "PC-DMIS CMM" documentation.

Description of the MSE

Group - This shows an alphabetized list of all your saved strategies along with the original settings. When you first use the Measurement Strategy Editor, this list only shows **Factory Defaults** because you haven't yet saved any new groups. This list grows as you use **Save As** to create more groups. Each group in the **Group** list stores its settings in a .msexml file of the same name. The software stores them in the C:\ProgramData\Hexagon\PC-DMIS\<version> folder, where <version> is your version of PC-DMIS.

Delete Group ( ) - This button deletes the selected group from the database and selects the top-most item in the **Group** list. This also deletes the associated .msexml file. You cannot delete the **Factory Defaults** group. A green check mark indicates when the operation finishes.

Set Group as Default - Sets the selected group as the default group when you next use the Measurement Strategy Editor. A green check mark appears in the **Group** list to indicate the default group. This also sets the default group in the **Measurement Strategy** toolbar. For more information see, "Measurement Strategy Toolbar" in the "Using Toolbars" chapter.

Restore Default Values - Restores the values for the selected group to their stored defaults. If you add a feature to the Edit window and press F9 on that feature and make changes to it, the software writes those changes to the JSON file. This means the software uses those changes as the new default for that feature type and not what's in the selected group. The **Restore Default Values** button restores your default settings on the feature to match the settings stored within the selected group.

Probe Mode - For a traditional CMM, this list shows **Manual** and **DCC** (Direct Computer Control). With this list, you can save settings according to machine type, either a manual CMM or a CMM capable of DCC motion.

Sensor Type - Determines the type of sensor. The application only shows settings that the selected sensor type can use. For a traditional CMM, this shows **Touch Trigger** and **Analog**. A touch-trigger sensor or probe records hits based on discrete hits. An analog sensor or probe remains in contact with the part and records hits based on time and distance as it scans.

Strategy - Sets the strategy to modify. This list initially shows a default strategy for the feature and sensor type. A green check mark appears in the **Strategy** list to indicate the default strategy. You can select a strategy to modify it, or you can click **New** to create a custom strategy. When you save the settings to a group, PC-DMIS also saves the current settings for the feature to the current selected strategy.

Strategy Type - This list does not appear at first. It only appears if, under **Sensor Type**, you choose **Analog** and then you select a supported feature (High Point, Circle, Plane, Line, Cylinder, or Cone). It allows you to choose an internal strategy. These strategies provide predefined ways to measure that feature, and they provide additional parameters that you can modify. You can select one of these internal strategies and use it as a base

for your own custom strategy. These internal strategies may also hide parameters that don't make sense in that context (such as **Number of Hits** for a continuous contact probe).

Features Toolbar - The toolbar down the left side contains all the Auto features. When you select an Auto feature, its settings appear in the window.

Settings - This topic does not cover the individual settings for each Auto feature. You can find those in the Auto Feature documentation. Many Auto feature settings have a switch to the right ( ). You can turn a switch on to have that setting appear in the Measurement Strategy Widget. You can turn a switch off to hide that setting from the widget.

Unit - If this shows **mm**, the application displays values in millimeters. If it shows **in**, the application displays values in inches. The server always stores its settings in millimeters.

Save - This button saves all the defaults settings for all the features in the current group to the PC-DMIS JSON files. This also saves the settings for each feature to each feature's current selected strategy. You cannot overwrite the original factory defaults. If **Group** shows **Factory Defaults**, a **Group Name** input box appears so you can save the modifications as a new group of settings.

Save As - A **Group Name** input box always appears so you can save the group as a new set.

Close - Closes the MSE application. If you haven't saved your changes, PC-DMIS asks if you want to save them.

How to Use the MSE

1. Select **Edit | Preferences | Measurement Strategy Editor** to access the Measurement Strategy Editor application. You can position the application and resize the window height as needed.
2. Choose the desired **Probe Mode** and the **Sensor Type**.
3. From the toolbar on the left, you can scroll up and down and then select an Auto feature to modify its settings.
4. Modify a feature's default settings for that group. (You can also define strategies. For information, see "Creating or Modifying Strategies" below.) By default, PC-DMIS decides what features to send to the Measurement Strategy Widget but you can override that with the switches to the right of each setting ( ).
5. Once you're done modifying one feature, if you like, click to another feature. The application temporarily stores your modifications as you click between the different features.
6. Continue to modify the settings as needed.

7. Save your changes. This also inserts your settings as defaults into PC-DMIS.
    - To save the current settings for the feature to the current selected group of settings, click **Save**. If you attempt to save the settings in **Factory Defaults**, the software opens the **Group Name** input box. You can type a name to save your group of settings under a different group name.
    - To save your changes to a new group of settings, click **Save As**. The software opens the **Group Name** input box. Type a name to save your group of settings under the new group name.
8. Save as many groups of settings as you want.
9. From the **Group** list, choose the group of settings you intend to use most frequently, and click **Set Group as Default**. This tells PC-DMIS to use those settings as the new default settings.
10. Click **Close** to close the MSE.
11. Create your Auto features.

The Unit option displays the units (inches or millimetres) that are being applied. This is the same as the measurement routine’s units setting.

How the MSE Works

You can use the MSE to modify the settings for all Auto feature and then store them as strategies and custom groups. Strategies are specific to each feature. Groups contain any modified settings for all the features.

The MSE saves the settings for each custom group in text files. These text files use the XML format. Each text file has the group name as its name with an .msexml filename extension. Whenever you delete a group, PC-DMIS deletes the corresponding .msexml file.

PC-DMIS stores these files in the C:\ProgramData\Hexagon\PC-DMIS\2026.2 folder.

When you set a group of settings as the default (and at other times discussed in "About Default Settings" below), the MSE writes those settings into the JSON file for PC-DMIS Auto features to use.

When you create Auto features, PC-DMIS takes into account the current probe mode (DCC or Manual) and the current sensor type (Touch-trigger or Analog). It then uses the appropriate settings for that mode or type. For example, a circle feature that you insert after a DCC Mode command can have a different number of hits than one that you insert after a Manual Mode command.

Touch-trigger and Analog probes only work in DCC Mode. Scan strategies only work in DCC Mode.

The MSE does not yet manage laser and vision settings.

About the Default Settings

You can use the MSE to modify the default settings (number of hits, depth, void detection, strategy types, and so on) for all Auto feature types. By default, PC-DMIS writes these changes to the JSON file whenever you modify and then save settings in the Measurement Strategy Editor.

If your default settings don't update, mark the **Use Measurement Strategy Widget** check box from the **General** tab of the **Setup Options** dialog box (F5). The measurement strategy widget applies modifications into the JSON file. If that check box is cleared, the MSE stores your changes only in the .msexml files. For information, see "Using the Measurement Strategy Widget" in the "Creating Auto Features" chapter.

PC-DMIS queries the JSON file and uses those settings when you create an Auto feature.

You can also update the JSON file for a specific setting if you modify that setting in the **Auto Feature** dialog box or in the Edit window command.

PC-DMIS updates the JSON file with the settings stored in the default MSE group whenever you do one of the following:

- Save a change to the default group of settings with the MSE.
- Change to a new probe mode (such as from Manual to DCC) in the Edit window.
- Change to a new probe type in the Edit window.

Creating or Modifying Strategies

Each feature in the MSE has one or more internal strategies that come with PC-DMIS. You can modify settings for these strategies or you can create a custom strategy.

1. Access the MSE.
2. From the MSE, under **Strategy**, choose the strategy to modify. If you want to create a new strategy for the feature, click **New**, and type the strategy name.
3. Make modifications as usual to the feature's settings.
4. When you click **Save** or **Save As** for the group, PC-DMIS also saves the current settings for the feature to the current selected strategy.

5. Create as many strategies as you want.
6. From the **Strategy** list, choose the strategy you intend to use most frequently, and click **Set Strategy as Default**.
7. Create your Auto features.

You can select the strategy to use when you create or edit features with the Measurement Strategy Widget.

How to Attach a Scan Adaptive Strategy to a Supported Feature

1. Access the MSE.
2. From the **Probe Mode** list, choose **DCC**.
3. From the **Sensor Type** list, choose **Analog**.
4. From the toolbar on the left, select the feature to modify (High Point, Circle, Plane, Line, Cylinder, or Cone).
5. From the **Strategy Type** list, select the desired internal measurement strategy.
6. Beneath the **Strategy Type** list, click **Set Strategy as Default**. Only the strategy that you set as default is assigned to that feature. If you don't set the strategy as the default, the software uses TTP_STRATEGY.
7. Click **Save** to update the .msexml file and to use the defined settings for that feature.

Working with Smart Parameters

Terminology Note: In this topic, the word "parameter" also means "setting".

Typically, in the MSE, you can only provide a single value for a setting. In some cases, however, you may need to provide more than a single value, based on different conditions. This is best explained with an example:

Suppose you need to measure circle features on a part that have different sizes, and you want the number of hits to adjust based on the size of the diameter. With smart parameters, you can do this.

By default, a circle feature has these smart parameters defined for its **Hits** setting:

Maximum Diameter	Number of Hits
6	4
15	6
25	10
50	18

If you create a circle feature with smart parameters enabled, PC-DMIS checks the diameter and uses the numbers of hits for the row up to the diameter's maximum value. So, with the above grid of values, if you create a circle feature with a diameter of 12 units, then PC-DMIS uses six hits to create that circle. With a diameter of 20 units, PC-DMIS gives the circle 10 hits.

Now, suppose you want to have a different number of hits (maybe 20 hits) for circles with a diameter within the range of 51 to 75. You can add a new row and then for that row, set the **Maximum Diameter** to 75 and **Number of Hits** to 20.

If the measured circle feature is larger than the highest diameter in the grid, PC-DMIS doesn't use any smart parameter for the number of hits; instead, it uses the default number of hits (usually seven) from the **Auto Feature** dialog box.

Available Smart Parameters

These are the available smart parameters where you can provide multiple values:

Feature	Strategy	Smart Parameters
Circle	Baseline_ Touch Trigger Default	Hits
Circle	Baseline_ Adaptive Circle Scan	Point Density Scan Speed Acceleration
Cylinder	Baseline_ Touch Trigger Default	Hits per Level Ending Offset
Cylinder	Baseline_ Adaptive Cylinder Line Scan	Ending Offset
Cylinder	Baseline_ Adaptive Cylinder Concentric Circle Scan	Point Density Scan Speed Acceleration Ending Offset

Plane	Baseline_Adaptive Free Form Plane Scan Baseline_Touch Trigger Free Form Plane	Inner Offset Outer Offset Perimeter Jump Hole You can access these Smart Parameter options from the **Measurement Strategy Editor** dialog box and the **Measurement Strategy Widget**. For details, see the "Using Smart Parameters from the Auto Feature dialog box, the Measurement Strategy Editor, and the Measurement Strategy Widget" section in the "Using Smart Parameters on the Measurement Strategy Widget" topic of the PC-DMIS Core documentation.

Enabling and Editing Smart Parameters

To enable smart parameters:

1. Locate the setting in the MSE. If the setting supports smart parameters, a grid button appears next to it. If smart parameters is disabled, it has a gray crossed out grid button next to it.

2. Click the gray, crossed-out grid button ( ) to the right of the parameter. The button turns to a green grid button ( ) to indicate it's enabled.

To edit smart parameters:

1. Click the **Edit Smart Parameter** button ( ) to the right of the setting. A smart parameters dialog box opens with a table of values.
2. Use **Add Row** ( ) and **Remove Row** ( ) buttons to define the number of rows in the grid. You can have up to seven rows.
3. Define the values in the grid.
4. Click **Apply** ( ) to accept your changes. You can also click the **Cancel** button ( ) to not save your changes.
5. When you enable smart parameters, during feature creation, PC-DMIS chooses the number of hits based on the diameter.
6. When you create the feature, if the diameter size is less than or equal to the set maximum diameter value, PC-DMIS uses those numbers of hits.

Using Smart Parameters on the Measurement Strategy Widget

You can also enable or disable smart parameters from the measurement strategy widget. For more information about the widget, see "Using the Measurement Strategy Widget".

When the measurement strategy widget appears, click the **Properties** button ( ) to see if that feature type uses smart parameters.

- If the button is gray, smart parameters are currently disabled. You can click this button to enable them.
- If the button is green, smart parameters are currently enabled. You can click this button to disable them.

The value of the setting to the left of the grid button shows the value PC-DMIS chooses for the feature. You can click the grid button to switch between the smart parameter value and the default value for the feature you're creating.

If you want to define a custom value, turn off smart parameters and type the value.

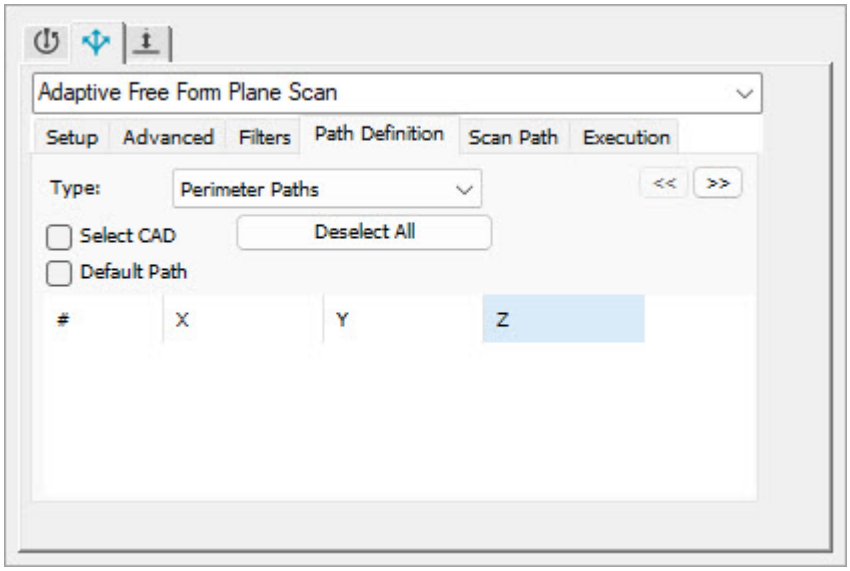
Using Smart Parameters from the Auto Feature dialog box, the Measurement Strategy Editor, and the Measurement Strategy Widget

Using Smart Parameters from the Auto Features Dialog Box

To use Smart Parameters on an Auto Feature Plane using the Free Form Plane Strategy, follow these steps:

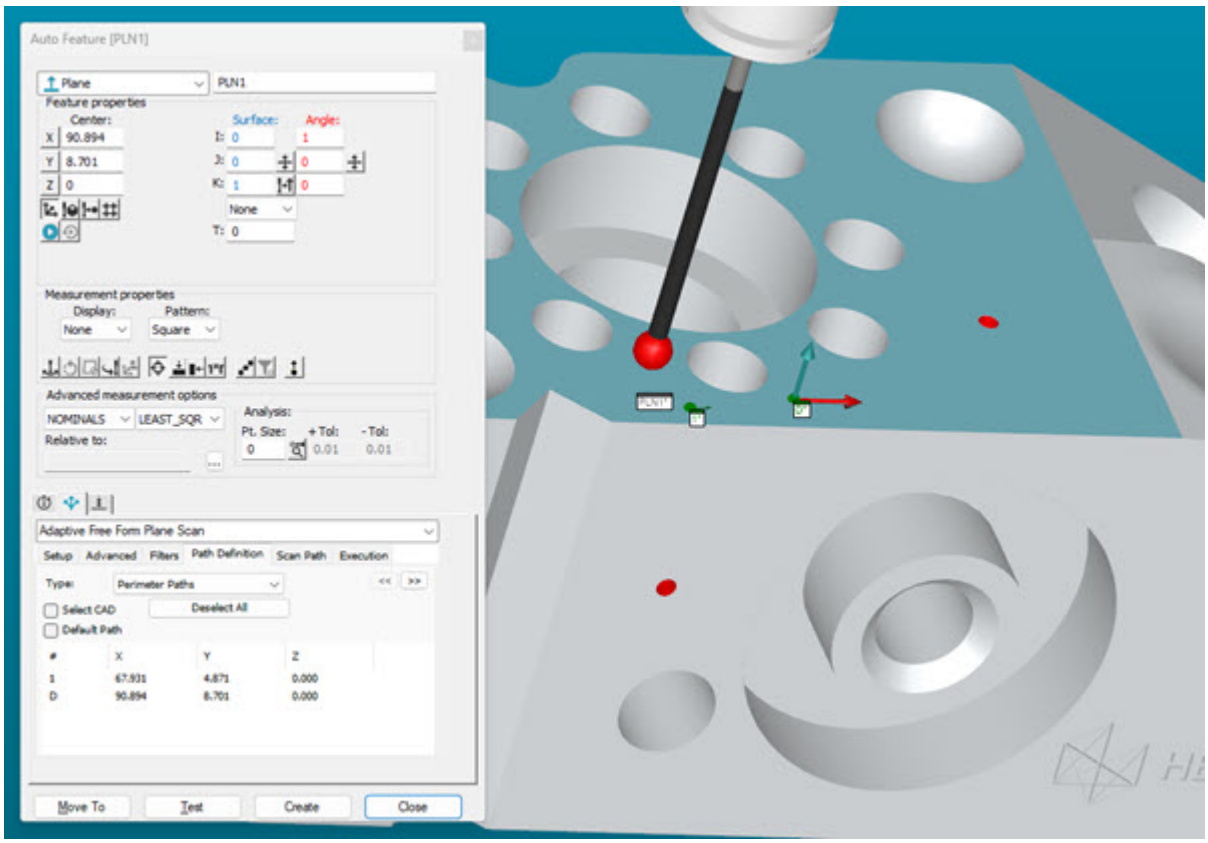
1. Select the Auto Feature Plane option (**Insert | Feature | Auto | Plane**) or click the **Auto Plane** button from the **Auto Feature** or **QuickFeature** toolbar (**View | Toolbars**).

2. From the **Measurement Strategy** tab at the bottom of the **Auto Feature** dialog box, select the **Path Definition** tab.



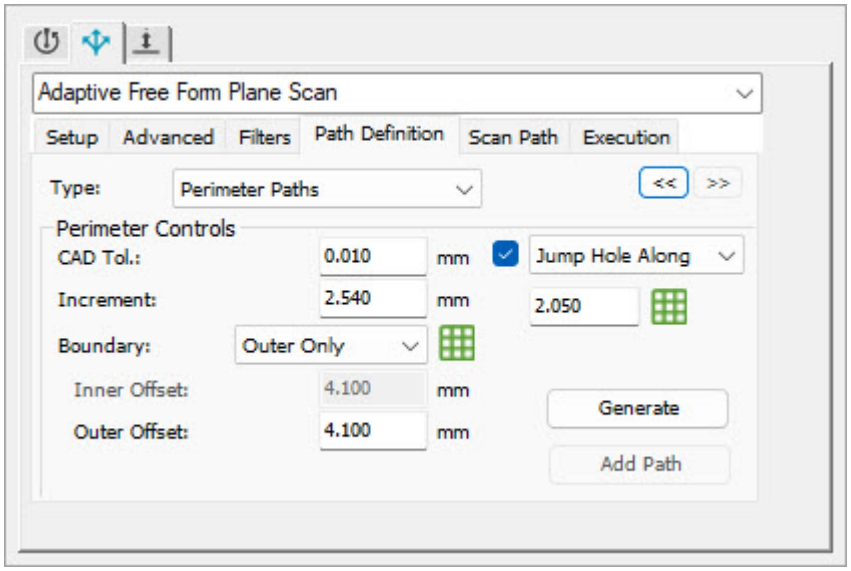
Probe Toolbox - Path Definition Tab

3. Select the **Adaptive Free Form Plane Scan** option from the **Measurement Strategy** list and then select the first point (**1**) and the direction point (**D**) on the CAD model in the Graphic Display window.



Example showing the Auto Feature dialog box for a Plane feature using the Adaptive Free Form Plane Scan strategy.

4. Click the double-arrow button **>>** to show the next set of **Path Definition** options.

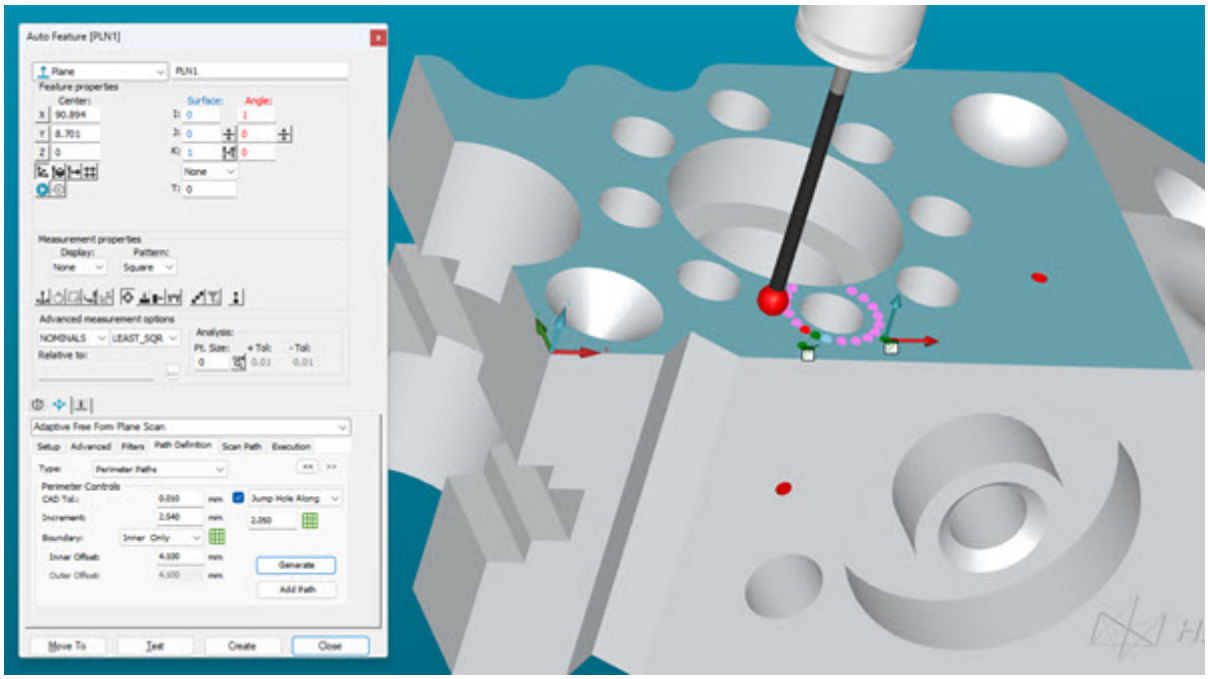


This example uses an 8 mm diameter probe tip. You can see in this example that, with the Smart Parameters turned on , the **Inner Offset** and **Outer Offset** values are set to one half of the probe tip's diameter plus 0.1 mm, or 4.100 mm. Also, the **Jump Hole Along** value is set to half the **Inner Offset** and **Outer Offset** values, or 2.050 mm in this example. You can set the values manually by typing a new value

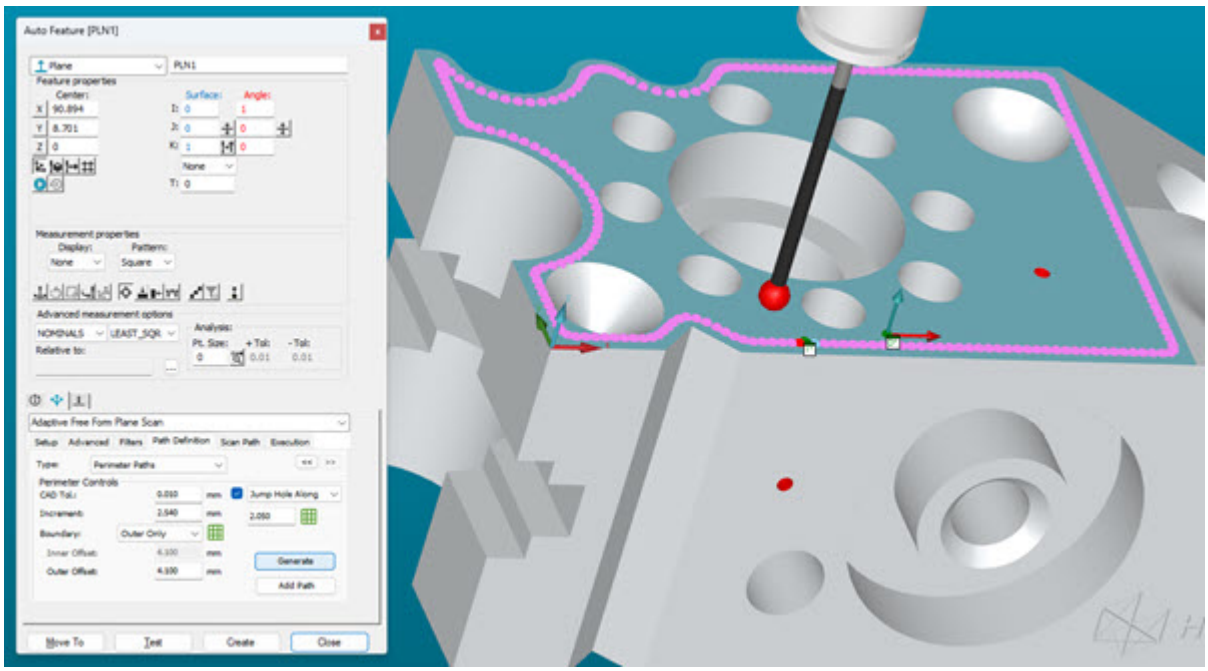
directly into any of the boxes. As soon as you start to type a new value into a box that has Smart Parameters turned on, the Smart Parameter option turns off .

The **Boundary** option can be set to **Inner Only**, **Outer Only**, or **Inner and Outer**. You can see examples of the resulting path after you click the **Generate** button for each of the **Boundary** options below.

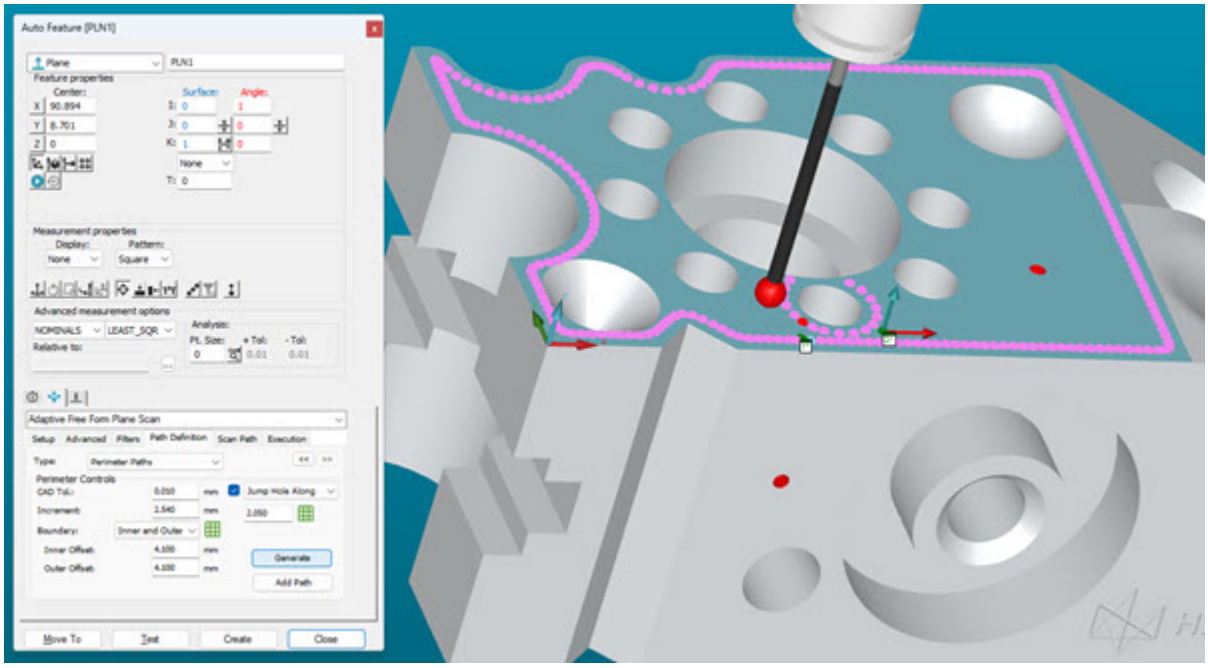
Smart Parameters functionality is initially off by default but retains the state it was in the last time your measurement routine was closed.



Example showing the Boundary option set to Inner Only.



Example showing the Boundary option set to Outer Only.



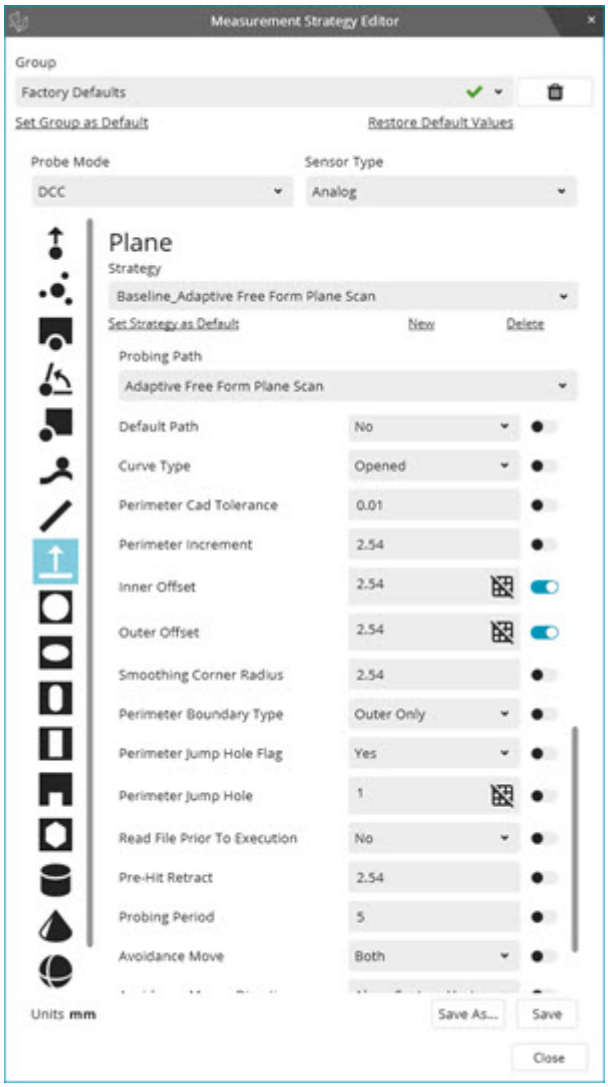
Example showing the Boundary option set to Inner and Outer.

Using Smart Parameters from the MSE dialog box and the Measurement Strategy Widget

The Smart Parameter functionality is identical to how it works from the **Auto Feature** dialog box described above.

To use Smart Parameters from the **Measurement Strategy Editor** dialog box, follow these steps:

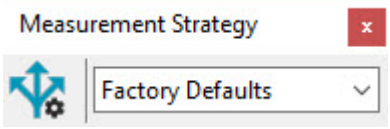
1. From the **Measurement Strategy** toolbar (**View | Toolbars**), click the **Measurement Strategy Editor** button to open the **Measurement Strategy Editor** dialog box.
2. From the list of feature types on the left, click the **Plane** icon.
3. Click each toggle option to turn it on. Each option you turn on will show up in the **Measurement Strategy Widget** for the defined **Group**.
4. From the **Strategy** list, select the **Baseline_Adaptive Free Form Plane Scan** or the **Baseline_Touch Trigger Free Form Plane** strategy.



Measurement Strategy Editor dialog box showing the Smart Parameter options for a Plane feature.

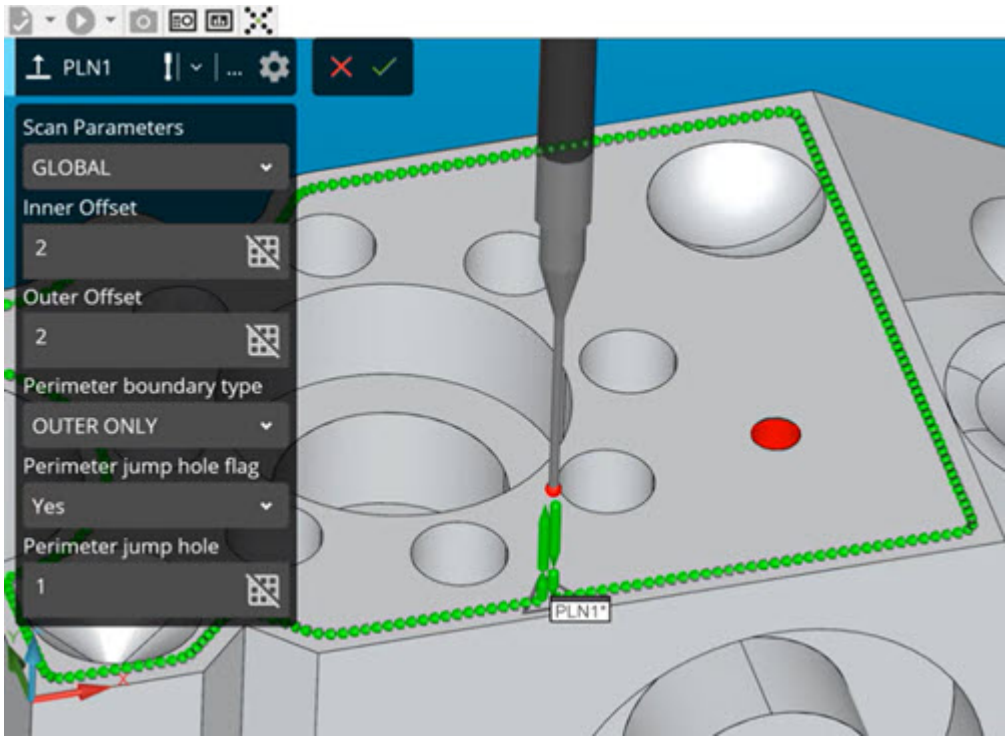
5. Click the **Save As** button and type a unique **Group** name for the settings, and then click the **Close** button.

To use Smart Parameters from the **Measurement Strategy Widget**, follow these steps:

1. Select the Measurement Strategy **Group** from the **Measurement Strategy Editor** toolbar 

(**View | Toolbars**).

2. Press and hold the Shift button and click a surface to create a QuickFeature plane. PC-DMIS shows the **Measurement Strategy Widget** automatically when you create a QuickFeature.



QuickFeature Plane feature showing the Measurement Strategy Widget with Smart Parameter options.

You can see from the status of the **Smart Parameter** button in the example above that the functionality is off.

3. Click the **Smart Parameter** button to turn the functionality on. The **Smart Parameter** buttons for the **Inner Offset** and **Outer Offset** settings work together. If you click one on or off, PC-DMIS turns the Smart Parameters functionality on or off for both.

4. Select the **Perimeter boundary type** option from the list. The options are **INNER ONLY**, **OUTER ONLY**, and **INNER AND OUTER**.

5. For the **Perimeter Jump hole** option, click the **Smart Parameters** button to turn the Smart Parameters functionality on or off.

6. Click the **Apply** button to create the feature.

Setting Up Sensor Mapping

You can use the **Edit | Preferences | Feature - Sensor Mapping menu** item to open the **Sensor Mapping** dialog box.

With the **Sensor Mapping** dialog box open, you can then use QuickFeatures to populate this dialog box and then map those features to specific sensors (probes).

Once you click **Done**, the mapped features are added into the measurement routine with the appropriate `LOADPROBE` commands.

- For more information on QuickFeatures, see "Creating QuickFeatures" in the "Creating Auto Features" chapter.
- For more information on sensor mapping, see "Mapping Features to Sensors" in the "Using Inspection Plans in PC-DMIS" appendix.

Defining Colors on Imported CAD Entities

To define the default colors for imported CAD entities (points, curves, surfaces, and datums), use the **Import Options** dialog box, described in "Setting Import Options".

Understanding File Locations

The default file locations (path locations) for various user-specific settings and other files have changed in PC-DMIS 2010 MR2 and later to comply with newer Windows operating system standards. Before PC-DMIS 2010 MR2, most of this information was stored inside of the PC-DMIS installation directory. This topic shows where these files are now stored based on each file's role.

In the descriptions of the file locations, *<user name>* is the user name that you use to log on to Windows.

You can use the PC-DMIS Settings Editor application to back up, restore, or clear user-specific files and settings. Rather than manually delete or restore individual files on your own, we recommend that you use the tools that the PC-DMIS Settings Editor provides. For more information, refer to the PC-DMIS Settings Editor documentation.

For specific information about data files, see "Understanding Data Files".

Public Documents

These files include measurement routine (.prg), CAD, probe (.prb), and other files.

File location: C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2

Public Documents (Reporting Directory)

These files include report template, label template, rule definition, and color definition files.

File location: C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2\Reporting

Public Documents (Demo CAD Files)

File location: C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2\CAD

Public Documents (Fixturing Files)

File location: C:\Users\Public\Public Documents\Hexagon\PC-DMIS\FiveUnique

Program Data Files [Hidden]

Hidden program data files include calibration, setup, and other files.

- abcalib.dat
- abcomp.dat
- abcomps.dat
- aboutput.dat
- arm1arm2.dat
- armarm.dat
- ccalib.dat
- ccomp.dat
- ccomps.dat
- changed_probe1.dat
- changed_probe2.dat
- changed_results1.dat
- changed_results2.dat
- ChangeManagerLog?.csv
- cme_existing.xml
- cme_new.xml
- comp.dat
- compens.dat

- ConstructionVersion?.txt
- coutput.dat
- debug.txt
- DMISMODULES.DAT
- dph9.dat
- export.IP
- flextool.dat
- ILLUMINATION.DAT
- localfixturedefs.dat
- MaterialCoefficients.xml
- PROBECHANGER.DAT
- RCMATRIX.dat
- RCSPHERE.dat
- rotab.dat
- rotab2.dat
- rotindx.dat
- rotindx2.dat
- startuplog.txt
- tlw_MapResultInfo.txt
- tool.dat
- toolc.dat
- toolcs.dat
- unitcal.dat
- usermachine.dat
- usermodule.dat
- userquickfix.dat
- usrMaterialCoefficients.xml

- usrprobe.dat
- wristm.dat
- wrists.dat
- XSTATS11.tmp

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

User Data Files

These files include files for toolbars and menu layouts, lighting and materials settings, and other files:

- accelerator.dat
- CustomMenuItem.dat
- colors.dat
- executebarstate.dat
- gbarstate.dat
- layouttoolbar.dat
- layout1.dat
- layout2.dat
- layout3.dat
- layout4.dat
- layout5.dat
- layout6.dat
- layout7.dat
- layout8.dat
- layout9.dat
- layout10.dat
- menu_chn.dat
- menu_cze.dat
- menu_dch.dat
- menu_eng.dat

- menu_frn.dat
- menu_grm.dat
- menu_ita.dat
- menu_jpn.dat
- menu_kor.dat
- menu_pol.dat
- menu_prt.dat
- menu_rus.dat
- menu_spn.dat
- menu_swe.dat
- menu_twn.dat
- MessageBoxOptions.dat
- ReportingToolbar.dat
- TemplRepPickerList.dat
- toolbar.dat

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

Fixturing Files

File location: C:\Program Files\Hexagon\2026.2\Models\QuickFix

Understanding Data Files

The entries in the PC-DMIS Settings Editor application store most of the settings in PC-DMIS. However, PC-DMIS also stores information in data files, which have a .dat extension.

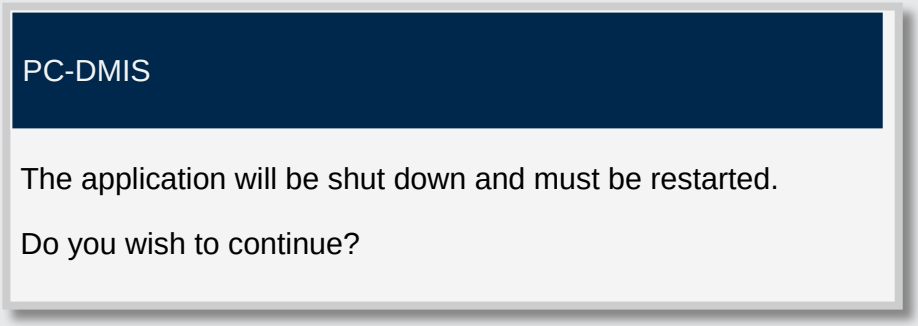
You can edit some of these files in a text editor, but you can only edit many of them within PC-DMIS. You can delete some data files if you want to return the respective PC-DMIS functionality to its original state.

The information below details many of the data files that are available, their file locations, whether you can edit them in a text editor, and where they're stored on the computer. In the descriptions of the file locations, <*user name*> is the user name that you use to log on to Windows.

PC-DMIS complies with current operating system standards required by Windows about where it stores user files.

When you first run PC-DMIS, it copies several .dat files from your folder where you installed PC-DMIS into your AppData user folder. From then on, PC-DMIS uses the user-specific file in that folder. However, if you have Administrator rights, you can modify the original factory-based file if needed.

Starting with PC-DMIS 2023.1, PC-DMIS creates device-specific folders in the AppData user folder. The software creates and updates these folders whenever you switch your device from the **Edit | Select Environment** menu option. When you select a new device, PC-DMIS saves the currently active .dat files to the previous device's folder and then it must restart. Before it restarts, PC-DMIS displays this prompt:



Click **Yes** to restart the application. When PC-DMIS starts up again, it copies the new device's .dat files from the relevant folders and applies them. This ensures that any device-specific window layouts, toolbars, and menu items are correctly recalled and applied each time you change your device.

Prior to PC-DMIS 2023.1, it might have been necessary for you to use the Settings Editor to perform a "Reset User" operation to restore certain functionality (for example, to restore DCC-related commands when you switch from a portable to a stationary CMM). With this new method of how PC-DMIS handles .dat files, this is no longer necessary. For details on how to configure and use the PC-DMIS Environment Configurator, see the "Environment Configurator" section in the PC-DMIS Core documentation.

An asterisk character (*) by the file name in the descriptions below indicates that the original, factory-based file exists in your installation directory.

You can use the PC-DMIS Settings Editor application to back up, restore, or clear user-specific files and settings. Rather than manually delete or restore individual files on your own, we recommend that you use the tools that the PC-DMIS Settings Editor provides. For more information, refer to the PC-DMIS Settings Editor documentation.

Manually moving or copying a data file to a different version of PC-DMIS is not recommended or supported.

For more information about file locations, see "Understanding File Locations".

colors.dat

This file stores your modified Edit window colors. PC-DMIS generates this file only if you modified any Edit window colors. When you exit PC-DMIS, it creates this file.

You cannot edit this file in a text editor.

If you delete this file, your Edit window colors revert to their default settings.

For information about how to define colors, see "Defining Edit Window Colors" in the "Setting Your Preferences" chapter.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

CustomMenuItem.dat

This file contains information for the default items and user-defined items in the **User Defined Commands** list on the **Menu** tab in the **Customize** dialog box (**View | Toolbars | Customize**).

You cannot edit this file in a text editor.

If you delete both CustomMenuItem.dat and toolbar.dat, PC-DMIS removes any user-defined customizations made in the **Customize** dialog box and any custom toolbars. PC-DMIS regenerates custom items if you load a toolbar.dat file that has custom items.

For more information, see "Customizing the User Interface" in the "Navigating the User Interface" chapter.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

DefaultDimensionColors.dat

PC-DMIS automatically creates and modifies this file whenever you modify the **Edit Dimension Colors** dialog box (**Edit | Graphic Display Window | Dimension Colors**).

This file contains dimension color information that:

- Defines the dimension's color limits
- Determines whether tolerance colors display in two directions
- Defines the colors that PC-DMIS uses for dimensions that are still within tolerance with percentages from 1 to 40
- Defines the color that displays when a dimension is out of tolerance with a negative value
- Defines the color that displays when a dimension is out of tolerance with a positive value

- Determines whether PC-DMIS displays tolerances as a percentage of the deviation in the Dimension Colors window (color bar)

You cannot edit this file in a text editor.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

elogo.dat *

This file controls the format of the Edit window's footer on the last page.

You can edit this file in a text editor.

DO NOT DELETE THIS FILE. If you delete this file, PC-DMIS erases the footer from the last page of the Edit window.

For more information, see "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window" chapter.

File location: In the directory where you installed PC-DMIS.

executebarstate.dat

This file contains the layouts of toolbars and dockable windows during measurement routine execution. PC-DMIS switches back and forth when it reloads these two files (gbarstate.dat and executebarstate.dat) and when you start and finish measurement routine execution.

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS restores a default layout.

File location: In the directory where you installed PC-DMIS.

gbarstate.dat

This file contains the layouts of toolbars and dockable windows within the measurement routine (but not during execution).

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS restores the default layout.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

header.dat *

This file controls the format of the Edit window's header on all pages but the first page.

You can edit this file in a text editor.

DO NOT DELETE THIS FILE. If you delete this file, PC-DMIS erases the header from the affected pages of the Edit window.

For more information, see "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window" chapter.

File location: In the directory where you installed PC-DMIS.

illumination.dat

This file is used for vision machines. It stores the defined illumination quick sets. Illumination quick sets contain information about the machines lamps, which bulbs are on, and their brightness.

You cannot edit this file in a text editor.

If you delete this file, any custom quick sets are lost, although PC-DMIS recreates this file with a few default quick sets.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

layout#.dat

These files store information about a specific layout stored on the **Window Layouts** toolbar. There is one file for each stored layout, similar to the gbarstate.dat file. PC-DMIS automatically increments "#" for each new layout.

You cannot edit these files in a text editor.

If you delete a specific layout#.dat file, PC-DMIS removes the related layout button from the **Window Layouts** toolbar.

For more information, see "Window Layouts Toolbar" in the "Using Toolbars" chapter.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

layouttoolbar.dat

This file contains a list of layouts when you use the **Window Layouts** toolbar to save a layout.

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS removes your layouts from the **Window Layouts** toolbar.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

LIGHTINGMATERIALS.DAT

This file defines the relative path to the default system-defined lighting and materials settings in default.txt. This appears as DEFAULT on the **Archive** tab in the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | Lighting, Materials**). Note that default.txt is not human readable.

The comments at the beginning of this file explain the file's format. In the file, the characters "L", "M", and "U" specify the list in which each item appears ("L" for Lighting, "M" for Material, and "U" for User Recall).

You can edit this file in a text editor.

DO NOT DELETE THIS FILE. If you delete this file, it causes the **CAD and Graphic Setup** dialog box to not know anything about the lighting or material files. You would then need to manually recreate the file or reinstall PC-DMIS.

File location: In the directory where you installed PC-DMIS.

logo.dat *

This file controls the format of the Edit window's header on the first page.

You can edit this file in a text editor.

DO NOT DELETE THIS FILE. If you delete this file, PC-DMIS erases the header from the first page of the Edit window.

For more information, see "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window" chapter.

File location: In the directory where you installed PC-DMIS.

MACHINE.DAT *

This file defines virtual machine definitions. Virtual machines are animated in the Graphic Display window as a representation of the real machine.

DO NOT DELETE THIS FILE. Instead, use the usermachine.dat file to customize MACHINE.DAT entries.

DO NOT DELETE THIS FILE. If you delete this file, the machine models become unavailable.

File location: In the directory where you installed PC-DMIS.

menu_<language>.dat

This file stores customized menus for the user. In the file name, <language> is the three-letter language code for the language you use to display PC-DMIS.

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS removes any customized menus.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

MessageBoxOptions.dat

This file stores a list of the warning messages that you have turned off.

You cannot edit this file in a text editor.

If you delete this file, all warning messages reappear.

File location: In the directory where you installed PC-DMIS.

PROBE.DAT *

This file defines the components that are available in PC-DMIS when you use the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**) to create a probe assembly.

DO NOT DELETE THIS FILE. Instead, use the usrprobe.dat file to create or display customized probe hardware.

DO NOT DELETE THIS FILE. If you delete this file, those components become unavailable.

File location: In the directory where you installed PC-DMIS.

PROBECHANGER.DAT

This file defines virtual probe changer definitions. Virtual probe changers are animated in the Graphic Display window and appear within the volume space for collision detection. It is a representation of the actual probe changer.

DO NOT DELETE THIS FILE. Use the tools that PC-DMIS provides to define your animated probe changer.

DO NOT DELETE THIS FILE. If you delete this file, the probe changer models are unavailable.

File location: In the directory where you installed PC-DMIS.

QUICKFIX.DAT *

This file displays all of the quick fixture items that are available to add into the Graphic Display window.

You can edit this file in a text editor.

DO NOT DELETE THIS FILE. If you delete this file, PC-DMIS erases the library of quick-fixture items that are available, which makes it impossible to work with these files.

File location: In the directory where you installed PC-DMIS.

ReportingToolbar.dat

This file contains a list of the saved report templates and custom reports on the **Reporting** toolbar.

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS removes any saved items from the **Reporting** toolbar.

File location: In the directory where you installed PC-DMIS.

TablesOnly.dat

This file defines the available virtual tables. It contains a list of tables and defines each table's width, length, and height, separated by commas. The values are in millimeters.

You can edit this file in a text editor.

To edit this value and show the virtual table, see "Defining Machines" in the "Defining Hardware" chapter.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

TemplRepPickerList.dat

This file contains a list of the report templates that you added to the **Report Templates** dialog box. To open the dialog box, click the **Template Selection** icon on the **Report** toolbar in the Report window (**View | Report Window**).

You can edit this file in a text editor.

If you delete this file, PC-DMIS removes those templates from the **Report Templates** dialog box, but it does not delete the actual report templates.

File location: In the directory where you installed PC-DMIS.

tool.dat

This file defines the required calibration artifacts when you want to calibrate a probe.

You cannot edit this file in a text editor.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

toolbar.dat

This file contains all of the customized toolbars definitions (buttons) and names. PC-DMIS creates this file when you customize or create any toolbars.

You cannot edit this file in a text editor.

If you delete this file, PC-DMIS removes any customized toolbars.

File location: C:\Users\<*user name*>\AppData\Local\Hexagon\PC-DMIS\2026.2

UserLightingMaterials.dat *

PC-DMIS automatically creates this file whenever you use the **Archive** tab in the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | Lighting, Materials**) to save a custom user configuration of lighting and material settings. As with the LIGHTINGMATERIALS.DAT file, it defines relative paths to any custom configuration settings files that you saved.

You can edit this file in a text editor.

File location: In the directory where you installed PC-DMIS.

usermachine.dat

This file defines custom animated machines. You can also copy machines (listed as items) from MACHINE.DAT, paste the information into this file, and customize the machines either

directly inside the file or from the **Load Virtual Machine** dialog box (**Insert | Hardware Definition | Machine**). Whenever you access that dialog box later, PC-DMIS reads this file in addition to the MACHINE.DAT file when it displays the available machines.

If you delete this file, PC-DMIS removes any custom machines.

For more information, see "Editing the usermachine.dat Data File" in the "Defining Hardware" chapter.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

usrprobe.dat *

You must manually create this file. You can then copy the probe information from PROBE.DAT, paste it into this file, and customize that information as needed. PC-DMIS then reads both this file and the PROBE.DAT file when it displays probes in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).

If you delete this file, PC-DMIS removes customized probe information.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

userquickfix.dat

This file defines the custom quick fixtures in the **Available fixtures** list in the **Quick Fixture** dialog box (**Insert | Hardware Definition | Quick Fixture**). You can also copy fixtures (listed as items) from QUICKFIX.DAT and paste the information into this file. You can then customize them directly inside the file, or you can use the **Quick Fixture** dialog box to create new custom fixtures. Whenever you access that dialog box later, PC-DMIS reads this file and the QUICKFIX.DAT file when it displays the available fixtures.

If you delete this file, PC-DMIS removes any custom fixtures.

File location: C:\ProgramData\Hexagon\PC-DMIS\2026.2

Defining Default Environment

You can set a default environment to open PC-DMIS with that environment.

To set a default environment or switch environment, follow these steps:

1. Before you open a measurement routine, from the Home page, select **Edit | Environment Configuration | Switch Environment**. This opens the **PC-DMIS Environment** dialog box.

2. From the **Select an environment** list, select the environment.

3. To set the selected environment as a default environment, click **Set as Default**.

You can set a default environment to always open PC-DMIS with that default environment. To clear the default environment, select the default environment from the **Select an environment** list and then click **Clear Default**.

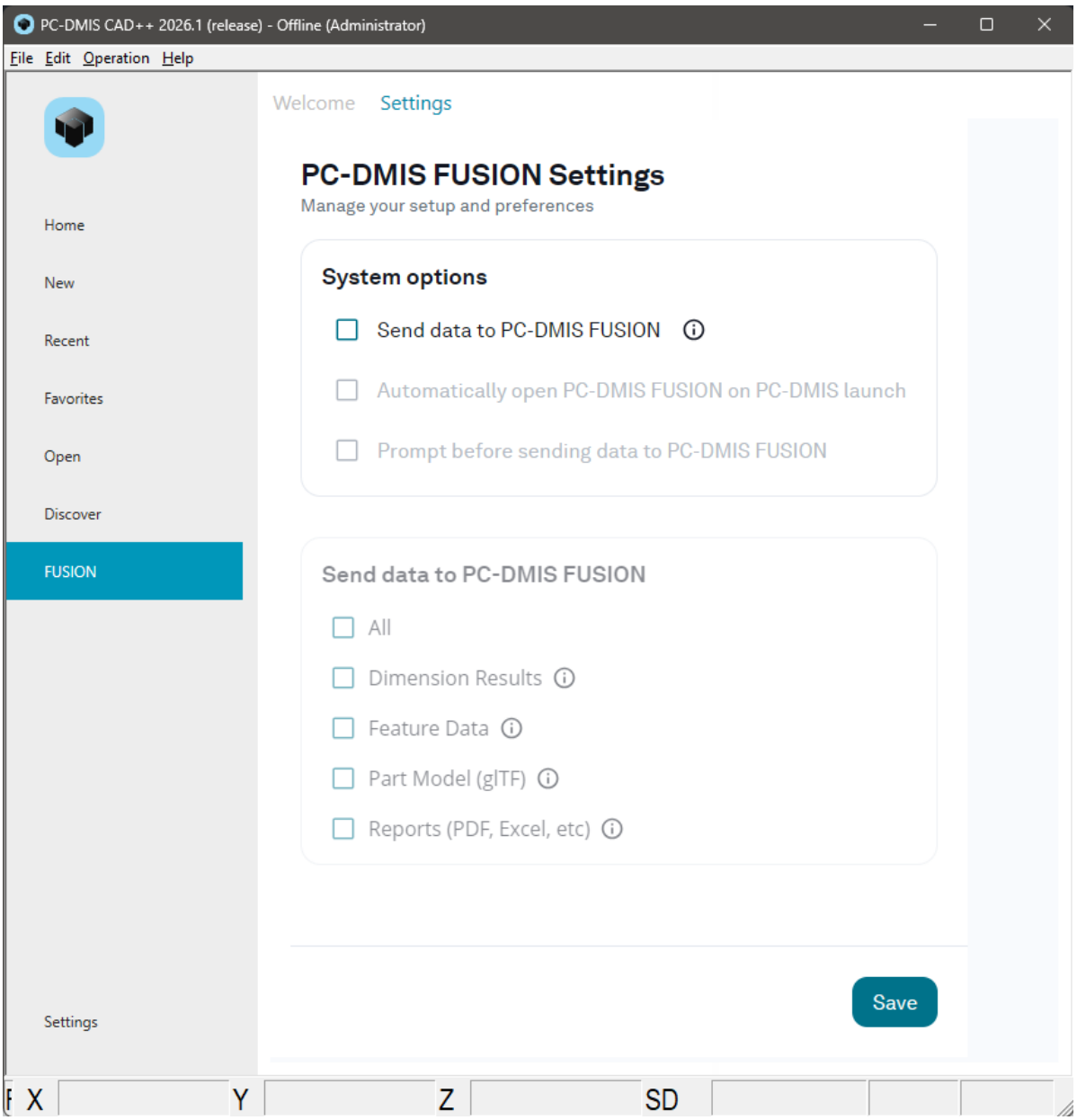
4. Click **Yes** at the prompt to switch environment. The software then restarts with the new environment settings.

You can use the **/envFile** switch to run PC-DMIS with selection of an environment or with the default environment.

If you create a shortcut that includes the name of an environment, PC-DMIS runs with that specific environment (such as **/envFile: Online**).

PC-DMIS FUSION Settings

From the Home page's left pane, when you click on the **FUSION** view and select the **Settings** tab, the **PC-DMIS FUSION Settings** screen appears with these options.



PC-DMIS FUSION Settings

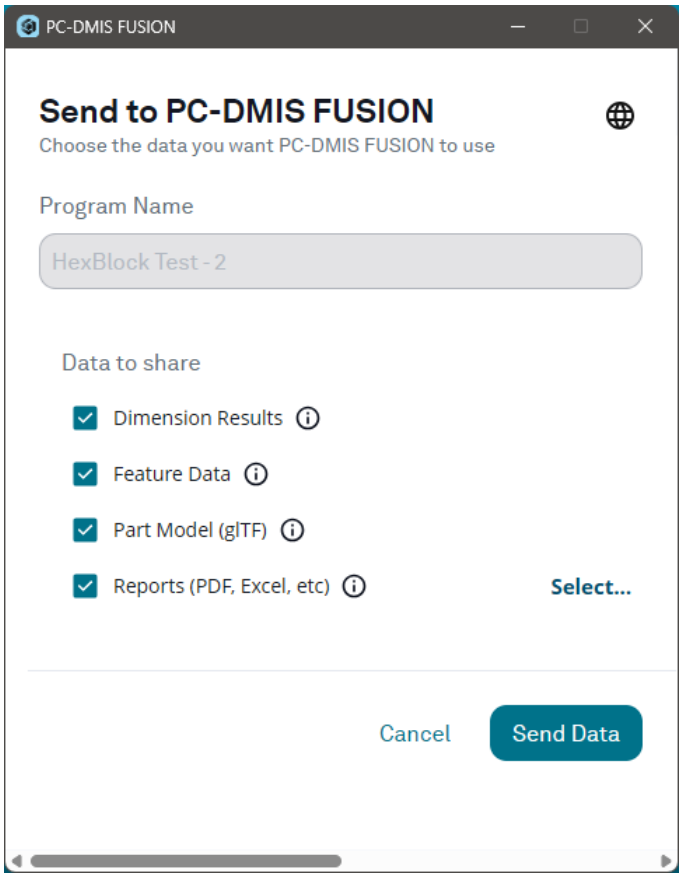
These options determine whether to send data to PC-DMIS FUSION and what data to send. PC-DMIS sends only what you choose to send, and all sent data remains on your computer.

To access the settings on this screen, you must run PC-DMIS as an administrator.

System Options

- **Send data to PC-DMIS FUSION** - This enables the other settings on this screen. The transmission of data occurs after PC-DMIS finishes execution of a measurement routine.

- **Automatically open PC-DMIS FUSION on PC-DMIS launch** - This starts the PC-DMIS FUSION application whenever you start PC-DMIS.
- **Prompt before sending data to PC-DMIS FUSION** - This provides a **Send to PC-DMIS FUSION** dialog box that prompts you to choose what data to send to PC-DMIS FUSION.



Send Data to PC-DMIS FUSION

All - This selects all the check boxes below. For each of these check boxes, you can view the information tooltip for detailed information on what PC-DMIS sends to PC-DMIS FUSION.

Dimension Results - This sends various pieces of dimension data necessary for the Sample Report, Statistics Report, and Insights.

Feature Data - This sends feature data (id, type, and other parameters) necessary for drawing features on the CAD Report.

Part Model (glTF) - This sends a light weight version of your part model. Your native CAD model is not sent.

Reports - This sends any PDF, Excel, or RTF reports that your measurement routine generates to PC-DMIS FUSION. If you have multiple reports generated from your execution, you can click **Select** on the right to send only those reports that you specify.

Editing the CAD Display

Editing the CAD Display: Introduction

PC-DMIS lets you edit the display of the CAD in the Graphic Display window.

The main topics in this chapter describe how to edit the CAD display. These include:

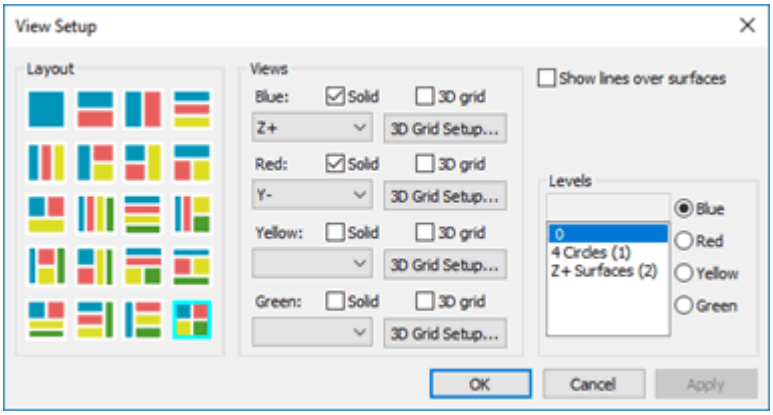
- Setting Up the Screen View
- Setting Up CAD and Graphics
- Working with CAD Levels
- Making CAD 3D by Using the Create Levels Window
- Working with CAD Groups
- Working with CAD Model Views
- Managing and Positioning On-Screen Elements
- Switching Between Curve and Surface Modes
- Drawing Surfaces
- Highlighting CAD Elements
- Displaying Mouse Coordinates in Screen Counter Text
- Repainting the Screen
- Changing Screen Modes
- Changing Screen Colors
- CAD GD&T Tab
- Changing Trihedron Settings
- Scaling the Drawing
- Rotating the Drawing
- Using the Rotation Widget
- Changing Rotation and Other Motion Options

- Showing and Hiding Graphics
- Working with CAD GD&T Callouts
- Working with Assemblies of Parts
- Creating a CAD Section Cut
- Working with a Machine Assembly
- Viewing CAD Information
- Verifying CAD Information
- Using Screen Captures of the Graphic Display Window
- Editing CAD Elements
- Editing CAD Vectors
- Selecting Features Using the Graphic Display Window
- Identifying Features by Using Tooltips
- Automatically Positioning Feature ID Labels
- Editing Dimension Colors
- Editing Feature Appearance
- Showing, Animating, and Moving Path Lines
- Graphical Hit Selection Method
- Deleting CAD Elements
- Deleting Features
- Deleting Dimensions
- Transforming a CAD Model
- Working with CAD Coordinate Systems
- Checking and Fixing Point Nominal Deviation
- Modifying Display Symbols
- Changing CAD Tolerances
- Centering the Probe on the Screen
- Viewing Clearance Planes

Setting Up the Screen View

The **Edit | Graphic Display Window | View Setup** option opens the **View Setup** dialog box. Click the **View Setup** icon from the **Graphic Modes** toolbar to also display this dialog box.

Use the **View Setup** dialog box to specify the number of views (maximum of four) of the part that PC-DMIS displays in the Graphic Display window. You can also determine the orientation of each view and if the particular view appears as a wireframe or a solid.



View Setup Dialog Box

To quickly change how the part appears in the "blue" view, you can click the icons on the **Graphic View** toolbar.

To store your view for later use, you can create and recall the view. See "Working with Viewsets" in the "Inserting Report Commands" chapter.

Changing Layout and Views

To change the screen layout and the orientation of the views, do the following:

1. Open the **View Setup** dialog box (**Edit | Graphic Display Window | View Setup**).
2. From the **Layout** area, select the desired screen style. For example, to split the screen into two different views of the part image, click on button two or three (in the first row). The window splits horizontally or vertically, based on the button you choose.
3. From the list box, select the desired view orientation for each window to display. For example, to view the part image from the Z+ direction, simply select **Z+** from the drop-down list box. If you want to view the same image from the Y- axis, select **Y-**. All

of the display options only affect how PC-DMIS displays the part image. They do not affect the measured data or inspection results.

4. If you want to display your part in the selected view of your part as a solid, select the **Solid** check box. You can leave the check box blank to display the part as a wireframe model.

5. Click either the **Apply** button or the **OK** button.

    - If you click **Apply**, PC-DMIS re-draws the Graphic Display window to reflect the current settings. This lets you preview your changes.

    - If you click **OK**, PC-DMIS closes the **View Setup** dialog box and applies the changes to the Graphic Display window. At any time before you click **OK**, you can click **Cancel** to close the dialog box and revert the views to what they were.

To redefine the sizes of the views in the Graphic Display window:

1. Position the pointer on the line separating the windows. The single arrow cursor changes to a double arrow.

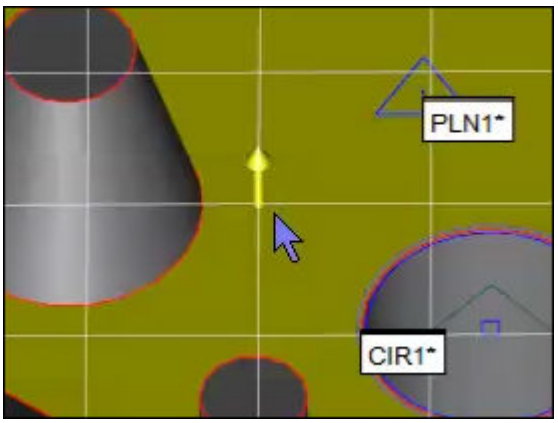
2. Click and drag the pointer to the desired location to change the view size.

This procedure works in either the horizontal or vertical position. You can then select the **Scale to Fit** option to re-scale the drawing so that it fits within the new window. For more information, see "Scaling the Drawing".

The software can be in any mode when you change the view size.

Adding a 3D Grid

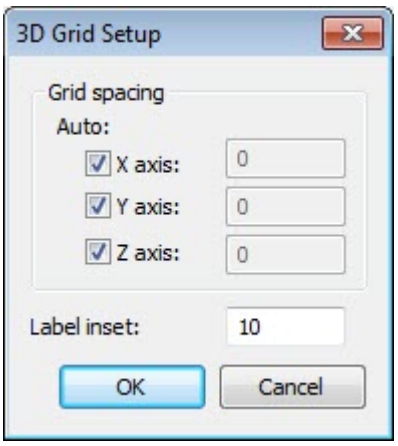
From the **View Setup** dialog box, select a **3D Grid** check box and click **Apply** to turn on the 3D grid for the selected view. The origin of the 3D grid is always located at the part's current alignment. If you enable a grid, and attempt to create a QuickFeature point (Vector, Edge, and Angle), the software snaps the point to the nearest intersection on the grid:



Example showing the Vector Point snapping to an intersection.

For information on creating QuickFeatures, see "Creating QuickFeatures by Hovering over CAD Elements" in the "Creating Auto Features" chapter.

From the **View Setup** dialog box, the **3D Grid Setup** button brings up the **3D Grid Setup** dialog box where you can modify the grid spacing and label inset.

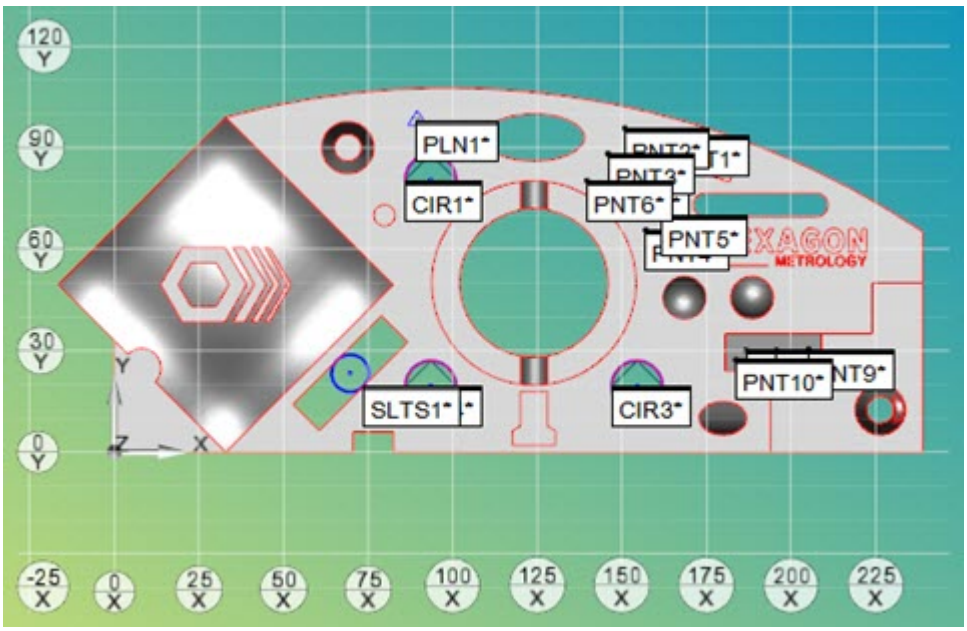


3D Grid Setup Dialog Box

Modify the grid line spacing by typing a value in the **X axis**, **Y axis**, and **Z axis** boxes or mark the **Auto** check box for that axis to have PC-DMIS automatically compute reasonable spacing.

Grid line labels are placed around the outside of the window. These lines follow the current alignment defined in the Edit window. You can move these labels along the lines by changing the value shown in the **Label Inset** box. This value determines the pixel buffer size that PC-DMIS places between the outer edge of the Graphic Display window and the 3D grid labels. You can choose any integer value between 0 and 1000.

Click **OK** to display the 3D grid in the Graphic Display window.



Example showing a 3D grid with a Label Inset of 10 pixels in the X-Y Plane.

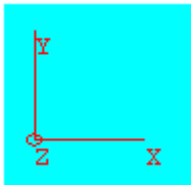
There are three possible grids that can be drawn:

X-Y

Y-Z

Z-X

PC-DMIS draws only one grid at a time. The grid drawn depends on the axis plane rotated closest to your screen (determined by the axis indicator). For example, the following axis indicator from the Graphic Display window shows the X-Y axis plane facing the user; PC-DMIS draws the X-Y axis grid.



Example of the axis indicator showing the X-Y plane.

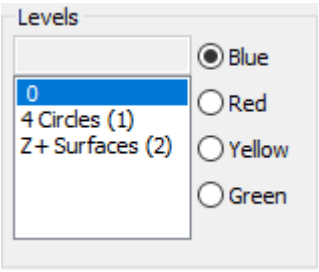
The color of the 3D grid can be modified from the **Screen Color** dialog box. See "Changing Screen Colors".

Showing lines over surfaces

This check box applies to solid view only. Select this check box to cause points and lines that hide behind surfaces to show through. This option is useful for looking at measured features that are behind surfaces. For example, suppose a measured line in your part is

slightly underneath the CAD surface. Selecting this check box will force the measured line to be visible.

Viewing CAD Levels



View Setup Dialog Box - Levels Area

You can view predefined levels of CAD elements in an associated "view" when you click on the level from the list. Then click the **Blue**, **Red**, **Yellow**, or **Green** option button to select the view.

By default, even if you haven't defined a level, PC-DMIS always displays a hidden 0 level. The 0 level is a default level that contains all the CAD elements in the CAD model. To create and work with CAD levels, see the "Working with CAD Levels" topic.

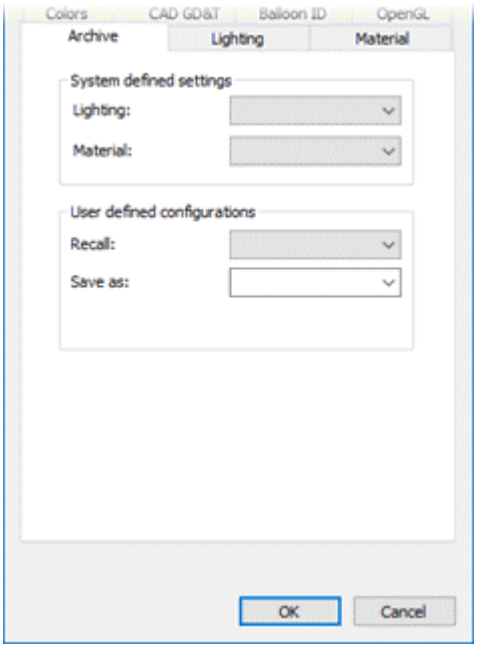
Setting Up CAD and Graphics

The **CAD and Graphic Setup** dialog box gives you a set of tools that allows you to further manipulate the display of CAD models (such as your part model) inside the Graphic Display window. These models include your part model, probe model, machine model, and other fixture models.

You can also use the dialog box to change other properties that affect how things are drawn on the screen.

You can open the dialog box by doing one of the following:

- From the **View | Toolbars | Graphic Modes** toolbar, click the **CAD Lighting** icon .
- From the **Edit** menu, point to **Graphic Display Window**, and choose **Lighting, Materials**.



CAD and Graphic Setup Dialog Box

This dialog box contains these tabs:

Archive - This tab saves and recall these CAD configurations for later use.

Lighting - This tab defines and enables light sources, their colors, their directions, light models, and CAD transparency.

Material - This tab applies texture mapping to the surfaces of your CAD models. You can also change the colors and brightness of the materials making up the textures.

Clip Planes - This tab defines up to four Clipping Planes that hide your part model in the clipped area. You can use this feature to cut your part model to show cross sections and cutaway views.

Trihedron - This tab modifies the main alignment trihedron's size as well as the color of each axis and component of the trihedron symbol.

Symbols - This tab sets the properties for the display of the following:

- Wireframe and surface objects
- Point symbols
- Path lines

Wireframe - This tab defines the color and width of surface edges.

Colors - This tab defines the colors for the following:

- The screen's background color
- The highlight color
- The mouse-over highlight color

- The 3D grid color
- The Vision's Field of View (FOV) color.

CAD GD&T - This tab defines the colors and other options for CAD GD&T callouts.

Pan, Zoom, Rotate - This tab defines how PC-DMIS interprets mouse gestures for pan, zoom, and rotate in the Graphics Display window.

Balloon ID - This tab uses a Microsoft Excel file to map characteristic IDs to balloon numbers.

OpenGL - This tab defines the OpenGL options that affect the display of the model in solid view mode.

These four buttons are at the bottom of the dialog box:

OK - Accepts your changes and closes the dialog box.

Cancel - Rejects your changes and closes the dialog box.

Save System Defaults - Saves your changes as the new default lighting and material settings. This overwrites the default settings that came with PC-DMIS.

Restore System Defaults - This brings back the original lighting and material settings that came with PC-DMIS.

Any modifications you make to the **CAD and Graphic Setup** dialog box take place in real time in the Graphic Display window. This lets you immediately see the effects of your selected changes. Until you click **OK**, however, the changes you make are only temporary.

Settings to appropriately display CAD models

PC-DMIS renders CAD models best when the CAD geometry, including the surface features, is set to transparent. This is accomplished by selecting the **Enable transparency** check box on the **Lighting** tab of the **CAD and Graphics Setup** dialog box.

If the transparent CAD model needs to be more visible, lower the transparency:

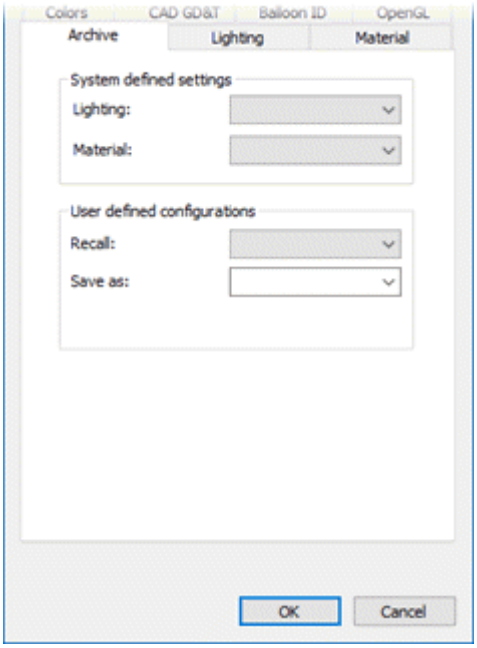
1. Access the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | Lighting, Materials**).
2. From the **Lighting** tab, set the **Transparency** slider to a lower value, such as 50%.

To produce a better and more accurate rendering of your model:

1. From the **CAD and Graphic Setup** dialog box, click the **OpenGL** tab.
2. From the **OpenGL** tab, select the **High quality transparency** check box.
3. From the **Lighting** tab, select the **Single layer** check box.

4. Set the **Transparency** slider as needed, such as 70%.

Archive Tab



CAD and Graphic Setup Dialog Box - Archive Tab

The **Archive** tab of the **CAD and Graphic Setup** dialog box contains two areas, the **System defined settings** area and the **User defined configurations** area.

The **System defined settings** area lets you select from system defined lighting and material settings. These are stored in a configuration file named LightingMaterials.dat in the directory where you installed PC-DMIS.

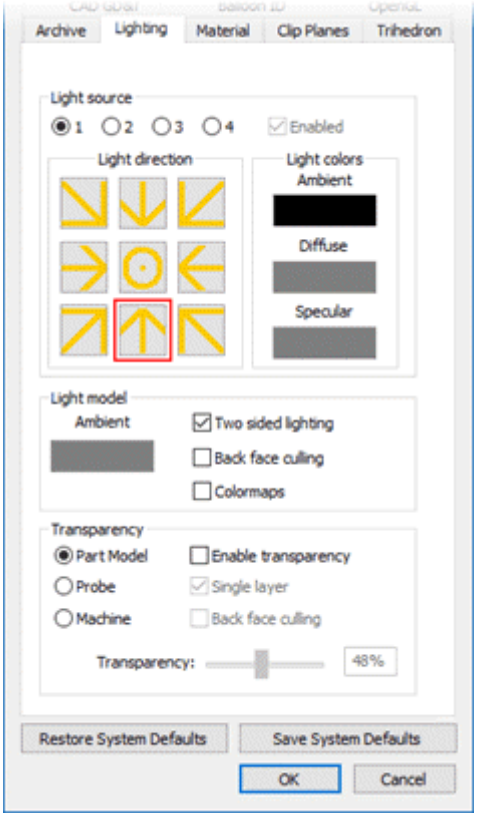
The **User defined configurations** area lets you save and recall your own custom lighting and materials configurations. The software stores them in the default.txt file located in the \Models\LightingMaterials sub directory where you installed PC-DMIS.

To save a custom defined configuration:

1. Make any changes you want in the other tabs of the **CAD and Graphic Setup** dialog box.
2. Click in the **Save as** box. A **Save Now** button appears.
3. Type a name for your saved configuration.
4. Click the **Save Now** button.

To recall a user-defined configuration, select the saved configuration from the **Recall** list.

Lighting Tab

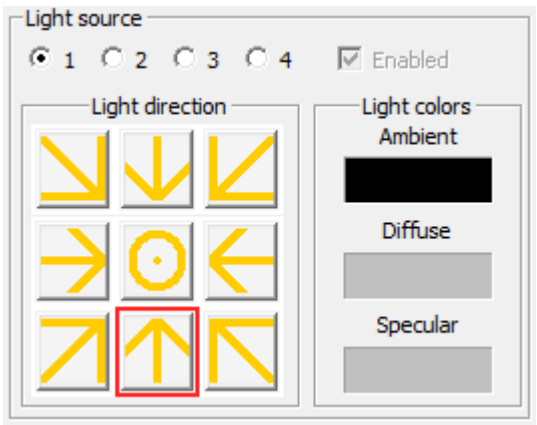


CAD and Graphic Setup Dialog Box - Lighting Tab

The **Lighting** tab of the **CAD and Graphic Setup** dialog box consists of three areas that allow you to define your light sources, your light model, and transparencies.

Light source:

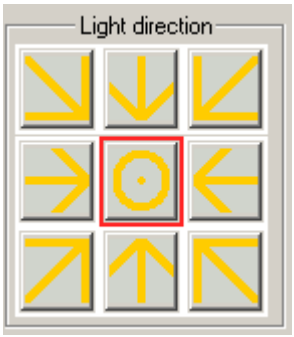
The **Light source** area defines up to four light sources, each specifying a light direction and color. You can click the **Enabled** check box and select the appropriate number to create a new light source.



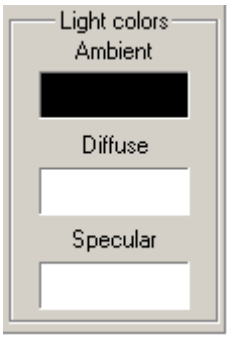
Lighting Tab - Light source Area

With the exception of the first light source, which remains always enabled, the **Enabled** check box lets you turn on or off the current light source.

Once you enable a light source, an active **Light direction** button becomes selected with a red outline. These buttons define the direction from which the light originates.



The center light direction button acts as if the light comes from the direction you are located as if shining through your computer screen onto the CAD part. The other direction buttons define light coming from other angles.



The **Light colors** section lets you change the **Ambient**, **Diffuse**, or **Specular** color for the current light source. To change the color, perform the following steps:

1. Click on the colored rectangle beneath each type of light. This action opens the **Color** dialog box.

2. From the **Color** dialog box, select the color.

Light source **1** has gray as the default color for **Diffuse** and **Specular**, while the other three light sources use black as the default color.

Ambient illumination is light that has been scattered so much by the environment that its direction is impossible to determine - this light seems to come from all directions. Backlight in a room has a large ambient component, since most of the light that reaches your eye has bounced off many surfaces. A spotlight outdoors has a tiny ambient component; most of the light travels is the same direction, and since you are outdoors, very little of the light reaches your eye after bouncing off other objects. When ambient light strikes a surface, it is scattered equally in all directions.

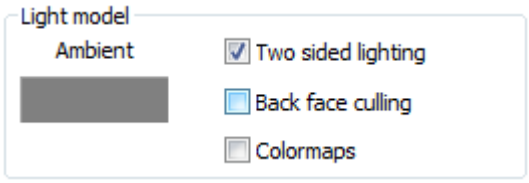
Diffuse light comes from one direction, so it is brighter if it comes squarely down on a surface than if it barely glances off the surface. Once it hits a surface, however, it is scattered equally in all directions, so it appears equally bright, no matter where the eye is

located. Any light coming from a particular position or direction probably has a diffuse component.

Specular light comes from a particular direction, and it tends to bounce off the surface in a preferred direction. A laser beam bouncing off a mirror produces almost 100 percent specular reflection. Shiny metal or plastic has a high specular component, and chalk or carpet has almost none. You can think of specular light as "shininess".

If you disable a light source, the **Light direction** and **Light colors** features become unavailable for selection.

Light model:



Lighting Tab - Light model Area

The **Light Model** area defines information applied to the entire scene, regardless of the light source.

Two sided lighting - This check box determines whether or not the front and back faces of surfaces should be lighted. You should select this check box for imported IGES files and for some other CAD formats when the surface normals are not correct.

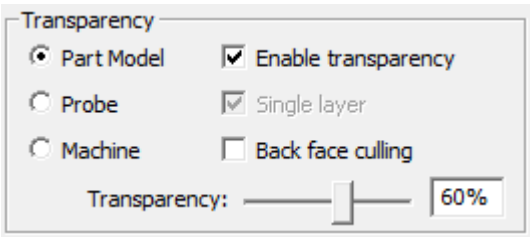
Back face culling - This check box determines whether or not PC-DMIS should cull, or hide, the back faces of surfaces. You should select this check box for IGES files, and some other CAD formats when the surface normals are not correct.

Colormaps - This check box determines whether or not PC-DMIS applies the lighting properties on the **Lighting** tab to colormaps.

- if you mark this check box, the software applies the lighting properties to colormaps whenever it renders the CAD model.
- If you clear this check box, the software does not apply the lighting properties; the colormap colors have a flat appearance, but the colormap colors exactly match the color scale that the colormap uses.

Ambient – This box defines the ambient color applied to the entire scene. Click on the box to change colors.

Transparency:



Lighting Tab - Transparency Area

Working with simulated probes or machines on your screen can be difficult if they block your sight to your part model. To reduce this problem, the **Transparency** area lets you make certain objects transparent inside the Graphic Display window. In this way, you can see your part model or other CAD objects even if your probe or machine is in the way.

You can make these objects transparent inside the Graphic Display window:

- Part model
- Probe
- Machine

You can also turn on or off the part model's transparency with the **Part Model Transparency** icon ( ) on the**Graphic View** toolbar.

Once you have enabled transparency for a simulated machine, you can then select features and other items on your part with a click on your part model *through the transparent machine*.

Once you select the **Enable transparency** check box, you can select the other items in the **Transparency** area. When you enable transparency for an object, the other settings in this area control how the transparency is to be shown.

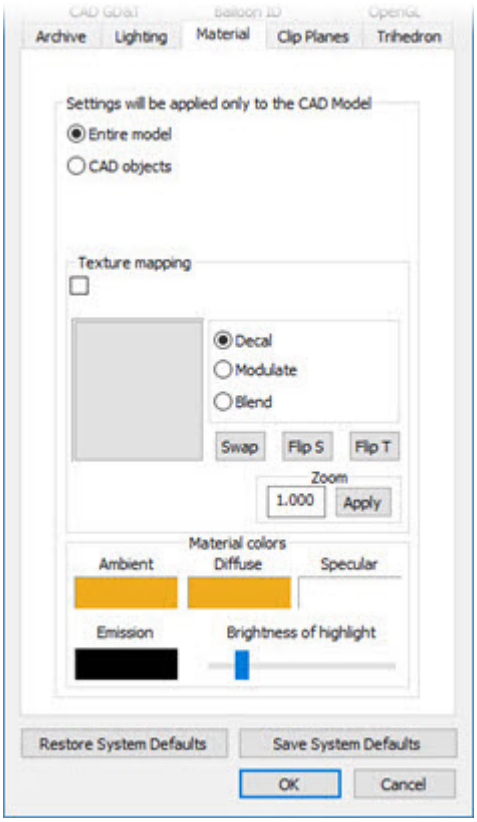
Single layer - This check box is only enabled when the **High quality transparency** check box from the **OpenGL** tab is enabled and checked. If you select the **Single layer** check box, only the top-most transparent surface of the selected model (CAD Model, Probe, or Machine) is rendered. In effect, this check box hides the internal features of a transparent model even though other models can still be viewed through the transparent model. Note that rendering is faster when this option is selected.

Back face culling - This check box causes PC-DMIS to not draw the back faces of surfaces for the selected transparent object.

Transparency - This slider lets you determine the percentage of transparency for the selected object. Moving the slider to the left makes the display object more opaque, while moving the slider to the right makes it more transparent. The value (0 - 100) can also be updated in the **Transparency** box to the right of the slider.

Material Tab

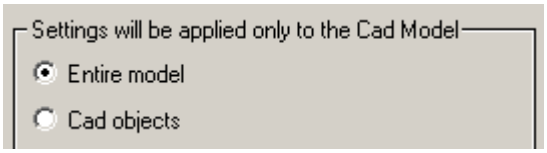
From the **Edit** menu, point to **Graphic Display Window**, choose **Lighting, Materials** and select the **Material** tab.



CAD and Graphic Setup Dialog Box - Material Tab

The **Material** tab of the **CAD and Graphic Setup** dialog box consists of settings that you can apply to CAD objects that make up the imported CAD part model or the entire part model altogether.

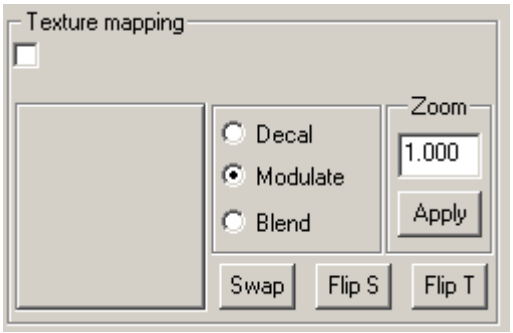
The first area of this tab contains two option buttons that allow you to determine the "active selection", either **Entire model** or **CAD objects**.



Entire model – This option applies the settings to the entire CAD model.

CAD objects – This option applies the settings to specific selected CAD objects that make up the entire model.

Texture mapping



The **Texture mapping** area defines the texture that you want to apply to the active selection. PC-DMIS automatically applies the texture image as often as necessary to cover the entire active selection.

The check box displays the current bitmap name once you select a bitmap file to use for the texture. This check box is always selected when the active selection is a CAD object.

The large gray square area under the check box displays the current texture.

To apply a texture to your part:

1. Click the gray square area to open the standard **Open** dialog box.
2. You can use the **Open** dialog box to navigate and select a valid bitmap file. A bitmap file must have a width and height of 2 to some power. For example, 2 to the 5th power is 32 and 2 to the 4th power is 16. So, if your bitmap is 32 X 16 pixels, it would be a valid bitmap. If you had something like 32 X 20, PC-DMIS would display an error message.
3. PC-DMIS displays a preview of the selected texture inside the gray square area.
4. Click **Open** to accept the texture.

To apply the check box to the active selection, click the **Enable** check box as needed.

The other items in the **Texture Mapping** area determine how PC-DMIS should apply the texture and surface colors:

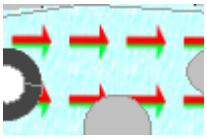
Decal prevents any of the surface's color from being emitted, so you can only see the texture's color.

Modulate and **Blend** use OpenGL algorithms to determine the final display colors.

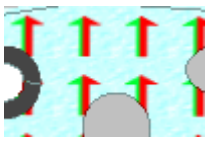
Zoom controls the "zoom" factor applied to the current texture. Values greater than 1.0 zoom *into* the texture, repeating the texture less often. Values less than 1.0 zoom *out of* the texture, repeating the texture more often. For example, a value of 2.0 makes the bitmap twice the size (repeat half as often), and a value of 0.5 makes the bitmap half the size (repeat twice as often).

The buttons **Flip T**, **Flip S**, and **Swap** determine the orientation of the texture. The texture is a two dimensional image, the orientation is described in terms of **S** and **T**. In the images below, a custom texture with an arrow drawn atop helps to visualize what happens:

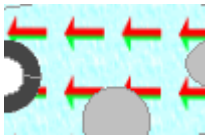
The original texture looks like this, an arrow pointing to the right with the top half of the arrow red and the bottom half of the arrow green:



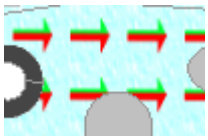
Click **Swap** on the original to switch the orientation so that **S** becomes **T** and **T** becomes **S**. This causes the arrow to point up but to also flip the bottom and top of the arrow:

 (Swap example)

Click **Flip S** on the original to change the direction of **S**. This causes the arrow to point to the left:

 (Flip S example)

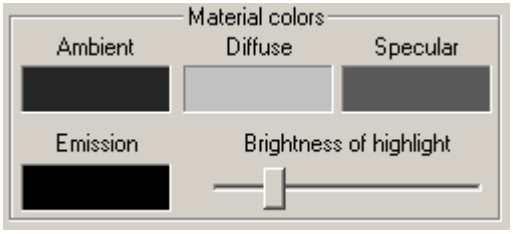
Click **Flip T** on the original to change the direction of **T**. This causes the arrow to flip, making the bottom and top switch places:

 (Flip T example)

You can click these buttons in different combinations to get a variety of orientations for your texture.

Click **Apply** to see any changes you made to texture orientations or other material colors.

Material colors



The **Material colors** area defines the color information that you want to apply to the entire CAD model.

The **Ambient**, **Diffuse**, and **Specular** function similarly to those already explained in the **Light Colors** area of the **Lighting** tab. See "Lighting Tab".

Emission - Materials may have an emissive color, which simulates light originating from an object. In the OpenGL lighting model, the emissive color of a surface adds intensity to the object, but is unaffected by any light sources. Also, the emissive color does not introduce any additional light into the overall scene.

Brightness of highlight - This slider controls the intensity of the highlight when looking at a curved surface.

Applying Textures to the Entire CAD Model

1. From the **Edit** menu, point to **Graphic Display Window**, and choose **Lighting, Materials**.

2. Click on the **Material** tab.
3. Select the **Entire model** option.
4. Under **Texture mapping**, select the gray square area. An **Open** dialog box appears.
5. Select a valid bitmap file.
6. Click **Open**. The texture appears in the dialog box.
7. Click the check box for the bitmaps in the **Texture mapping** area. The texture covers the entire CAD model.
8. Perform additional customization on the texture as needed from the other options on the dialog box.
9. Click **OK** to accept the finished texture.

Applying Textures to Selected CAD Objects

1. From the **Edit** menu, point to **Graphic Display Window**, and choose **Lighting, Materials**.
2. Click on the **Material** tab.
3. Select the **CAD objects** option. An empty list appears along with a **Clear** and a **Remove** button.
4. Under **Texture mapping**, select the gray square area. An **Open** dialog box appears.
5. Select a valid bitmap file.
6. Click **Open**. The texture appears in the dialog box.
7. Click on surfaces on the CAD model. PC-DMIS applies the surfaces you clicked with the current texture. The previously empty list now shows the individual CAD items that have textures applied to them.

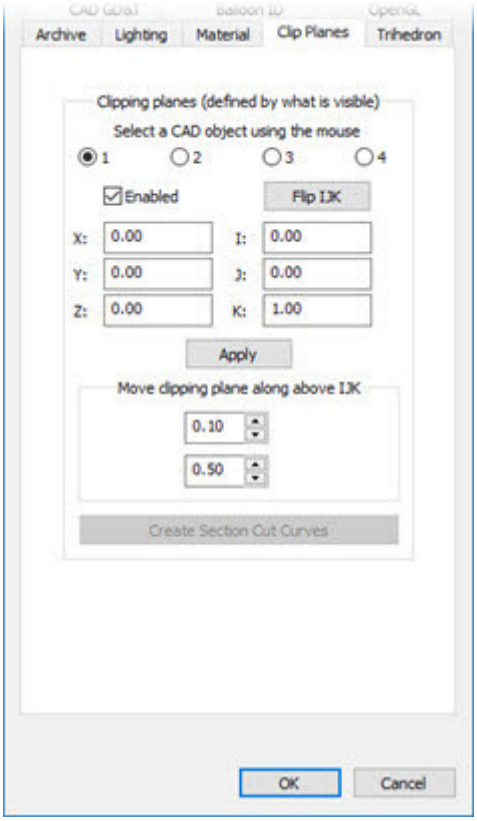
Removing Textures from Selected CAD Objects

1. Select **Edit | Graphic Display Window | Lighting, Materials**.
2. Select the **Material** tab.

   Once you apply a texture to a CAD object, the number representing that CAD object appears in a list.

- To remove textures from all selected CAD objects, click the **Clear** button.
- To remove textures from individual CAD objects, select the objects from the list and click **Remove**.

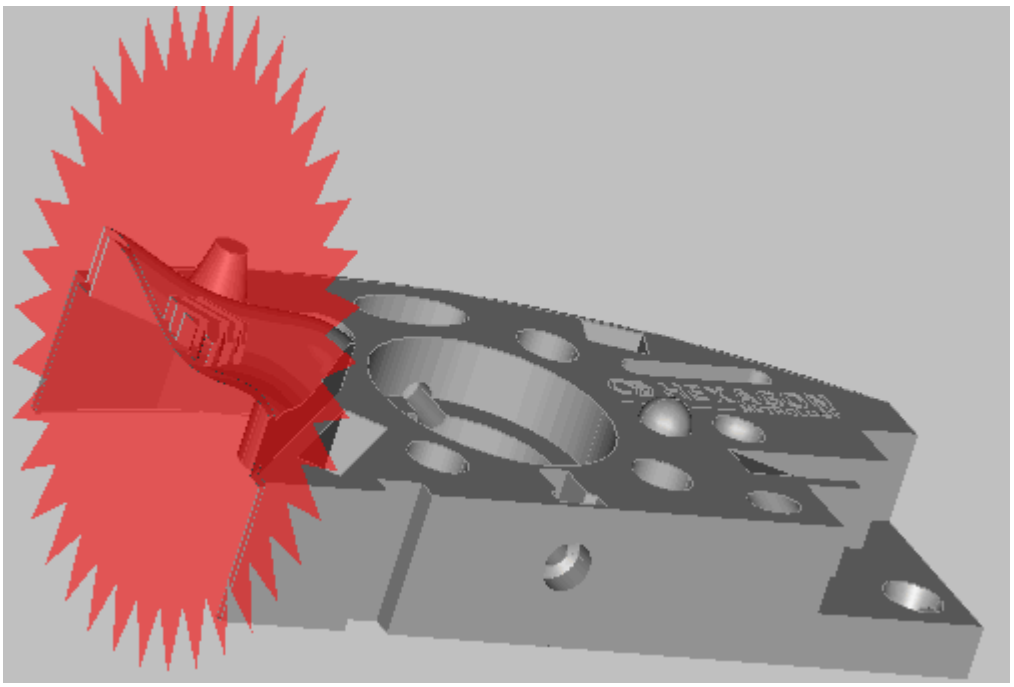
Clip Planes Tab



CAD and Graphic Setup Dialog Box - Clip Planes Tab

The **Clip Planes** tab of the **CAD and Graphic Setup** dialog box allows you to define up to four planes, called "clipping planes" that you can use to hide the display of your part model on one side of the plane. Your part model on the other side of the plane remains visible. These planes allow you to create cross sections of your part model.

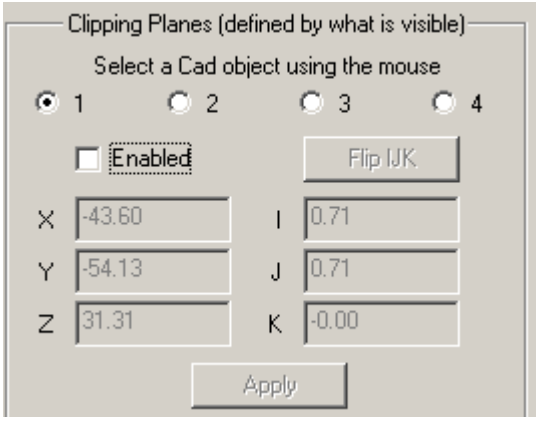
Clipping Planes appear in the Graphic Display window as round saw-like symbols.



Example of a clipping plane creating a cross section of the Hexagon test block

Defining a Clipping Plane

You need only two pieces of information to define your clipping plane: an XYZ location and an IJK vector. The IJK vector points in the direction of what remains visible.



There are two ways to define this information:

- You can click on your part in the Graphic Display window, and PC-DMIS takes the point that you clicked as the clipping plane's location. Once you click a location, PC-DMIS automatically enables your clipping plane and sets the vector to the surface normal at the location. You can then modify the IJK vector as desired.
- You can select the **Enabled** check box and then manually type in the XYZ location and IJK vector.

The XYZ and IJK values are in the active part alignment coordinate system and not the CAD coordinate system.

The **Flip IJK** button flips the vector so that it points in the opposite direction.

Once you have your clipping plane modified to where you want it, click **Apply** to see your changes. PC-DMIS hides all portions of your part model pointing away from your chosen clipping plane vector.

Clearing the **Enabled** check box disables the current clipping plane. Disabled clipping planes have a yellow color. The ability to place and move your clipping plane is also disabled. Enabled clipping planes have a red color.

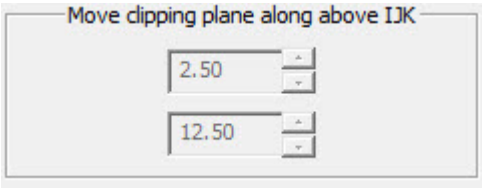
You can define multiple clipping planes by selecting a new clipping plane option button and then following the above instructions.

Adjusting a Clipping Plane

Once you have the clipping plane inserted, you can adjust it by:

- Clicking the **Flip IJK** push button to invert the IJK. This is useful when surface vectors from a CAD selection were incorrect, a relatively common problem with IGES files.
- Manually editing XYZ location and IJK vector information by typing in new values.
- Using the **Move clipping plane along above IJK** area.

The **Move clipping plane along above IJK** area contains two rows of boxes and up and down arrow buttons. The boxes define increments that the clipping plane uses to move along the defined vector when you click the up or down arrow buttons for that box.

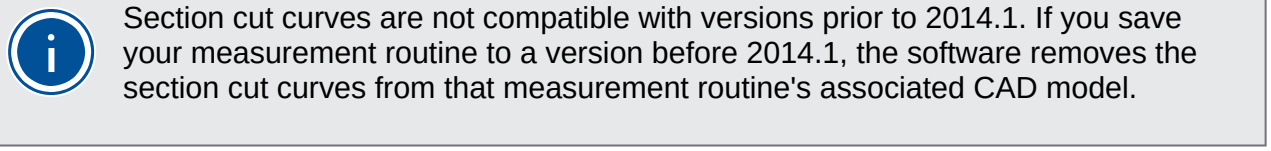


The top row allows you to move the clipping plane along the IJK vector in small adjustments.

The bottom row allows you to also move the clipping plane along the IJK vector, in larger increments.

Creating and Viewing Section Cut Curves

When you create a section cut, PC-DMIS generates a set of curves from where the clipping plane intersects the part. These section cut curves behave like any other curve in the CAD model. You can select and program the section cut curves so that your measurement routine measures them.



Section cut curves are not compatible with versions prior to 2014.1. If you save your measurement routine to a version before 2014.1, the software removes the section cut curves from that measurement routine's associated CAD model.

Creating Section Cut Curves

To create section cut curves (or polyline) from the intersection of the Clip Plane and the CAD:

1. From the **Edit** menu, point to **Graphic Display Window**, and choose **Lighting, Materials**.
2. Click the **Clip Planes** tab.
3. On your CAD model, click a CAD object.
4. From the **Clip Planes** tab of the **CAD and Graphics Setup** dialog box, select the **Enabled** check box to mark it.
5. Adjust the **X**, **Y**, **Z** and **I**, **J**, **K** values as needed.
6. Adjust the values for the **Move clipping plane along above IJK** as needed.
7. Click **Apply** to set the clipping plane and enable the **Create Section Cut Curves** button.
8. Click the **Create Section Cut Curves** button to create the section cut curves.
9. Click **OK** to close the dialog box.

Viewing Section Cut Curves

To view the section cut curve from the assembly tree view of the **CAD Assembly** dialog box:

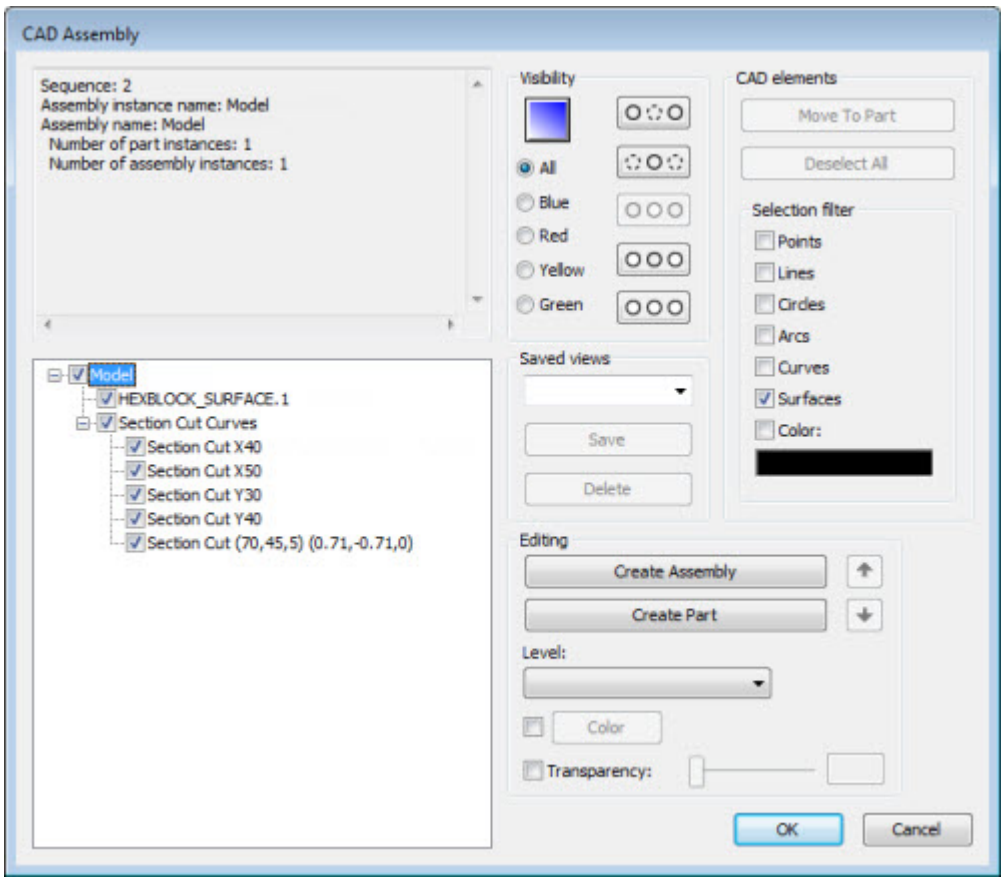
1. From the **Edit** menu, point to **Graphic Display Window**, and choose **CAD Assembly** to open the **CAD Assembly** dialog box.
2. From the assembly tree view, click the plus sign to expand the list of assembly components.
3. At the root of the assembly tree view is an assembly component named "Section Cut Curves". Click the plus sign to expand it and show the section cut curves.

4. The items that begin with "Section Cut" are followed with some descriptive text:

    - If the clipping plane is aligned with a trihedron axis, the descriptive text is that axis's character followed by the position of the plane along that axis. For example, a clipping plane located at X = 20, Y = 10, Z = -4 has an entry of "Section Cut Z-4"

    - If the clipping plane is not aligned with a trihedron axis, the descriptive text contains both the plane point and the vector values. For example, a clipping plane located at X = 80, Y = 40, and Z= -12 and a vector of 0.87, 0.0, 0.50 has an entry of "Section Cut (80,40,-12) (0.87,0,0.50)".

5. Mark or clear the check box next to each section cut to show or hide the curves in the Graphic Display window.



Example of the CAD Assembly dialog box with several created section cut curves

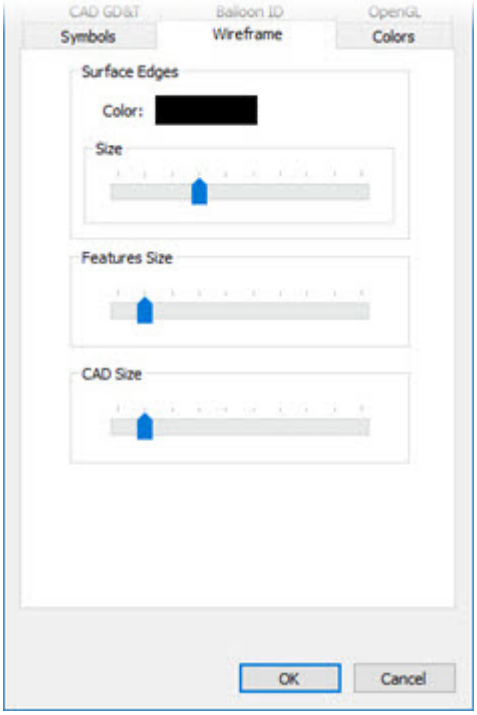
Trihedron Tab

See the "Changing Trihedron Settings" topic.

Symbols Tab

See the "Modifying Display Symbols" topic.

Wireframe Tab



CAD and Graphic Setup dialog box-Wireframe tab

The **Wireframe** tab of the **CAD and Graphic Setup** dialog box controls how to draw wireframe elements in the Graphic Display window.

Surface Edges area

You can have PC-DMIS draw bold lines around edges of surfaces. These lines are called "surface edges".

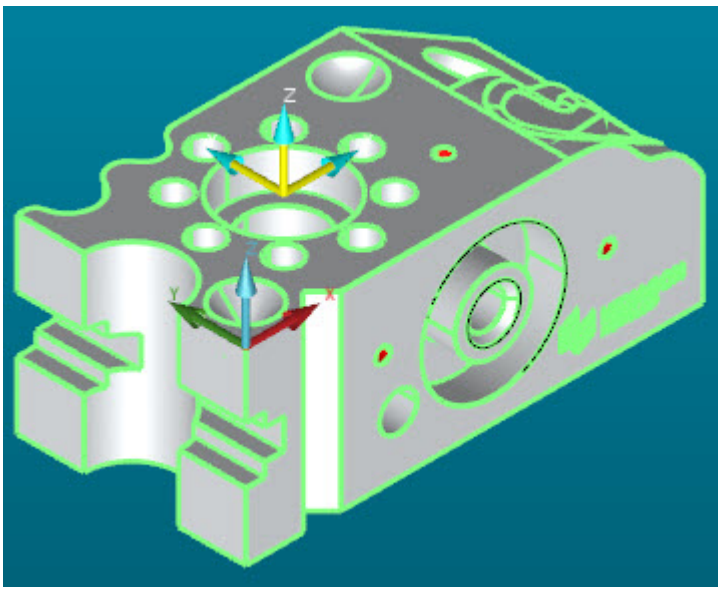
To show or hide the surface edges, from the Graphic View toolbar, click **Surface Edges** ( ).

This area controls how surface edges appear:

Color - This box opens the **Colors** dialog box, so that you can define the color of surface edges.

Size - This slider sets the width of surface edges.

This example shows surface edges with thick green lines, instead of the default black lines:

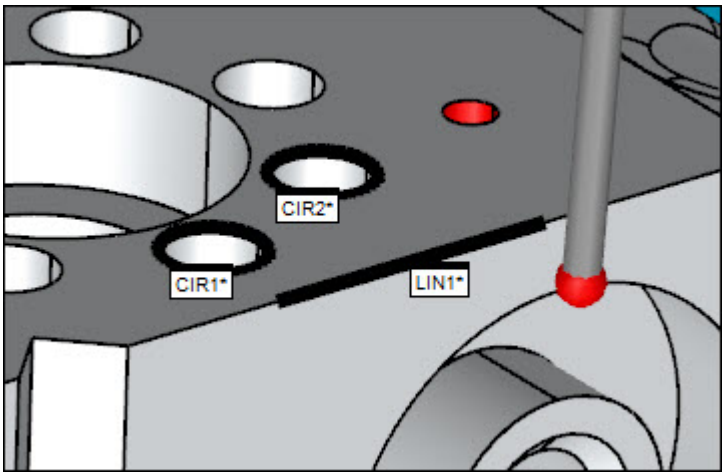


Example of the Color setting set to "Green" and the Size setting set to show thicker surface edges

Features Size area

This area contains a slider which sets the width of features that PC-DMIS draws on top of the CAD model in the Graphic Display window. For example, if you measure a line feature, PC-DMIS draws a line in 3D space at that location on the model.

This example shows two circle features and a line feature at a larger width:



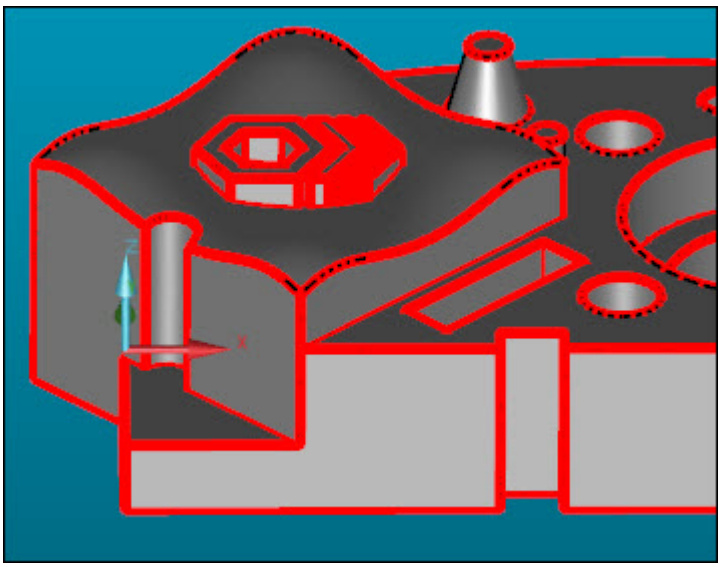
Example of the Figure Size setting set to show thicker feature widths

CAD Size area

This area contains a slider which sets the width of these items:

- CAD curves and CAD surfaces in a wireframe view
- Lines that render embedded CAD GD&T objects

This example shows part of a mixed solid and wireframe CAD model with larger wireframe lines based on this setting. (The CAD model's original format already had a red color for the wireframe lines.)

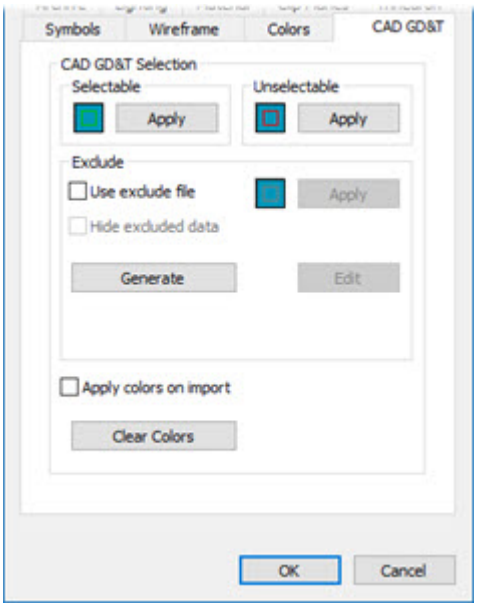


Example of the CAD Size setting that shows thicker wireframe and CAD GD&T object lines

Colors Tab

See the "Changing Screen Colors" topic.

CAD GD&T Tab



CAD and Graphic Setup Dialog Box - CAD GD&T Tab

The **CAD GD&T** tab of the **CAD and Graphic Setup** dialog box configures how CAD GD&T callouts appear in the Graphic Display window.

A CAD model with embedded CAD GD&T callouts typically has all of the dimensional requirements on it. However, some dimensional requirements you cannot measure (or should not measure) on a CMM.

This tab can help you indicate which dimensional requirements you want to include in the measurement routine when someone imports the CAD GD&T callouts as PC-DMIS features and dimensions and other commands. For information on how to work with GD&T callouts, see "Working with CAD GD&T Callouts".

The **CAD GD&T** tab consists of these areas and options:

CAD GD&T Selection area

These options configure the colors the software uses for CAD GD&T callouts in their different states.

- **Selectable** - This option defines the selectable color. These are CAD GD&T callouts that are valid for GD&T selection.
- **Unselectable** - This option defines the color for items that you cannot select. These are CAD GD&T callouts that are automatically invalid for GD&T selection. These include notes, surface finish, material specifications, and so forth.

Exclude area

You can use this area to manually exclude items from GD&T callouts that you could normally select. If you choose to exclude items, the lower part of this area lists the items that you have excluded. Excluded items are items that you have specifically identified as not wanted in the measurement routine. During CAD GD&T selection operations, PC-DMIS excludes those items.

This area also defines an optional exclude color for manually excluded items. You also have the option to hide excluded items entirely.

This area contains the following options:

Use exclude file - This check box tells PC-DMIS to use a .exclude file and exclude items you disabled from selection during CAD GD&T selection operations. These items take on the exclude color when you click **Apply**. The software lists the excluded items in the lower part of the **Exclude** area.

- The exclude file must remain in the same directory as the measurement routine.
- The exclude file's name must have the same name as the measurement routine. For example, if your measurement routine has the name of Circles.prg, the exclude file must have the name of Circles.exclude.

Color box - This opens the standard **Color** dialog box, so you can define the color for excluded items.

Hide excluded data - This check box hides any excluded items.

Generate - This button generates an exclude file from the CAD model. This text file contains all the embedded CAD GD&T callouts in the model. It has a filename of <measurement routine>.exclude, where <measurement routine> is the name of your

current measurement routine. PC-DMIS stores this file in the same directory as your measurement routine.

By default, all the lines in this text file are enabled for selection by a single quote character at the beginning.

A sample line might look something like this:

```
' "24", Diameter Dimension 'PMILW_DIM_hole(24)',
Characteristic ID: 24, Source seq: 2685, Sequence: 907
```

To make a line excluded from CAD GD&T selection, remove the beginning quote character.

Edit - This button opens the .exclude file inside of Notepad.

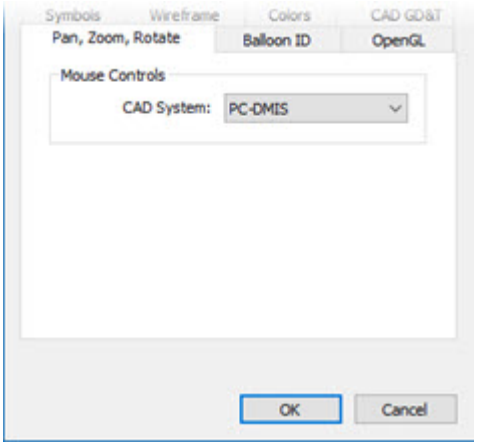
Apply Colors on Import - This check box is for convenience. It tells PC-DMIS to apply the selectable and unselectable colors defined here to the CAD model during import. The colors are applied automatically to the CAD GD&T objects at that time. Also, if you mark the **Use exclude file** check box above, PC-DMIS applies the exclude color to the imported model as well.

Clear Colors - This button resets all colors to their default values.

For information on how to work with the **Color** dialog box, see the "To Change a Color" topic.

Pan, Zoom, Rotate Tab

From the **Edit** menu, point to **Graphic Display Window**, and choose **Lighting, Materials**. Then click the **Pan, Zoom, Rotate** tab.



CAD and Graphic Setup Dialog Box - Pan, Zoom, Rotate Tab

The **Pan, Zoom, Rotate** tab defines how PC-DMIS interprets mouse gestures to perform pan, zoom, and rotate operations in the Graphic Display window.

From the **CAD Systems** list in the **Mouse Controls** area, you can choose the CAD system that PC-DMIS mimics for the pan, zoom, and rotate operations.

Common Controls

These mouse gestures are common to all **CAD System** options:

Left button -

Click to select an item.

Click and hold, then drag to box-select multiple items.

Click and hold, then right-click and hold, then drag to zoom the view to that box.

Mouse Wheel button -

Rotate the wheel forward or backward to zoom in and out.

Right button -

In **Translate Mode** ( ), right-click and hold, then drag to pan.

In **2D Rotate Mode** ( ), right-click and hold, then drag to do a 2D rotation.

In **3D Rotate Mode** ( ), right-click and hold, then drag to do a 3D rotation.

PC-DMIS

If you choose **PC-DMIS**, the software interprets mouse gestures like this:

Mouse Wheel button -

Click the wheel button and hold, then drag to do a 3D rotation.

Right button -

Right-click and hold, then drag to pan the view.

Right-click and hold, press Ctrl, and then drag to do a 3D rotation.

Right-click and hold, press Alt, and then drag to do a 2D rotation.

CATIA 5

If you choose **CATIA v5**, the software interprets mouse gestures like this:

Mouse Wheel button -

Click the wheel button and hold, then drag to pan.

Click the wheel button and hold, press Ctrl, and then drag to zoom in or out.

Click the wheel button and hold, then left-click and hold, then drag to do a 3D rotation.

Click the wheel button and hold, then left-click and release, then drag to zoom in or out.

Creo

If you choose **Creo**, the software interprets mouse gestures like this:

Mouse Wheel button -

Click the wheel button and hold, then drag to do a 3D rotation.

Click the wheel button and hold, press Shift, and then drag to pan.

Click the wheel button and hold, press Ctrl, and then drag left or right to do a 2D rotation.

Click the wheel button and hold, press Ctrl, and then drag up or down to zoom in or out.

NX

If you choose **NX**, the software interprets mouse gestures like this:

Middle button -

Click the wheel button and hold, then drag to do a 3D rotation.

Click the wheel button and hold, press Shift, and then drag to pan.

Click the wheel button and hold, press Ctrl, and then drag to zoom in or out.

SolidWorks

If you choose **Solidworks**, the software interprets mouse gestures like this:

Mouse Wheel button -

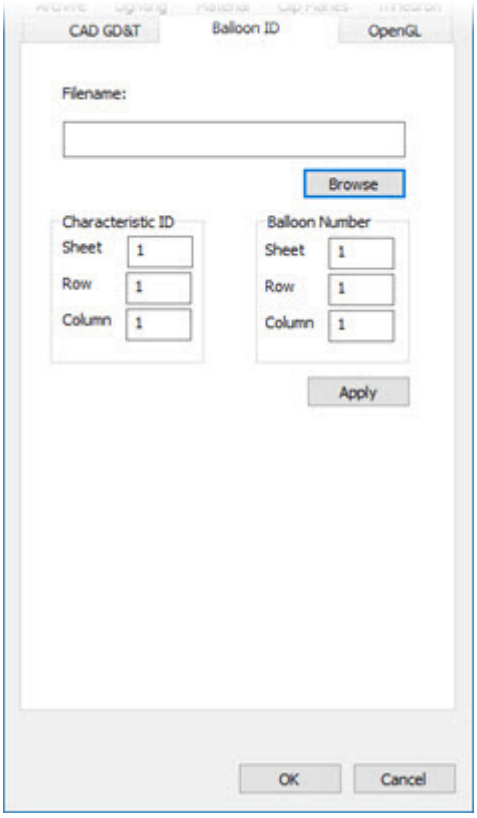
Click the wheel button and hold, then drag to do a 3D rotation.

Click the wheel button and hold, press Alt, and then drag to do a 2D rotation.

Click the wheel button and hold, press Shift, and then drag to zoom in or out.

Click the wheel button and hold, press Ctrl, and then drag to pan.

Balloon ID Tab



CAD and Graphic Setup Dialog Box - Balloon ID Tab

The **Balloon ID** tab uses a Microsoft Excel file to map characteristic IDs to balloon numbers. For information on showing the characteristic IDs, see the "Showing and Hiding CAD GD&T Characteristic IDs" subtopic under the "Working with CAD GD&T Callouts" topic.

You can use an Excel file with any of these file extensions: .xls, .xlsx, .xlsm

How It Works

Once you supply a pathway to an Excel file with data, and click **Apply** or **OK**, PC-DMIS uses an internal Excel reader to read the file.

Once you choose to display characteristic IDs, PC-DMIS reads the Excel file and uses the info on the **Balloon ID** tab to locate the starting characteristic ID.

For each ID in the defined characteristic ID column, PC-DMIS checks if that ID number is in the model.

If so, PC-DMIS looks up the corresponding balloon number and replaces that characteristic ID with the balloon number.

This does not replace the underlying characteristic IDs embedded in the model.

Items on the Tab

Filename - This box defines the full path to the Excel file. You can use the **Browse** button to select the folder.

- If this box contains a pathway to a valid Excel file, PC-DMIS uses the Excel file to determine what balloon number to show for each characteristic ID.
- If this box does not contain a pathway to a valid Excel file or is empty, PC-DMIS shows the default characteristic IDs.

Characteristic ID - This area defines the sheet, row, and column for the first cell in a column of characteristic IDs.

Balloon Number - This area defines the sheet, row, and column for the first cell in a column of balloon numbers.

Suppose you have an Excel file with this information on sheet 1:

	A	B
1	Characteristic ID	Balloon Number
2	20	1B
3	21	15B
4	22	20B
5	24	2B
6	25	3B
7	26	4B
8	27	5B
9	28	6B
10	29	21B

To replace the numbers, you would set these values in the **Characteristic ID** area:

Sheet - 1

Row - 2

Column - 1 (for column A)

And you would set these values in the **Balloon Number** area:

Sheet - 1

Row - 2

Column - 2 (for column B)

When PC-DMIS encounters the characteristic ID of 20 in the model, it would replace it with the balloon number of 1B, and so on.

OpenGL Tab

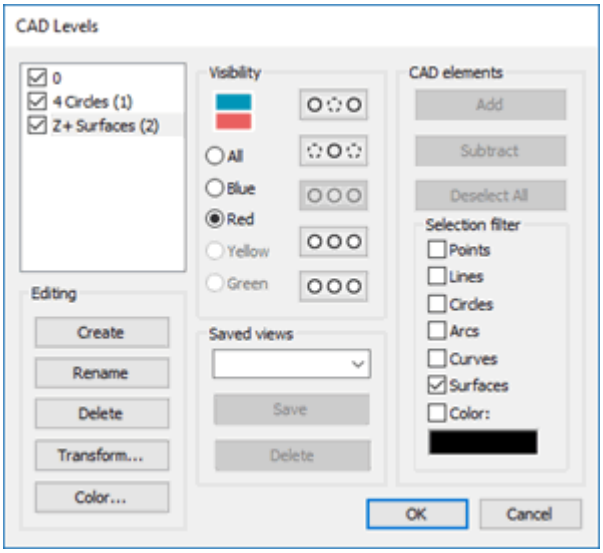
See "Changing OpenGL Options" in the "Setting Your Preferences" chapter.

Working with CAD Levels

A CAD level consists of a group of user-selected CAD geometries or elements selected from the Graphic Display window. The level information is saved with the .CAD file, so that if you import the .CAD into another measurement routine, the level information is available there too.

You can define which levels are displayed in each view of the Graphic Display window by using the **CAD Levels** dialog box (**Edit | Graphic Display Window | CAD Levels**). Once levels are defined, you can view the levels using either the **CAD Levels** dialog box or the **View Setup** dialog box (**Edit | Graphic Display Window | View Setup**).

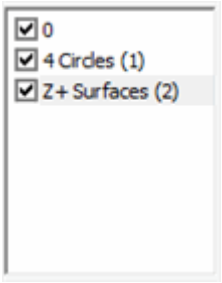
To open the **CAD Levels** dialog box, from the **Graphic Modes** toolbar, click the **CAD Levels** icon , or select **Edit | Graphic Display Window | CAD Levels**. This dialog box lets you define the CAD elements to assign to the levels. You can also use it to modify levels, delete levels, and view levels in the Graphic Display window:



CAD Levels Dialog Box

The dialog box contains these items:

Levels List



The list of levels in the upper-left area of the dialog box shows all the current levels in the CAD model. The list always contains the default level 0, a level that automatically contains all the CAD elements in the CAD file.

Here are some operations you can perform using the levels list:

- Double-click on a level from the list to select all CAD elements assigned to that level.
- Select a CAD element in the Graphic Display window, and press and hold Ctrl on the keyboard to select the corresponding level in the level list.
- Select the check box next to a level to show the selected level.
- Clear the check box to hide the selected level.

Visibility

The **Visibility** area provides option buttons and icon buttons that allow you to run visibility operations in order to change the hidden state of levels.

The colored icon at the top of this area shows your current screen layout. It corresponds with the icon used in the **Layout** area of the **View Setup** dialog box. See "Setting Up the Screen View".

The **Blue**, **Red**, **Yellow**, and **Green** options limit the visibility operation to only that "view" of the Graphic Display window. The **All** option applies the visibility operation to all views.

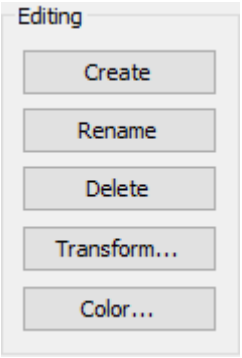
These command buttons have pictures on them that depict their action:

- **Hide** - Hides the level selected in the list.

- **Hide Siblings** - Hides all the levels except the level selected in the list.
- **Show** - Shows the level selected in the list.
- **Show Siblings** - Shows all the levels except the level selected in the list.
- **Show All** - Shows all the levels in the list.

These buttons are unavailable for selection until you select a level from the list of levels.

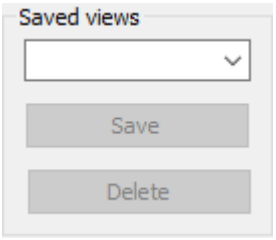
Editing



The **Editing** area allows you to modify CAD levels. For all buttons except **Create**, you need to first select a level from the levels list. The buttons affect that selected level.

- **Create** - Creates a new level in the CAD model. Initially, there are not any CAD elements assigned with this new level. You need to use the **Add** button in the **CAD Elements** section to add CAD elements selected from the Graphic Display window to the new level.
- **Rename** - Renames the selected level. You can also rename a level by clicking on a selected level in the list until the cursor appears.
- **Delete** - Deletes the selected level. PC-DMIS assigns any CAD elements that were on that level to the default level 0. You cannot delete the default level.
- **Transform** - Transforms the selected level. When you click this button, PC-DMIS displays the **CAD Transform** dialog box. See "Transforming a CAD Model" for information on using this dialog box.
- **Color** - Changes the color of all CAD elements in the selected level.

Saved Views

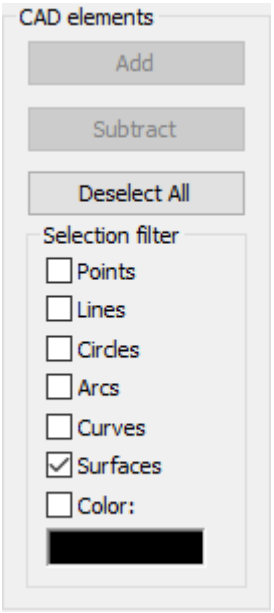


The **Saved Views** area stores the level visibility state in the CAD model. These are saved inside the .CAD file. Use the drop-down combo box to either select and recall a saved view or to specify a name for a new view.

This level visibility state does not include the number of graphics views. The number of graphics views can only be saved using viewsets. See "Working with Viewsets".

- **Save** - Saves the current level visibility state using the name entered in the combo box.
- **Delete** - Deletes the saved view selected in the combo box.

CAD Elements



The **CAD Elements** area modifies the level of the selected CAD elements. You can select (or deselect) CAD elements by simply clicking on or box-selecting them in the Graphic Display window. Selected elements are highlighted.

- **Add** - Adds the selected CAD elements to the selected level in the list.
- **Subtract**- Removes the selected CAD elements from the selected level in the list. The CAD elements are assigned the default level 0, but only if they are on the

selected level. If a selected CAD element is not on the selected level, then that CAD element is not modified.

- **Deselect All** - Clears any selected CAD elements on the Graphic Display window.
- **Selection Filter** - The check boxes in the **Selection Filter** area filter the type of CAD elements assigned to a CAD level. You can also use the **Color** check box in this area to filter your selection by the color of the CAD element. When you select the **Color** check-box, the color of the next CAD element that you select in the Graphic Display window is used as the filter color. For subsequent selections, you can only select CAD elements with that color.

Creating Levels

1. Access the **CAD Levels** dialog box.
2. Click **Create** from the **Editing** area. A new level number appears in the list of levels.
3. Select the new level, and rename it if desired. Even if you rename the level, PC-DMIS still shows the level number in parentheses.
4. Ensure that the CAD level is selected from the list.
5. From the Graphic Display window, select the CAD elements you want included in the CAD level. To limit which CAD element types are selectable, select the check boxes of CAD elements in the **Selection Filter** area.
6. Click the **Add** button. PC-DMIS assigns the selected elements to the selected level. Now when you show that level and hide other levels, PC-DMIS shows only those elements.
7. Click **OK** when finished to close the dialog box.

Modifying Levels

1. Access the **CAD Levels** dialog box.
2. Select the level from the level list.
3. To add elements to the level, select the CAD elements from the Graphic Display window, and click **Add**. PC-DMIS modifies the level accordingly.
4. To remove elements from the level, select the level, and display its assigned CAD elements. From the Graphic Display window, select the elements to remove, and click **Subtract**. PC-DMIS modifies the level accordingly.
5. Click **OK** when finished to close the dialog box.

Deleting Levels

1. Access the **CAD Levels** dialog box.
2. Select a level from the level list.

3. Click **Delete**. PC-DMIS deletes the level and assigns all CAD elements of that level to the 0 level.
4. Click **OK** when finished to close the dialog box.

Showing a Level for a Specific Layout Pane

1. Access the **CAD Levels** dialog box.
2. From list of levels, select the level to use, and ensure that its check box is selected.
3. From the **Visibility** area, select the **Blue**, **Red**, **Yellow**, or **Green** option button for the pane in which you want to view the level.
4. Either deselect the check boxes from other levels in the list of levels, or use the pictured **Hide Siblings** button in the **Visibility** area to hide other levels.
5. Click **OK** when finished to close the dialog box.

You can also view levels within the **View Setup** dialog box.

Making CAD 3D by Using the Create Levels Window

Note that the **Operation | Graphic Display Window | Make 3D** menu option and the associated Create Levels window were removed in PC-DMIS 2009. The functionality to create CAD levels is now contained in the **CAD Levels** dialog box. For more information, see the "Working with CAD Levels" topic.

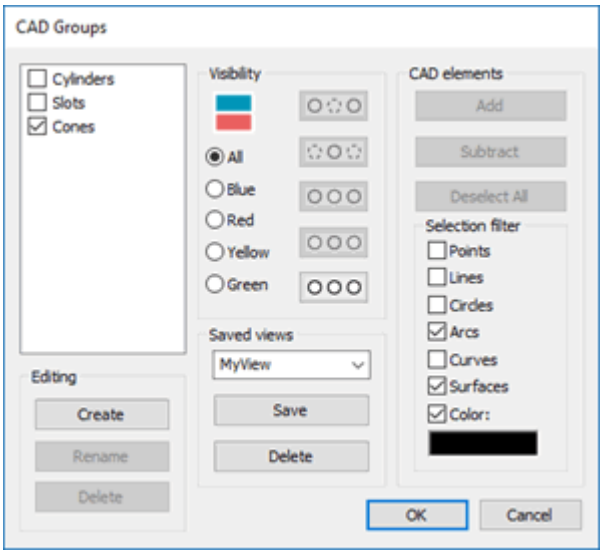
Working with CAD Groups

Similar to a CAD level, a CAD group consists of a group of user-selected CAD geometries called "elements" selected from the Graphic Display window.

However, CAD groups differ from CAD levels in these respects:

- CAD elements can belong to more than one CAD group, and there can be any number of these groups.
- CAD elements in a CAD group can span CAD levels.
- CAD elements in a CAD group can span across multiple assembly components.
- A CAD group's settings apply to all views of the graphical model.

Select **Edit | Graphic Display Window | CAD Groups** or from the **Graphic Modes** toolbar, click the **CAD Groups** button to open the **CAD Groups** dialog box.



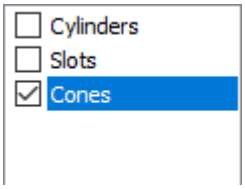
CAD Groups Dialog Box

This dialog box lets you define the CAD elements that are assigned to the groups. You can also use the dialog box to modify groups, delete groups, and view or hide groups in the Graphic Display window.

PC-DMIS saves CAD group information with the .CAD file, so that if you import the .CAD into another measurement routine, the group information is available there too.

The **CAD Groups** dialog box contains these items:

CAD Groups



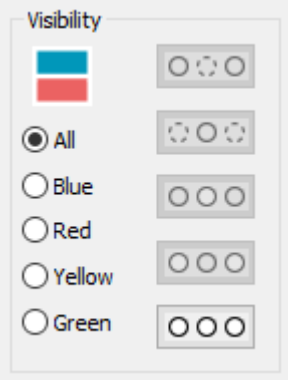
The list of CAD groups in the upper-left area of the dialog box shows all the current CAD groups in the CAD model. Because you can assign CAD elements to more than one group, this list lets you select (highlight) more than one group from the list at a time. So, for example, you could select a surface, then select two groups, and add that surface to both groups.

Here are some operations you can perform using the CAD Groups list:

- Double-click on a group from the list to select all CAD elements assigned to that group.
- Select a CAD element in the Graphic Display window and press and hold Ctrl on the keyboard to select the corresponding group in the group list.

- Mark the check box next to a group to show the CAD elements assigned to it.
- Clear the check box next to a group to hide the CAD elements assigned to it.

Visibility



The **Visibility** area provides option buttons and icon buttons that allow you to run visibility operations in order to change the hidden state of the CAD groups. If a CAD object belongs to more than one group, and at least one of the groups is hidden, that CAD object becomes hidden as well. So, even if the CAD object is visible in another group, it is still hidden.

The colored icon at the top of this area shows your current screen layout. It corresponds with the icon used in the **Layout** area of the **View Setup** dialog box. See "Setting Up the Screen View".

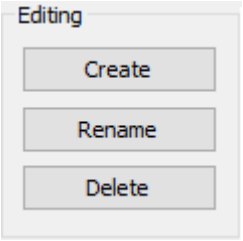
The **Blue**, **Red**, **Yellow**, and **Green** options limit the visibility operation to only that "view" of the Graphic Display window. The **All** option applies the visibility operation to all views.

These command buttons have pictures on them that depict their action:

- **Hide** - Hides the CAD group selected in the list.
- **Hide Siblings** - Hides all the CAD groups except the group selected in the list.
- **Show** - Shows the CAD group selected in the list.
- **Show Siblings** - Shows all the CAD groups except the group selected in the list.
- **Show All** - Shows all the CAD groups in the list.

These buttons have a dark gray background and are unavailable for selection until you select a CAD group from the CAD Groups list.

Editing

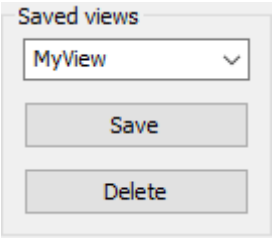


The **Editing** area allows you to modify CAD groups. For all buttons except **Create**, you need to first select at least one group from the CAD Groups list, and the buttons affect the selected group or groups.

- **Create** - Creates a new group. Initially, there are no CAD elements assigned to this new group. You need to select the CAD elements from the Graphic Display window and then use the **Add** button in the **CAD Elements** section to add the selected CAD elements to the new group.
- **Rename** - Renames the selected group. You can also rename a group by clicking on a selected group in the list until the cursor appears.
- **Delete** - Deletes the selected groups (but not the CAD elements in those groups).

No **Transform** or **Color** buttons are used with CAD groups. This is because the CAD elements can belong to more than one group.

Saved Views

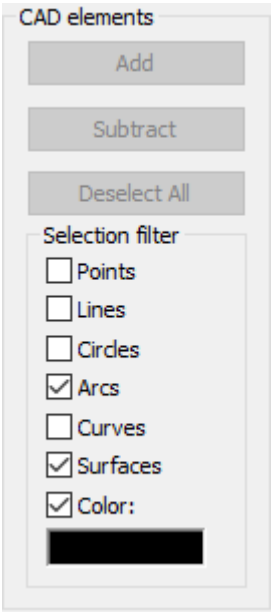


The **Saved Views** area stores the CAD group visibility state in the CAD model. These are saved inside the .CAD file. Use the drop-down combo box to either select or recall a saved visibility state or to specify a name for a new view.

This CAD group visibility state does not include the number of graphics views. The number of graphics views can only be saved using viewsets. See "Working with Viewsets".

- **Save** - Saves the current visibility state using the name that you typed in the combo box.
- **Delete** - Deletes the saved visibility state that you selected in the combo box.

CAD Elements



The **CAD Elements** area modifies the group using the selected CAD elements. You can select multiple CAD elements by pressing Ctrl and clicking on them or by box-selecting them in the Graphic Display window. Selected elements are highlighted.

- **Add** - Adds the selected CAD elements to the currently selected groups in the list.
- **Subtract** - Removes the selected CAD elements from the currently selected groups in the list. If a selected CAD element is not on the selected group, then that CAD element is not modified.
- **Deselect All** - Clears any selected CAD elements on the Graphic Display window.
- **Selection Filter** - The check boxes in the **Selection Filter** area filter the type of CAD elements assigned to a CAD group. You can also use the **Color** check box in this area to filter selection by the color of the CAD element. When you select the **Color** check-box, the color of the next CAD element that you select in the Graphic Display window is used as the filter color. For subsequent selections, you can select CAD elements only with that color.

Creating CAD Groups

1. Access the **CAD Groups** dialog box.
2. From the **Editing** area, click **Create**. A new group appears with "New Group" as the default name followed by a unique number.
3. Select the group and rename it if desired by clicking again on the group or using the **Rename** button.
4. Ensure that the CAD group is selected from the list.

5. From the Graphic Display window, select the CAD elements that you want to include in the CAD group. Your selection can span multiple part models as part of an assembly or multiple layers. To limit which CAD element types are selectable, select the check boxes of CAD elements in the **Selection Filter** area.

6. Click the **Add** button. PC-DMIS assigns the selected elements to the selected group. Now when you show that group and hide other groups, PC-DMIS shows only those elements.

7. Click **OK** when finished to close the dialog box.

Modifying CAD Groups

1. Access the **CAD Groups** dialog box.

2. Select the group from the CAD Groups list.

3. To add elements to the group, select the CAD elements from the Graphic Display window and click **Add**. PC-DMIS modifies the group accordingly.

4. To remove elements from the group, select the group and display its assigned CAD elements. From the Graphic Display window, select the elements to remove and click **Subtract**. PC-DMIS modifies the group accordingly.

5. Click **OK** when finished to close the dialog box.

Deleting CAD Groups

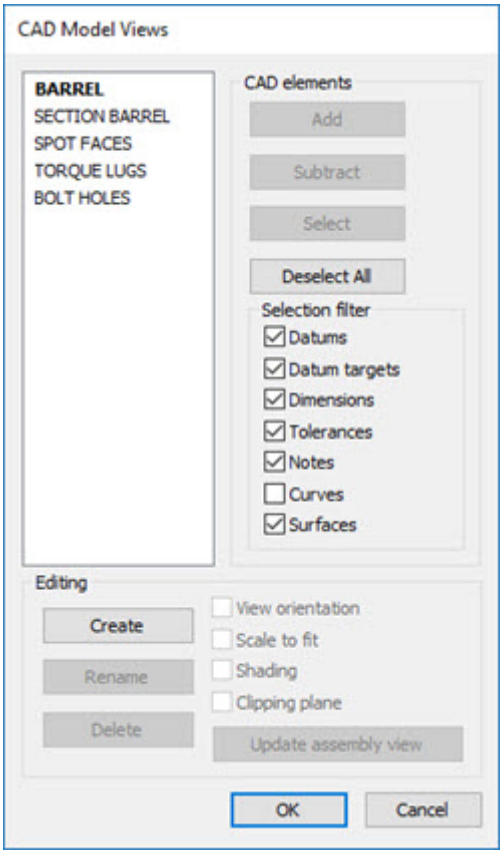
1. Access the **CAD Groups** dialog box.

2. Select a group from the CAD Groups list.

3. Click **Delete**.

4. Click **OK** when finished to close the dialog box.

Working with CAD Model Views



CAD Model Views Dialog Box

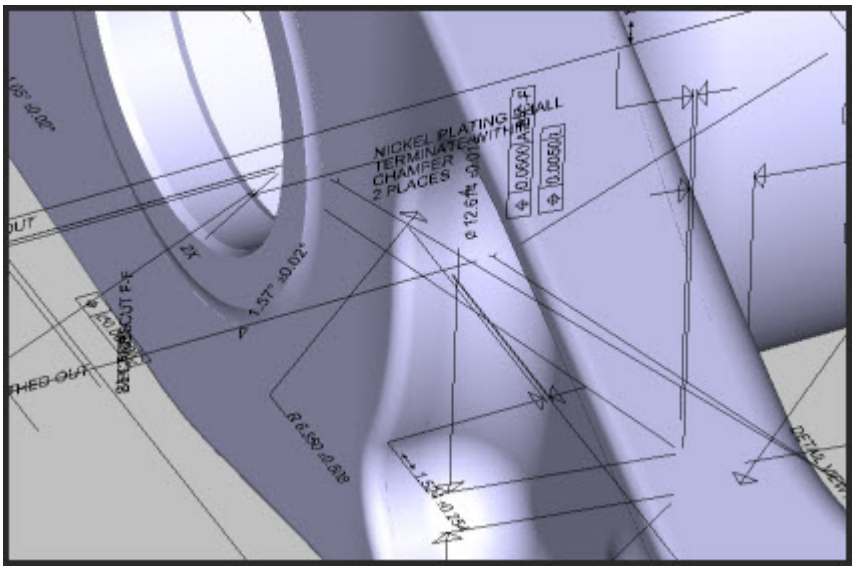
If a Model View is active, and you merge an imported CAD file, PC-DMIS adds all the visible geometry of the imported CAD model into the active Model View.

A single CAD model view (previously known as a "CAD capture") consists of one or more selected CAD elements stored in that view. Similar to CAD Groups, you can use the **CAD Model Views** dialog box (**Edit | Graphic Display Window | CAD Model Views**) to define multiple CAD model views and what CAD to store in each view. You can also activate or deactivate the CAD views to show or hide the stored CAD elements within it. It essentially controls the visibility of CAD elements. However, it does more than CAD Groups, because the dialog box can also control the following:

- Visibility of CAD elements
- Visibility of CAD assembly components
- Optional view orientation
- Optional scale-to-fit operation
- Optional rendering mode (shaded/wireframe, surface edges on/off)

- Optional clipping plane

Many CAD systems provide the ability to add views directly into the CAD model. When you import those models, you can access those views from the **CAD Model Views** dialog box.



Portion of a CATIA Model Showing CAD Model Views (the GD&T or other text-like CAD elements)

Creating CAD Views from the CAD Models View Dialog Box

This procedure assumes you haven't created any views yet.

1. Click **Create** to add **New Model View 1** into the list of views.
2. From the list, select **New Model View 1**.
3. From the Graphic Display window, ensure you have a visible part model.
4. You can use the **Editing** area to mark or clear the check boxes. These control whether or not the software saves the view orientation, scale to fit, shading, and clipping plane into the selected view.
5. From the part model, click on one or more CAD elements to highlight them.
6. Click **Add** to add the selected CAD elements to the selected view.
7. If you want to rename the view to something else to better reflect its the stored CAD elements, click **Rename** and type a new name.
8. Continue creating CAD model views as needed. The next time you click **Create**, the software increments the number in the newly added view (for example, **New Model 2**).

Activating CAD Views from the CAD Model Views Dialog Box

From the list of created views, double-click on a CAD view to activate that view. When you activate a view, the software does the following:

- It highlights the view in the list and shows CAD elements that are in the activated view in the Graphic Display window.
- It updates the visible CAD assembly components to the state stored in the view.
- Based on the selected options, it can change the view orientation, perform a scale-to-fit operation, change the shading, and enable a clipping plane.
- It temporarily hides all other CAD elements and CAD assembly components.

You can also activate a CAD view from the list on the **CAD Setup** toolbar. For information, see the "CAD Setup Toolbar" topic in the "Using Toolbars" chapter.

About the CAD Model Views Dialog Box

List - The list shows all the views in the CAD model. To activate a view, double-click on the view in the list. The activated view has a bold font face. To deactivate a view, double-click on the activated item in the list. If a CAD model view contains embedded CAD GD&T callouts, this icon ( ) appears next to that item in the list. For information on CAD GD&T callouts, see the "Working with CAD GD&T Callouts" topic.

Create - This button creates a new CAD model view and adds any visible CAD GD&T elements into it.

Rename - This button renames the selected view name.

Delete - This button deletes the selected view. It does not delete any CAD elements.

Add - This button adds selected CAD elements to the selected view.

Subtract - This button removes the currently selected CAD elements from the selected view. It does not delete any CAD objects.

Select - This button selects all CAD elements contained in the selected view.

Deselect All - This button deselects all selected CAD objects.

Selection filter - The check boxes in this area filter selected CAD elements when you box-select them so that the software only selects the selected object types.

View orientation - If marked, this check box stores the current view orientation into the selected view. If cleared, and you activate the model view, the orientation view does not change. If you change the orientation and then mark this check box again, it uses the new orientation.

Scale to fit - If marked, this check box performs a scale-to-fit operation when you activate the view.

Shading - If marked, this check box stores the current shading (solid or wireframe) and the current surface edges state (on or off) into the selected view. When you later activate the view, it uses the stored shading regardless of the current Graphic Display window's shading. If you clear this check box and you activate the view, the shading uses whatever the Graphic Display window uses. To change the stored shading state, you can clear and mark this check box again to tell the view to use the current Graphic Display window shading.

Clipping plane - If marked, this check box stores the current clipping plane into the selected view. If you import a model that has a clipping plane defined in its view, that clipping plane temporarily exists in the model for that view. If you then clear the **Clipping plane** check box, the clipping plane is permanently removed. This check box remains disabled until you define a clipping plane. For information on creating and using a clipping plane, see the "Clip Planes Tab" topic.

Update from assembly - This button updates the selected view with the assembly view defined in the **CAD Assembly** dialog box. For information on manipulating the assembly view in that dialog box see, "Working with Assemblies of Parts".

Managing and Positioning On-Screen Elements

The Graphic Display window holds more than just the CAD drawing of your part. It can also display feature ID labels, Datum Definition labels, Dimension Info text boxes, and Point Info text boxes. All these elements bring organization to your report. However, if you have too many of them displayed, they may cover or clutter your part drawing.

Fortunately, you can move your mouse pointer over an element to reposition it. When your mouse pointer changes to a cross-hair, click on the element and drag it to a new location. PC-DMIS then draws a leader-line from the ID label or text box to its corresponding feature.

You can also have PC-DMIS dynamically reposition all your labels and text boxes around your part drawing even when you change the zoom level for your part's display. Select the **Automatic label positioning** check box from the **General** tab of the **Setup Options** dialog box. See "Automatic Label Positioning" in "Setting Your Preferences".

If things still look cluttered, you can also control the visibility state of these various elements using shortcut menus. See "Feature Shortcut Menu" and "Box-Select Shortcut Menu" in "Using Shortcut Keys and Shortcut Menus".

You can only position elements in the active view. If you have a split screen showing additional views of the part, the IDs remain as before in the other views.

Switching Between Curve and Surface Modes

From the **Operation | Graphic Display Window | Change Curve/Surface Mode** menu, you can choose between Curve Mode or Surface Mode. These make a models curves (the wireframe model) or its surfaces visible and available for selection.

From the **Graphic Modes** toolbar, you can also use the **Curve Mode** and **Surface Mode** icons with the **Program Mode** icon to take offline hits on your CAD model's wireframe or surface entities.

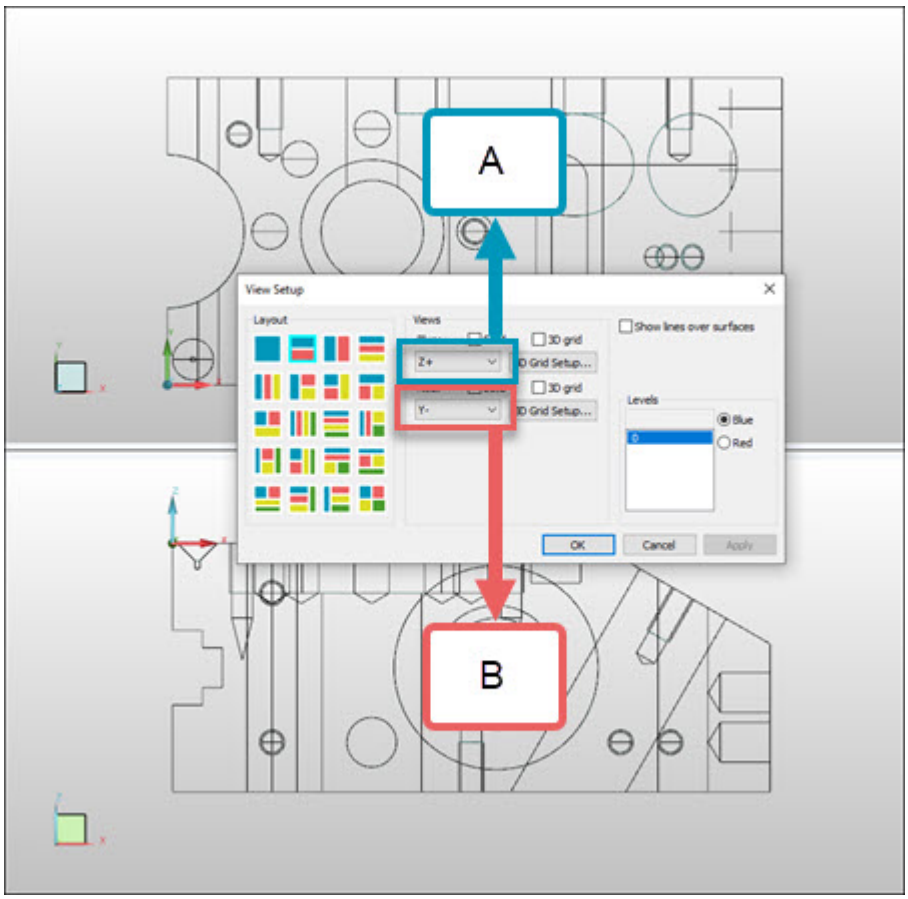
You must have the optional package called "Curves and Surfaces" purchased for your system in order to access these modes.

Curve Mode

The **Curve Mode** option makes a wireframe's model of curves and lines selectable when you click on the CAD data in the Graphic Display window. You must import a wireframe model for this option to become available. This mode works well for all measured features.

You need two graphic views to take hits in Curve Mode. You can set up multiple graphic views from the **View Setup** dialog box. For information on how to do this, see "Setting Up the Screen View".

The picture below illustrates this feature. The top half of the screen (A) shows the model in the Z+ view. The bottom half of the screen (B) shows the model in the Y- view. If your workplane is the Z+ plane, you can then use top half of the screen (A) to take hits, and use the bottom half of the screen (B) to set the probe's depth.



Sample Graphic Display window split to show two different wireframe views of the part

The following list references the above screen as it discusses the Curve Mode mouse operations you can use to set your probe's depth and take hits to insert measured features into your measurement routine. The information below assumes that Z+ is your workplane. You must also set PC-DMIS in Program Mode for these to work:

Operation: Right-click

Screen to Use: B

Description: This sets the approximate animated probe depth at the current mouse pointer position. Use this to create Move Points when you create measurement routines in Offline Mode. See "Inserting a Move Point Command" in "Inserting Move Commands".

Operation: Right-click + drag

Screen to Use: B

Description: This sets the depth for your next inserted hits on the nearest wireframe entity when you release the mouse button. Use this to set the depth for measured lines, circles, and cylinders.

Operation: Left-click

Screen to Use: A

Description: This selects the nearest line or circle and takes equally-spaced hits at the current depth setting based on the settings in the **General** tab of the **Setup Options** dialog box. See "Other Edit Boxes for the General tab" in "Setting Your Preferences".

Operation: Left-click + drag

Screen to Use: A

Description: This takes a single hit at the current depth setting on a line, arc, or circle. The hit direction depends on which side of the wire you release your mouse. Always approach from the side of the wire the machine would approach from.

Operation: Left-click + hold and release

Screen to Use: A

Description: This delayed click takes a hit at the pointer's position at the current depth setting. Use this method to take hits on a plane. Be sure to hold the mouse steady when you hold down the button so that when you release it, a valid hit is taken normal to the surface at the pointer's position.

For example, to define a Measured Cylinder feature in the Z+ view in **Curve Mode** follow these steps, and see the movie below:

1. In screen B (in the Y- view), right-click drag, to set the depth for the first level of hits.
2. In screen A (in the Z+ view), left-click on the cylinder to insert four hits for the first level.
3. In screen B, right-click + drag again, to set the depth for the second level of hits.
4. In screen A , left-click again on the curve to insert four more hits. You now have eight hits in two sets, with each set at different depths.
5. Press the End key to create the Measured Cylinder feature.

Surface Mode

The **Surface Mode** option makes a solid model's surfaces selectable when you click on the CAD data in the Graphic Display window. You must import a solid model for this option to become available. To take a hit, click on any surface. PC-DMIS pierces the surface and captures the X, Y, Z, I, J, and K information; the hit is recorded at the pointer's position. Select the correct number of hits to define the feature and then press the End key. PC-DMIS guesses the feature type. This mode works best for creating point, line, and

plane features. While you can also use this mode with circular features (circles, cylinders, cones, and spheres), it may be difficult to select hits at a constant cross section or depth, especially for inside features. In these cases, you may want to use Curve Mode.

For information on using these modes with scans, see the "Scanning Your Part" chapter.

Drawing Surfaces

The **Operation | Graphic Display Window | Draw Surfaces** option allows you to display surfaces on the screen. Note that you must first select the **Solid** check box in the **View Setup** dialog box. For information on the **View Setup** dialog box, see the "Setting Up the Screen View" topic.

To turn off the display of surfaces, select this option again.

Highlighting CAD Elements

You can hold down Shift key and move your mouse pointer over your part to see a highlight of the different CAD elements in your imported part model.

To create QuickFeatures from CAD elements, hold down the Shift key, hover over the CAD element and then click on it. This only functions with contact Auto features. For more information, see "Creating QuickFeatures by Hovering Over CAD Elements" in the "Creating Auto Features" chapter.

To change this mouse-over highlight color, see "Highlight" under "Changing Screen Colors" in this chapter.

Displaying Mouse Coordinates in Screen Counter Text

If you mark either the **Show CAD** or **Show Part** check boxes in the **Probe Readouts Setup** dialog box, PC-DMIS provides a continuous update of the location of your mouse pointer's location in the Probe Readouts window as it moves over the part model. If you move the mouse cursor away from the part over an empty space in the Graphic Display window, PC-DMIS displays its last known location when it was over the part.

In previous versions, this information was displayed in the upper right-hand corner of the Graphic Display window. In this new consolidated design, it now appears displayed in the Probe Readouts window.

For related information, see these topics:

- "Setting Up the Probe Readouts Window" topic in the "Setting your Preferences" chapter.
- "Using the Probe Readouts Window" topic in the "Using Other Windows, Editors, and Tools" chapter.

Repainting the Screen

Select the **Operation | Graphic Display Window | Repaint the Screen** option to redraw the Graphic Display window.

Changing Screen Modes

PC-DMIS allows you to switch between these Screen Modes. The different modes tell PC-DMIS how to interpret mouse clicks.

Using Translate Mode

The **Operation | Graphic Display Window | Change Screen Mode | Translate** option places PC-DMIS into Translate Mode. This mode allows you to move and zoom in or out on the part while in the Graphic Display window. The following functions are available in translate mode:

- Shrink the size of the part within the Graphic Display window.
- Enlarge the size of the part within the Graphic Display window.
- Select a portion of the part within the Graphic Display window.
- Reposition the part within the Graphic Display window.

Shrinking the Part Image within the Graphic Display Window

To shrink (or zoom out of) the entire image of the part, rotate your mouse's wheel button away from you. PC-DMIS centers the zoom-out operation at the pointer's location.

Enlarging the Part Image within the Graphic Display Window

To enlarge (or zoom in on) the entire image of the part, rotate your mouse wheel button towards you. PC-DMIS centers the zoom-in operation at the pointer's location.

Enlarging a Portion of the Part Image within the Graphic Display Window

To enlarge (or zoom in on) a specified section of the part:

1. Place the pointer at a corner of the selection area.
2. Drag both the right and left buttons simultaneously over a portion of the Graphic Display window.
3. When the box contains the proper geometry, release both buttons to have PC-DMIS zoom in on the selected area.

Once you reach a certain point, the image doesn't enlarge any further.

Repositioning the Part Image within the Graphic Display Window

To change the position of the part image within the Graphic Display window:

1. Position the pointer over the part image.
2. Drag the pointer to a new position and then release it.

Using 2D Rotate Mode

You can select **Operation | Graphic Display Window | Change Screen Mode | 2D Rotate** to rotate the part in two dimensions. You can rotate the part a full 360 degrees, or any fraction of 360 degrees.

When you rotate, the software displays an icon to show the center of rotation. For more information see "Center of Rotation" in the "Rotating the Drawing" topic.

The following sections describe the different ways to rotate a drawing in two dimensions.

Rotating a drawing only changes the display. It does not change the origin or datums of the part.

Rotating 2D by Dragging

To drag the pointer and rotate around the imaginary center of the current view:

1. From the **Graphics Modes** toolbar, click the **2D Rotate Mode** icon.
2. Move the pointer in the Graphic Display window (in any area other than at the center).
3. Right-click and drag the pointer about the center of the view to redraw the image dynamically.
4. Release the button to keep the current rotation.

To rotate your part from within any mode, press Alt + right-click and drag the mouse. This does not work if you have the **Rotate** dialog box for 3D rotation open.

Rotating 2D to an Element (Square up the Drawing)

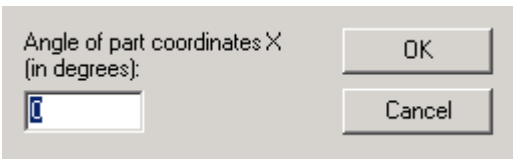
To "square-up" the part to the screen, from the **Graphic Modes** toolbar, select the **2D Rotate Mode** icon, and then right-click on a CAD element. PC-DMIS rotates the drawing so that the selected element is parallel to the closest screen axis (vertical or horizontal).

You can square-up the drawing to the closest X, Y, or Z axis at any time, even if you don't have the **Rotate** dialog box open. Click twice on the Graphic Display window with your mouse wheel button.

Rotating 2D by a Factor

To rotate by a precise factor, similar to the "Scaling the Model by a Factor" topic, do the following:

1. From the **Graphic Modes** toolbar, select the **2D Rotate Mode** icon .
2. In the Graphic Display window, press Shift and then right-click to show a small dialog box.

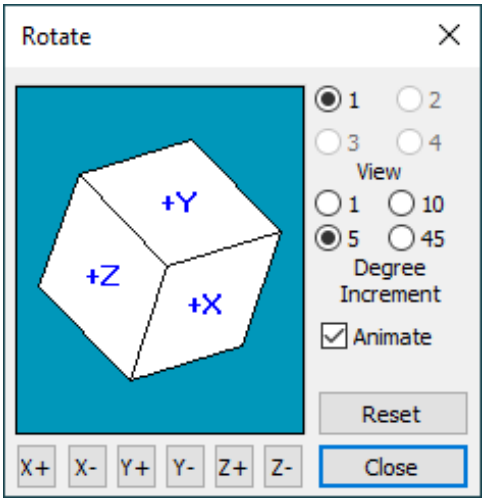


3. Type the angle (in degrees) into the **Angle of part coordinates X** box.

4. Click **OK** to close the box and perform the 2D rotate of the part in the Graphic Display window.

Using 3D Rotate Mode

You can select **Operation | Graphic Display Window | Change Screen Mode | 3D Rotate** to display the **Rotate** dialog box. With this dialog box, you can rotate a part drawing in three dimensions, up to 45 degrees per rotation.



Rotate Dialog Box

When you rotate, the software displays an icon to show the center of rotation. For more information see "Center of Rotation" in the "Rotating the Drawing" topic.

The "Rotate 3D using Rotate Dialog Box", "Rotate 3D by Dragging", and "Rotate 3D to an Element" topics below describe three different ways to rotate a drawing in three dimensions.

For additional information on how to rotate a drawing, see "Rotating the Drawing".

Rotate 3D by Dragging

You can drag the pointer to rotate the part:

1. From the **Graphic Modes** toolbar, selecting the **3D Rotate Mode** icon to open the **3D Rotate** dialog box.

2. Right-click and drag the pointer.

To rotate your part with or without the **Rotate** dialog box open, press Ctrl + right-click, and drag the pointer. Also, you can do this if you hold the wheel button and drag the pointer.

Rotate 3D using Rotate Dialog Box

To rotate the drawing using the dialog box, do the following:

1. From the **Graphic Modes** toolbar, select the **3D Rotate Mode** button ( ) to open the **3D Rotate** dialog box.
2. From the **3D Rotate** dialog box, select the view to be altered (numbered as 1-4 option buttons).
3. Select the **Degree Increment** (1, 5, 10, or 45) option to determine the rotation amount in degrees.
4. Click on one of the axis buttons (**X+**, **X-**, **Y+**, **Y-**, **Z+**, or **Z-**) to rotate the drawing by the defined increment around the selected axis.
5. Click **Close** to display the changes to the CAD image in the Graphic Display window.

Reset - This button snaps the rotation of the drawing in the **Rotate** dialog box and in the Graphic Display window to the closest axis.

Animate - If you select this check box, PC-DMIS dynamically re-draws the part in the Graphic Display window as it is rotated inside the dialog box. If you clear this check box, and then use a rotation button in the dialog box, the rotation of the actual part on the screen does not take place until you release the clicked button.

Rotate 3D to an Element (Square up the Drawing)

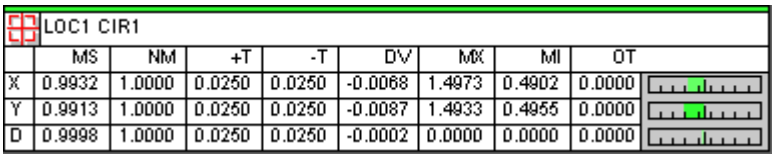
To "square-up " the part to the screen,

1. From the **Graphic Modes** toolbar, select the **3D Rotate Mode** icon
2. Right-click on a CAD element. PC-DMIS rotates the drawing and makes the selected feature parallel to the closest screen axis (vertical, horizontal, or perpendicular to the screen).

You can square-up the drawing to the closest X, Y, or Z axis at any time, even if you don't have the **Rotate** dialog box open. Click twice on the Graphic Display window with your mouse wheel button.

Text Box Mode

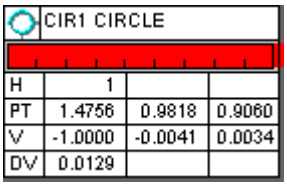
This button places PC-DMIS into a mode that lets you manipulate feature IDs, scan points, and CAD GD&T callouts. You can also use this mode to quickly create and modify Dimension Info and Point Info text boxes.



LOC1 CIR1

	MS	NM	+T	-T	DV	MX	MI	OT
X	0.9932	1.0000	0.0250	0.0250	-0.0068	1.4973	0.4902	0.0000
Y	0.9913	1.0000	0.0250	0.0250	-0.0087	1.4933	0.4955	0.0000
D	0.9998	1.0000	0.0250	0.0250	-0.0002	0.0000	0.0000	0.0000

Sample Dimension Info Box



CIR1 CIRCLE

H	1		
PT	1.4756	0.9818	0.9060
V	-1.0000	-0.0041	0.0034
DV	0.0129		

Sample Point Info Box

To create these text boxes, use the shortcut menus that appear when you right-click or box-select feature IDs or features. For more information, see the "Inserting Dimension Info Boxes" and "Inserting Point Info Boxes" topics in the "Inserting Report Commands" chapter.

The following shortcut menus are available inside Text Box Mode:

- Right-click on the feature, feature's ID label, or text box to display a shortcut menu for that feature. For more information, see "Feature Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.
- Right-click on a Dimension info or Point Info box to display a shortcut menu for that text box. For more information, see "Feature Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.
- Right-click on a CAD GD&T callout to display a shortcut menu for that callout. For more information, see "Feature Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.
- Box-select one or more features to display a shortcut menu. For more information, see "Box-Select Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.
- Right-click on a scan point and use the **Move Cursor To** menu to jump to that point. For more information, see the "Feature Shortcut Menu" topic in the "Using Shortcut Keys and Shortcut Menus" chapter.

When your pointer isn't over a feature or text box, you can perform the normal Translate Mode operations to zoom and rotate the part with the right mouse button. For additional information, see "Translate Mode".

Tips

- You can create a DIMINFO or a POINTINFO box regardless of the current mode with the mouse. Double-click on a feature or feature ID label. PC-DMIS automatically creates an info box based on current **Edit Dimension Info** or **Edit Point Info** dialog box settings. For example, suppose you created a Dimension Info box for a feature. If you double-click subsequent feature labels, the software creates additional Dimension Info boxes for those features. It uses the same settings selected the last time someone used the **Edit Dimension Info** dialog box.
- You can use your pointer to identify the feature associated with a particular label. From the Graphic Display window, move your pointer over either a feature or a feature's ID label. PC-DMIS highlights the feature on the CAD drawing.

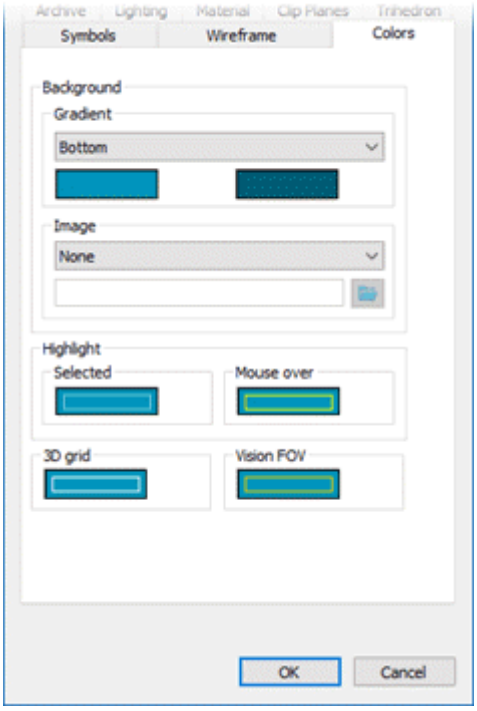
Program Mode

Select **Operation | Graphic Display Window | Change Screen Mode | Program** to learn and edit measurement routines using CAD data from IGES files.

You can use Program Mode to create measured features from a wireframe or surface CAD model. While in Program Mode, you take hits on the model with an animated probe simulating taking hits with the CMM in Guess Mode. This capability is available in either the Offline or Online mode of PC-DMIS.

For more information on how to use graphics to create measurement routines, refer to the "Working in Offline Mode" appendix.

Changing Screen Colors



CAD and Graphic Setup Dialog Box - Colors Tab

The **Colors** tab of the **CAD and Graphic Setup** dialog box determines the display settings of the Graphic Display window.

These display settings include:

- Background color
- Background Gradient color
- Background Image
- Highlight colors for feature selection and mouse-over
- 3D Grid color
- The Vision module's Field of View (FOV) color

If you want to change the color of components in your part model, use the **CAD Assembly** dialog box. For more information, see "Working with Assemblies of Parts".

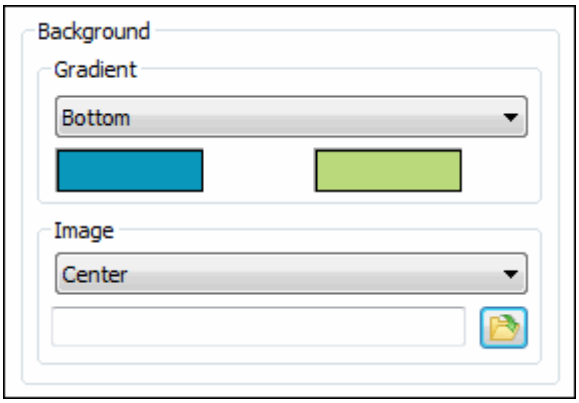
To Change a Color

To change the colors:

1. Access the **Colors** tab on the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | Screen Color**).

2. Under **Background**, **Gradient**, **Highlight**, **Mouse Over Highlight**, **3D Grid**, **Vision FOV**, or **CAD GD&T**, click on the rectangular color box to open the **Color** dialog box.
3. Choose a new color for the option you selected. In most cases, PC-DMIS immediately displays the selected color change and provides a preview of the change.
4. Click **OK**. The **Color** dialog box closes and returns to the **Screen Color** dialog box.
5. If you changed a **CAD GD&T** color, you need to click the **Apply** button next to it to preview it.
6. When finished, click the **OK** button on the **CAD and Graphic Setup** dialog box to accept your changes and close the dialog box.

Background

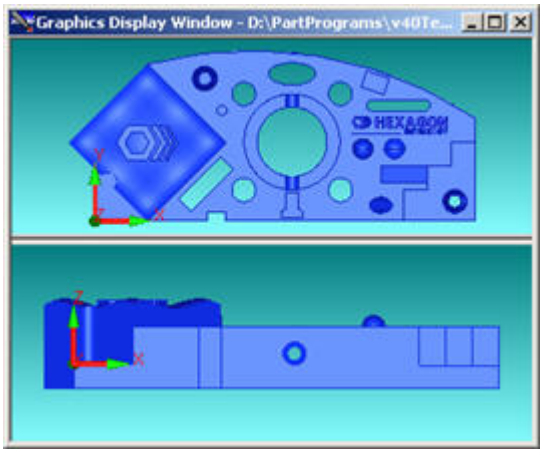


The **Background** area allows you to change the background screen color or set an image as the background. PC-DMIS also uses the color or image as the background for any Geometric Tolerances displayed in the Graphic Display window. Follow the directions discussed in "To Change a Color" to set the background color.

Setting the Gradient Background Color

The **Gradient** area lets you define a secondary background gradient color. When you set the gradient to something other than **None** in the list, a rectangular color box appears. You can click that box to define a secondary gradient color. Then, when setting the screen's background, PC-DMIS starts from the setting location (for example, the Bottom) and displays the secondary gradient color, and then gradually changes from that color to the primary gradient color as it draws the colors near the other end of the screen.

For example, a **Bottom** gradient setting might look like this:



Example that shows gradient colors.

The available items in the **Gradient** list are:

- **None** - No secondary gradient. The screen's background will be the solid primary color.
- **Bottom** - The secondary gradient color starts at the bottom of the screen and gradually changes to the primary color as it nears the top.
- **Right** - The secondary gradient color starts at the right side of the screen and gradually changes to the primary color as it nears the left side.
- **Top left** - The secondary gradient color starts at the top-left side of the screen and gradually changes to the primary color as it nears the bottom right side.
- **Top right** - The secondary gradient color starts at the top right side of the screen and gradually changes to the primary color as it nears the bottom-left side.
- **Bottom Left** - The secondary gradient color starts at the bottom-left side of the screen and gradually changes to the primary color as it nears the top-right side.
- **Bottom Right** - The secondary gradient color starts at the bottom-right side of the screen and gradually changes to the primary color as it nears the top-left side.

Setting an Image as the Background

The **Image** settings area sets an image as the background. To do this, from the **Image Mode** drop-down list, set the image to something other than **None**. This enables the **File Navigation** text box and the **Open File** button . Then, do one of the following:

- Manually enter the path to the image file in the **File Navigation** text box then click **OK** on the **CAD and Graphic Setup** dialog box.

  or:

- Click **Open File** to display the **Open** dialog box, navigate to the image file and then select it. Click **Open** from the **Open** dialog box, and then click **OK** on the **CAD and Graphic Setup** dialog box.

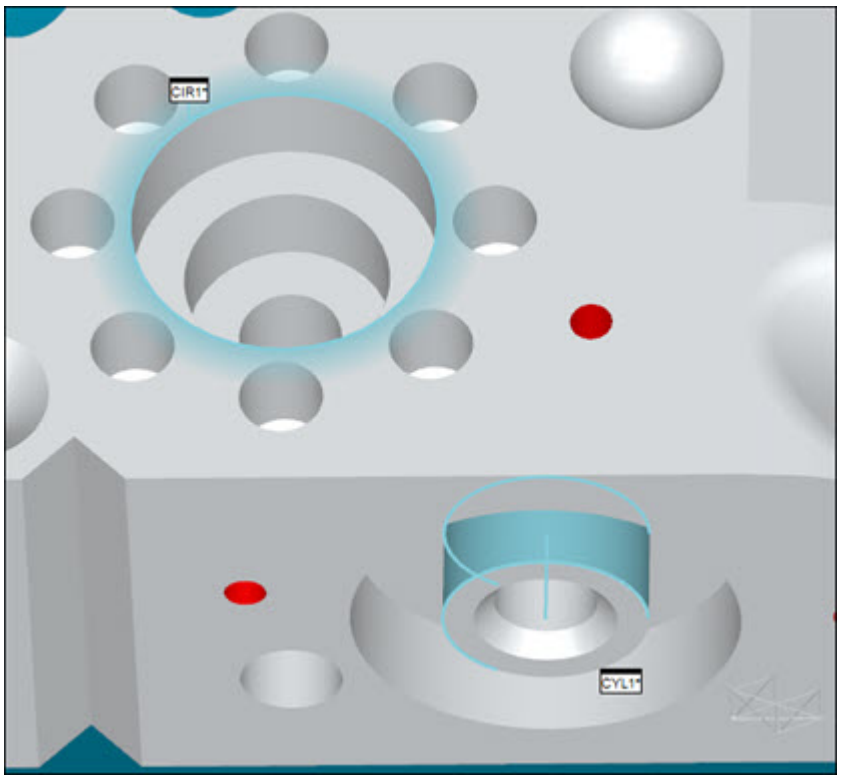
The background image is drawn based on the image mode selected:

- **None** - No background image is displayed.
- **Center** - The image is centered in the Graphic Display window and is not scaled.
- **Top left** - The image is positioned at the top-left of the Graphic Display window and is not scaled.
- **Top right** - The image is positioned at the top-right of the Graphic Display window and is not scaled.
- **Bottom left** - The image is positioned at the bottom-left of the Graphic Display window and is not scaled.
- **Bottom right** - The image is positioned at the bottom-right of the Graphic Display window and is not scaled.
- **Fit** - The image is scaled to fit just inside the Graphic Display window, preserving the image's aspect ratio.
- **Stretch** - The image is scaled to fill the entire Graphic Display window, not preserving the image's aspect ratio.
- **Fill** - The image is scaled to fill the entire Graphic Display window, preserving the image's aspect ratio.
- **Tile** - The image is drawn repeatedly over the entire Graphic Display window and is not scaled.
- **Tile mirrored** - The image is drawn repeatedly over the entire Graphic Display window with every other image being mirrored. The images are not scaled.

Highlight

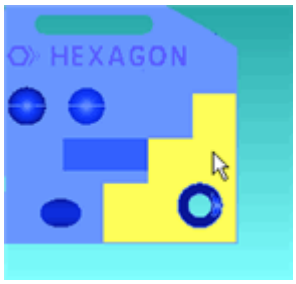
The **Highlight** area allows you to change the color PC-DMIS uses for items that you select or when you hover your pointer over surfaces or curves. To change the color, follow the directions discussed in the "To Change a Color" topic.

Selected - This option controls the color that PC-DMIS uses when you select geometric features from the CAD model. You typically need to select features from the CAD model for operations like constructions and dimensions. PC-DMIS also automatically selects features one by one as it executes them.



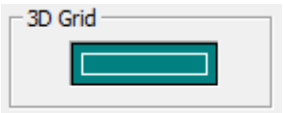
A selected circle (2D feature) and a selected cylinder (3D feature)

Mouse over - This option controls the highlight color that PC-DMIS uses when you move the pointer over geometric features on the CAD model. PC-DMIS typically uses this color when you create QuickFeatures. (For details on how to create QuickFeatures, see "Creating QuickFeatures" in the "Creating Auto Features" chapter.) PC-DMIS also uses this color for the line you drag to create a Quick Scan. (For information on the Quick Scan functionality, see "Creating Quick Scans" in the "Scanning" chapter of the PC-DMIS CMM documentation.)



The default mouse-over color during a QuickFeature operation

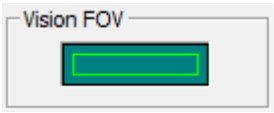
3D Grid



The **3D Grid** area allows you to change the color used for the 3D grid in the Graphic Display window. Follow the directions discussed in "To Change a Color".

For more information on the 3D Grid, see "Adding a 3D Grid".

Vision FOV

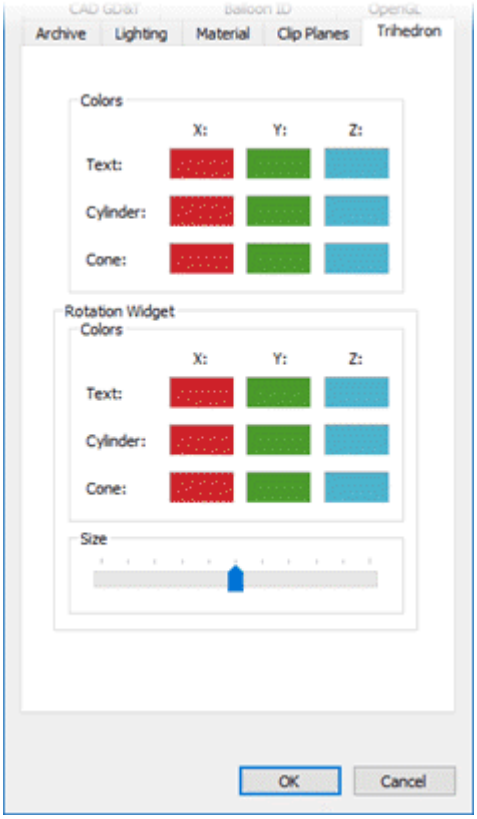


The **Vision FOV** area lets you change the color used for the Field of View (FOV) when using the PC-DMIS Vision module. Follow the directions discussed in the "To Change a Color" topic. For information on the FOV and the Vision module, see the PC-DMIS Vision documentation.

Changing Trihedron Settings

A trihedron is an XYZ symbol in the Graphic Display window that shows the intersection of three planes. It defines the coordinate system's origin, and shows the current orientation when you rotate the part.

The **Edit | Graphic Display Window | Trihedron** menu item displays the **Trihedron** tab of the **CAD and Graphic Setup** dialog box. You can use this tab to change the colors for the different components that make up the trihedron and to size the rotation widget.



CAD and Graphic Setup Dialog Box - Trihedron Tab

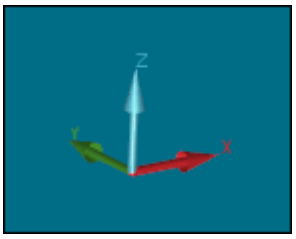
You can change colors for these trihedrons with these areas:

- **Colors** - This affects the standard part alignment trihedron.

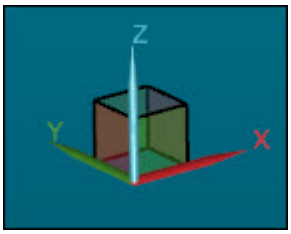
- **Rotation Widget Colors** - This affects the rotation widget trihedron. For more information on this widget, see "Using the Rotation Widget".

Original Colors

The original colors for the part alignment symbol and the rotation widget are red (X), green (Y), and blue (Z):



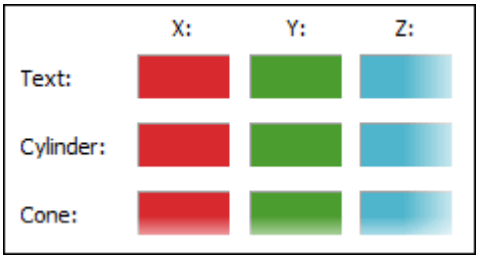
Example showing the original colors of the part alignment trihedron.



Example showing the original colors of the rotation widget.

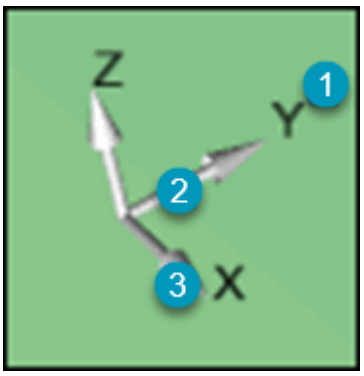
Colors area

The **Colors** area contains a grid of colored boxes.



These grids of boxes represent the different components for each of the three axes of the two trihedrons.

- **Text** - This sets the colors of the "X", "Y", or "Z" letters (or text) displayed on the trihedrons (1 in image below).
- **Cylinder** - This sets the color of the X, Y, or Z axis line (or cylinder) on the trihedrons (2 in image below).
- **Cone** - This represents the color of the X, Y, or Z arrow head (or cone) on the trihedrons (3 in image below).



You can click on a colored box to change that component's color.

Rotation Widget area

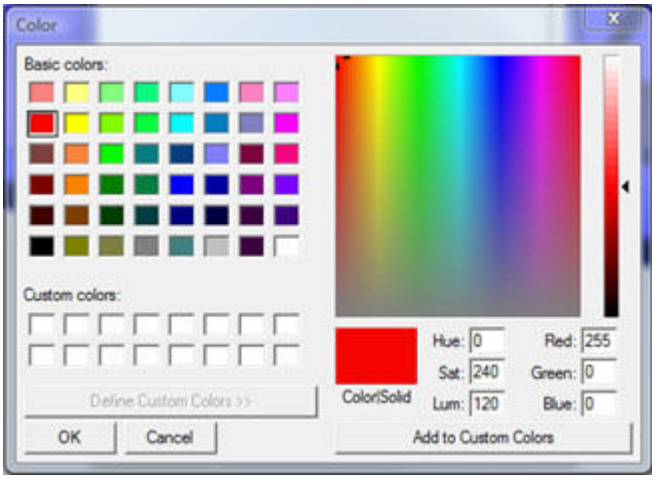
The colored boxes in this area work the same way as the colored boxes in the **Colors** area above, except that they affect the rotation widget.

The **Size** slider changes the relative size of the rotation widget.

> You can temporarily change the size with the mouse pointer. Hover your pointer over a face on the rotation widget until it highlights that face. Then rotate the mouse wheel. The size change remains until you close the measurement routine. To make the size change permanent, first open the **Trihedron** tab, resize the widget, and then click **OK**.

Changing Colors

You can change these colors by clicking on one of the colored boxes and choosing a new color from the **Color** dialog box:



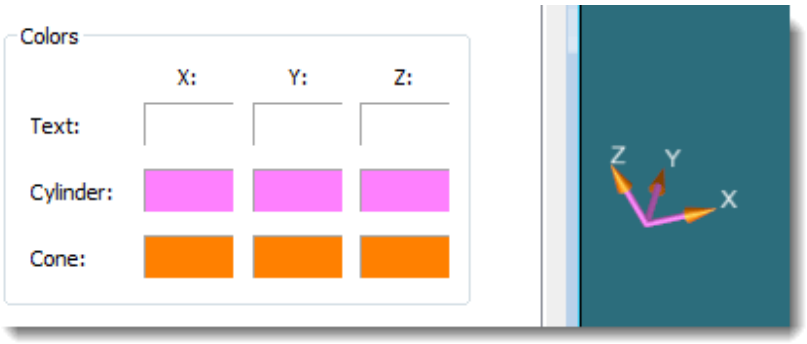
Color Dialog Box

As soon as you click **OK**, the **Color** dialog box closes, and PC-DMIS updates the trihedron color in the Graphic Display window.

When you click **OK** on the **CAD and Graphic Setup** dialog box, PC-DMIS stores the color information in the JSON file. All measurement routines then use these colors for that trihedron.

Example

This example shows a modified alignment trihedron. If you change its **Text** colors to white, **Cylinder** colors to pink, and **Cone** colors to orange, the trihedron looks like this:



Example showing a modified trihedron.

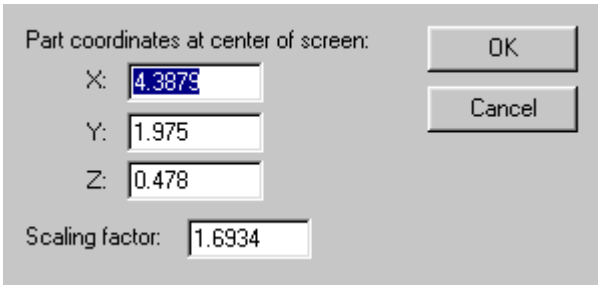
Scaling the Drawing

You can scale the CAD drawing to fit the size of your selected view(s) in the Graphic Display window, or you can scale by a factor. These options are helpful whenever the image becomes too large or too small.

Scaling the Model to Fit the View

The **Operation | Graphic Display Window | Scale to Fit** menu option re-displays the part image to fit entirely within the Graphic Display window. This option is helpful whenever the image becomes too large or too small. To alter the part image so all features and CAD elements are visible, select the **Scale to Fit** menu option.

Scaling the Model by a Factor



Scale Drawing Dialog Box

Scaling a drawing alters the part's image in the Graphic Display window. This option does *not* alter the CAD data in any way. The Scale Drawing option is *not* available from the menu bar, and it only functions when PC-DMIS is in Translate Mode.

To use this option:

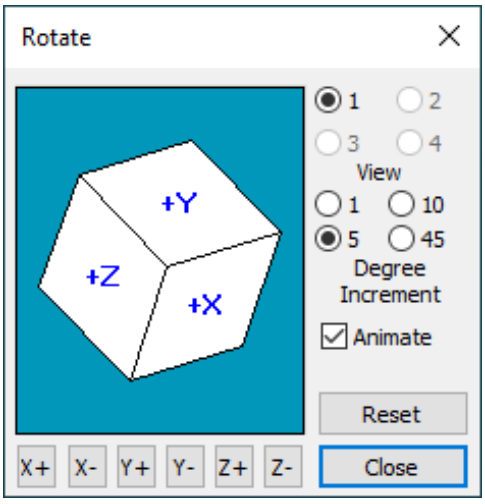
1. Click the **Translate** mode icon from the **Graphic Modes** toolbar. For more information on the screen modes, see "Changing Screen Modes".
2. Press Shift and right-click to open the **Scale Drawing** dialog box.
3. Type the **X**, **Y**, **Z** part coordinates to display in the center of the screen.
4. Type the desired value for the **Scaling factor**. This is an actual scale factor, not a relative scale factor.

A scale factor of 0.5 scales the part to half its actual size. A scale factor of 2.0 scales the part to twice its actual size. A scale factor of 1.0 scales the part to its actual size.

You can also scale the graphical image within the Graphic Display window by right-clicking above or below an imaginary horizontal line that splits the Graphic Display window.

Rotating the Drawing

Select **Operation | Graphic Display Window | Rotate** to display the **Rotate** dialog box.



Rotate Dialog Box

This dialog box allows you to rotate a part drawing in three dimensions. You can rotate up to 45 degrees per rotation for each view of the part. You can also activate this option from the **3D Rotate Mode** button ( ) on the **Graphic Modes** toolbar.

To rotate the drawing using the **Rotate** dialog box, do the following:

1. Select the view to be altered (1-4).

2. Determine the amount of rotation by selecting the degree increment (1, 5, 10, or 45).
3. Click on the desired axis button to rotate the drawing in the indicated direction.
4. An icon shows the point of rotation. For information, see "Center of Rotation Icon" below.

Click the **Reset** button to snap the rotation of the drawing in the **Rotate** dialog box and in the Graphic Display window to the closest axis.

If you select the **Animate** check box, PC-DMIS dynamically re-draws the part in the Graphic Display window as it is rotated.

For additional information on rotating, see "3D Rotate Mode".

To quickly rotate your part with or without the **Rotate** dialog box open, press Ctrl + right-click and drag the mouse. If your mouse has a center wheel button, hold the wheel button and drag the mouse. Also, you can double-click the wheel button at any time to perform the same action as the **Reset** button on the **Rotate** dialog box: it snaps the rotation to the closest axis.

Determining the Point of Rotation with a CAD Model

If you have a CAD model in the Graphic Display window, there are two ways to determine the point of rotation. Which way to use depends on where your pointer is when you first click the right mouse button. Either your pointer is over the Graphic Display window background, or it is over the part's CAD.

- If the pointer is *over the Graphic Display background*, the part rotates about the part's origin.
- If the pointer is *over the part*, the part rotates about the point on the part directly below the pointer.

Determining the Point of Rotation without a CAD Model

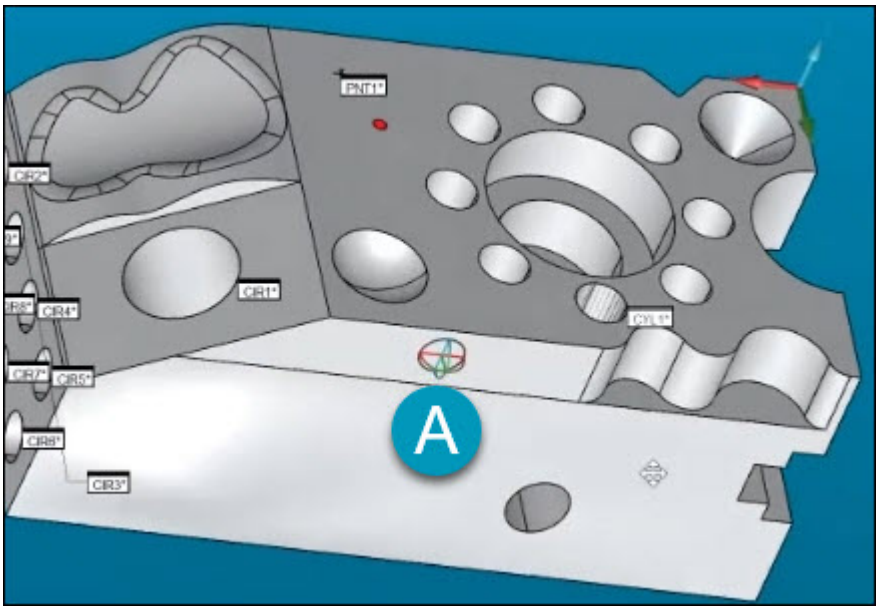
If you don't have a CAD model in the Graphic Display window, PC-DMIS can rotate about a user-defined point instead. This may be useful when there are relatively large distances between measured objects or CAD objects.

- If the **Rotate** dialog box is active, PC-DMIS treats the screen center position as if you clicked there.
- If PC-DMIS cannot find a pierced point, it instead uses the closest display object (pixel) and rotates about that point. If the closest object is a CAD object, then the point PC-DMIS uses is the closest point on the closest CAD object.

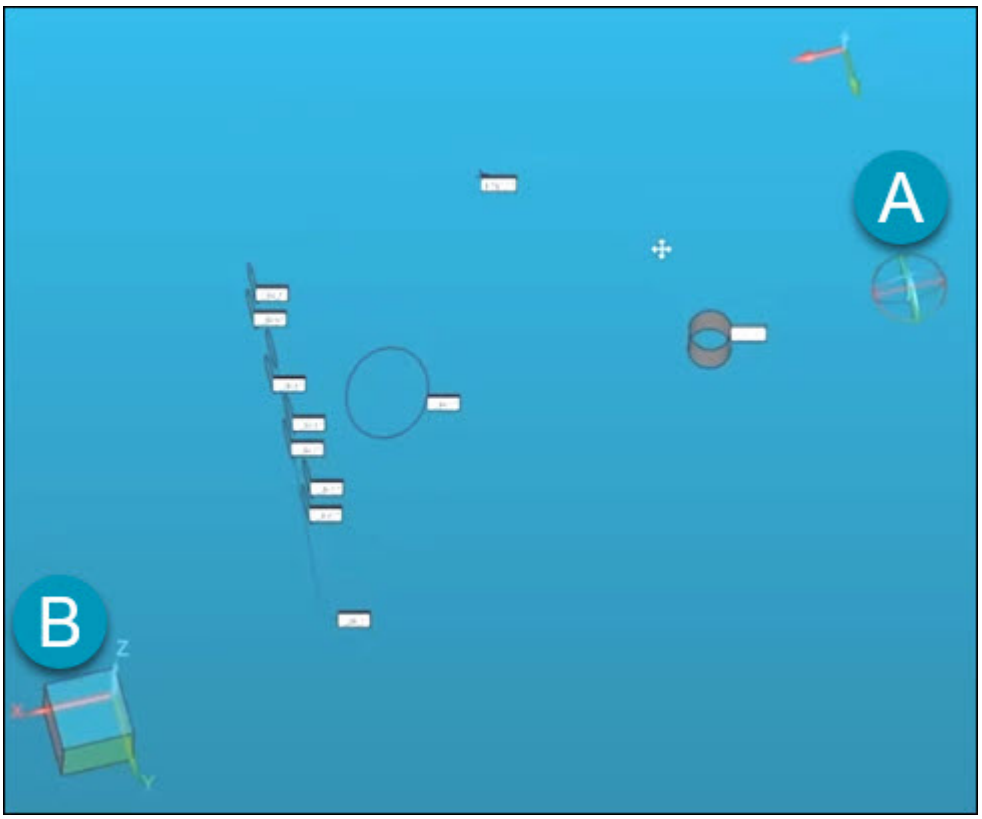
Center of Rotation Icon

A colored, spherical, wire-frame icon indicates the current center of rotation:

Center of rotation Icon



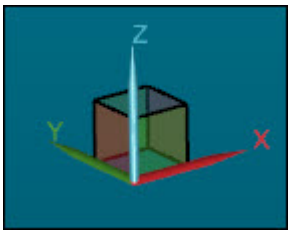
Graphic Display window with center of rotation icon (A) on the part.



Graphic Display window with center of rotation icon in space (A) and Rotation Widget (B).

- Like the Rotation Widget, the center of rotation icon uses the same colors as the trihedron. For information, see "Changing Trihedron Settings".
- The size of the center of rotation icon matches the size of the Rotation Widget. For information, see "Using the Rotation Widget".

Using the Rotation Widget



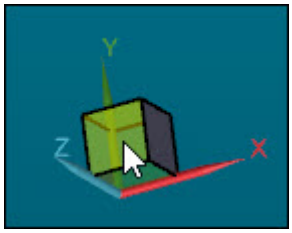
The Rotation Widget is another trihedron with a cube that you can show or hide in the bottom-left corner of each view of the Graphic Display window. The trihedron shows the current rotation of the CAD model. You can also use it to rotate and scale to fit the CAD model.

To show or hide this widget, use from the **Graphic Items** toolbar, use the **Show Rotation Widget** icon ( ). For more information on this toolbar, see "Graphic Items Toolbar" in the "Using Toolbars" chapter.

You can temporarily change the size with the mouse pointer. Hover your pointer over a face on the rotation widget until it highlights that face. Then rotate the mouse wheel. The size change remains until you close the measurement routine. To make the size change permanent, first open the **Trihedron** tab, resize the widget, and then click **OK**.

The Cube

The Rotation Widget contains a translucent colored cube. When you hover over a cube face, it highlights that face on the cube with the mouse-over color discussed in the "Highlight" topic.



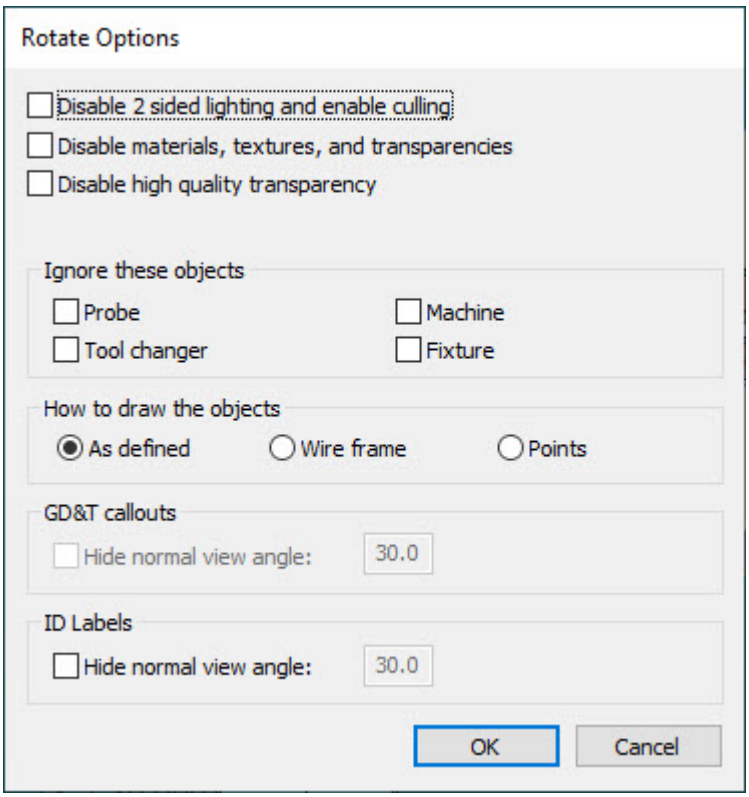
- You can click on any cube face to rotate the model to show that face.
- You can double-click on the cube face to rotate the model and scale the model to fit the window.

The default colors for this trihedron are the same as those that the alignment trihedron uses. You can change the colors on the **Trihedron** tab of the **CAD and Graphic Setup** dialog box. For information, see "Changing Trihedron Settings". For information, see "Trihedron tab".

Changing Rotation and Other Motion Options

You can use the **Rotate Options** dialog box to modify how PC-DMIS displays CAD models. You can access the dialog box by doing one of the following:

- From the **Edit** menu, point to **Graphic Display Window**, and choose **Rotate Options**.
- From the **Graphic Modes** toolbar, click the **Rotate Options** icon .



Rotate Options Dialog Box

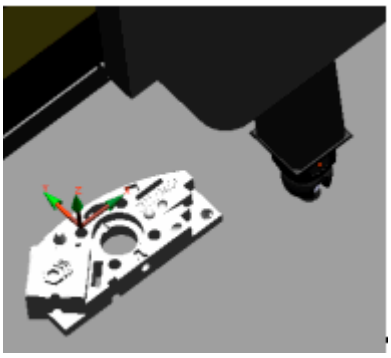
This dialog box controls whether or not certain items get drawn and how they get drawn inside the Graphic Display window when you rotate your part model. The options on this dialog box can help speed up the display of your rotation. PC-DMIS does not apply these settings globally to all measurement routines; instead, they are only applied to your current measurement routine. This allows you to set different rotation options for each measurement routine.

If you are using *3Dconnexion 3D Mouse* device, this dialog box behaves just like the **Pan, Zoom, Rotate Options**dialog box but with a different dialog box title. *3Dconnexion 3D Mouse* devices extend the functionality described for rotation to zooming and panning as well. For information, see the "Configuring 3Dconnexion 3D Mouse Devices" topic in the "Getting Started: An Overview" chapter.

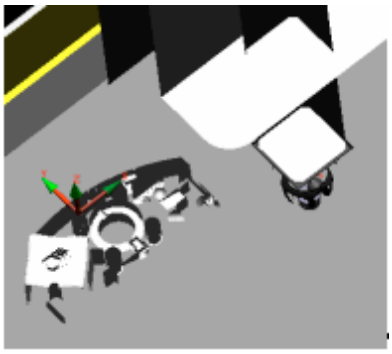
Changing Animation Speeds: If you want to modify your offline animation speeds, see the **Execution** area on the **Animation** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). In addition, see the "Executing and Debugging Measurement Routines Offline" topic in the "Working in Offline Mode" chapter.

Disable 2 Sided Lighting and Back Face Culling

Selecting this check box disables two sided lighting and uses back face culling during rotation. The resulting lighting effects will roughly display only half of the CAD elements. See the comparison below.



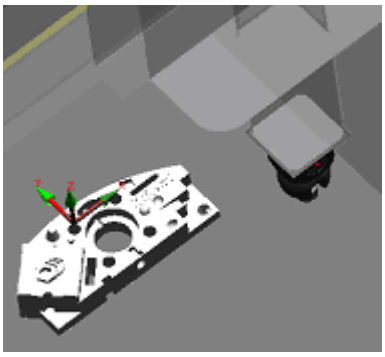
Before Rotation



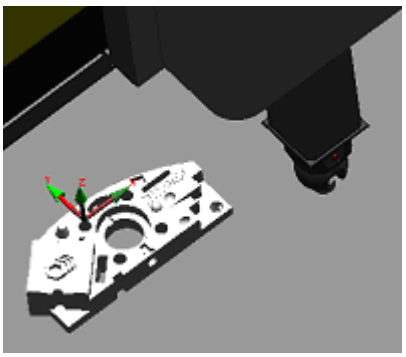
During Rotation – Disable 2 sided lighting and back face culling

Disable Materials, Textures, and Transparencies

Selecting this check box disables the rendering of applied materials, textures, and transparencies during rotation. See the comparison below.



Before Rotation with Transparency



During Rotation - No transparencies

Disable High Quality Transparency

If high-quality transparency is enabled, it will be temporarily disabled when the graphic is rotated, zoomed, or panned. This can be particularly helpful for very complex models that take a long time to render using high-quality transparency.

Animate Speed

You can set this value by using the slider or typing a decimal value. It applies to all measurement routines. This decimal value determines the relative speed of the animated rotation of the part in the Graphic Display window when you click one of the seven view buttons on the **Graphic View** toolbar (**View | Toolbars | Graphic View**).

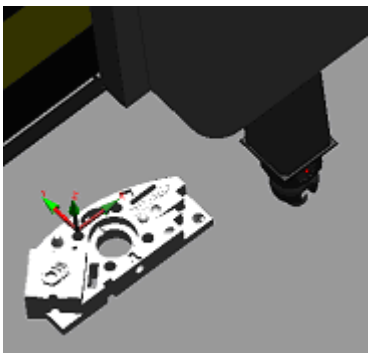
The default value is 1. This represents a normal rotation speed. Smaller values provide slower rotations, larger values produce quicker rotations.

The minimum allowed value is 0.1 and results in a very slow rotation.

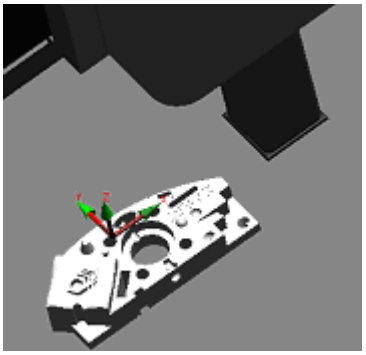
The maximum allowed value is 10 and results in an instantaneous rotation for view changes.

Ignore These Objects

Selecting this check box disables the display of specified objects. Objects will be ignored during rotation for those object types that are selected using the associated check boxes. You can choose to ignore probes, machines, changers, or fixtures. The example below hides the probe during rotation.



Before Rotation



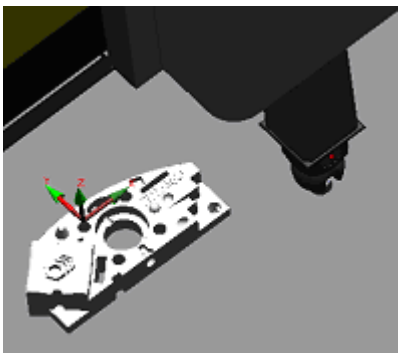
During Rotation - Ignore Probe

How to Draw Objects - As Defined

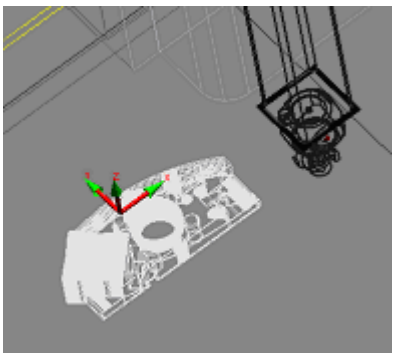
Selecting this option draws objects as originally defined during rotation.

How to Draw Objects - Wireframe

Selecting this option draws objects as wireframes during rotation. See the comparison below.



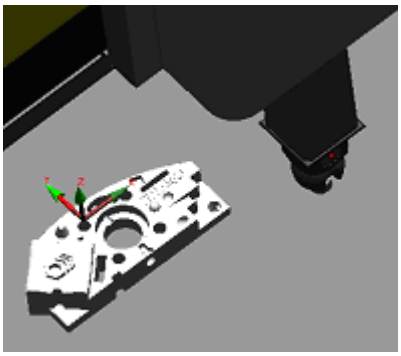
Before Rotation



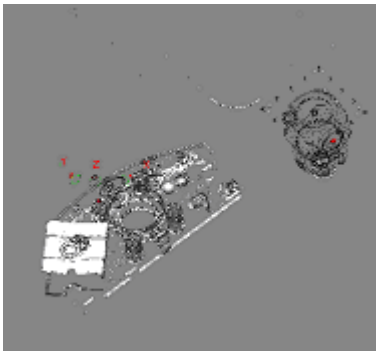
During Rotation - Draw Wireframe

How to Draw Objects - Points

Selecting this option draws objects as points. The **Points** option gives a representation of the objects using points. See the comparison below.



Before Rotation with Transparency



During Rotation – Draw Points

GD&T Callouts Area

The **GD&T callouts** area lets you control the display of GD&T callouts within an imported CAD model. It helps reduce screen clutter by making such callouts temporarily hidden as they reach or approach a view angle parallel to your current view angle. This option is disabled if your part model doesn't have any embedded GD&T callouts.

Do not confuse these GD&T callouts with the Geometric Tolerance dimensions that are created inside PC-DMIS. The GD&T callouts are essentially CAD objects themselves. These were not created inside PC-DMIS. Rather, they were created by the CAD software used to create the model and are stored with the model.

Hide normal view angle - This option tells PC-DMIS to hide the callouts when they cross a threshold of degrees parallel to your view angle. The box to the right of the check box lets you specify the threshold of degrees parallel to your view angle. The default value is 30.

Suppose you have a GD&T callout that points to a cylinder feature on the top of your part. Now, suppose you type a value of 45 degrees in the box. When you 3D rotate the part, the callout remains visible until the rotation of the part meets or exceeds 45 degrees.

If you want to always show the callouts, clear the check box.

ID Labels Area

The **ID Labels** area behaves the same as the GD&T callouts area, except that instead of GD&T callouts, it affects ID labels, such as feature ID labels, dimension info boxes, and point info boxes. It helps reduce screen clutter by making these labels temporarily hidden as they reach or approach a view angle parallel to your current view angle.

For more information on the items in this area, see the "GD&T Callouts Area" topic.

Showing and Hiding Graphics

You can choose to show or hide various labels and graphical objects inside the Graphic Display window with the **Graphic Items** toolbar (**View | Toolbars | Graphic Items**). Hiding some of these objects from time to time may de-clutter the Graphic Display window so that it's easier to perform certain operations.

These are the items you can show or hide:

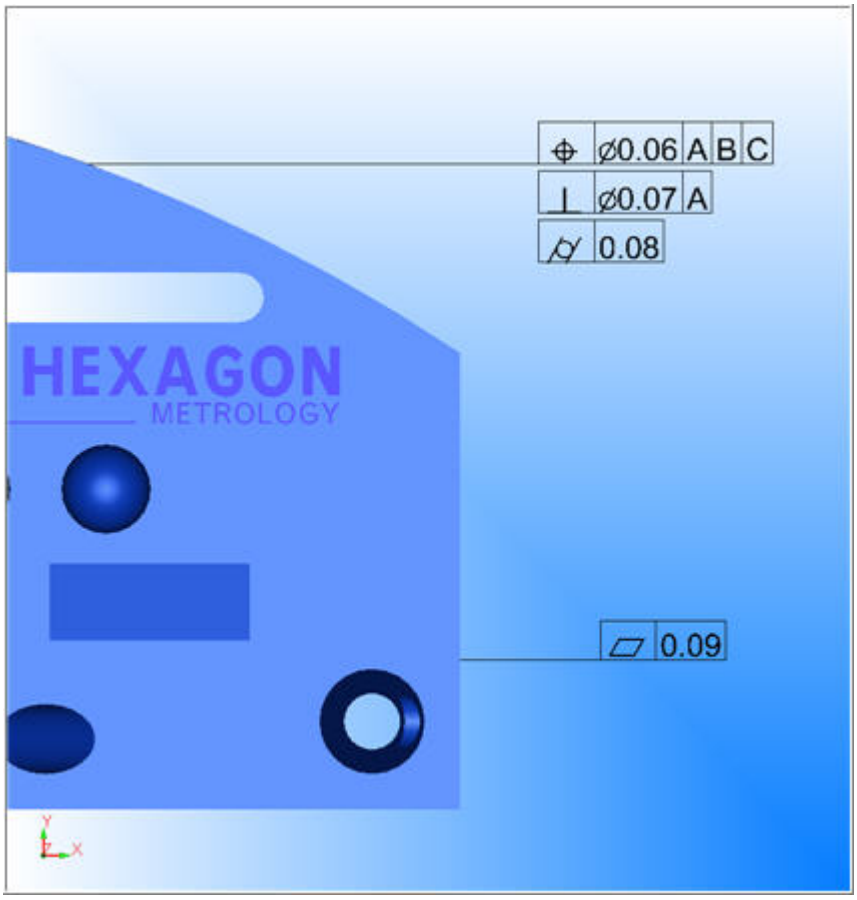
- CAD models
- GD&T
- Characteristic IDs
- Rotation Widget
- Feature Labels

- Point Info Labels
- Dimension Info Labels
- Cross Sections
- Label IDs
- Probe
- Machine
- Tool Changer
- Fixture
- Clearance Plane
- ClearanceCube

For information on the **Graphic Items** toolbar, see "Graphic Items Toolbar" in the "Using Toolbars" chapter.

Working with CAD GD&T Callouts

PC-DMIS lets you work with CAD GD&T callouts that are part of your CAD model in the Graphic Display window. You can show and hide these callouts, reposition them, change their view options, filter them, and import them into your measurement routine.



Sample CAD GD&T Callouts Displayed in the Graphic Display window

These CAD GD&T callouts are not actual Geometric Tolerances or Feature Control Frames. Instead, CAD GD&T callouts reside directly in your CAD model and appear as CAD elements. They do not initially perform any function in PC-DMIS other than providing visual information in the Graphic Display window. For details on Geometric Tolerances and Feature Control Frames, see "Introduction to Geometric Tolerances and Feature Control Frames".

To work with CAD GD&T callouts, your license must include the **CAD** and **GD&T Selection** options, or **CAD++**. Also, your CAD model must use a supported format. Supported formats include: CATIA v5, CATIA v6, Creo, NX, SolidWorks, and STEP AP242.

PC-DMIS also partially supports CAD GD&T callouts in IGES files. The IGES file must contain the following:

- 2D drawings with GD&T callouts
- A 3D model that corresponds to the 2D drawings

During the import process, PC-DMIS attempts to match the 2D GD&T callouts to the 3D features on the IGES model. Once PC-DMIS matches 2D callouts to the 3D features, you can then import the embedded callouts as features and FCF dimensions.

Showing or Hiding CAD GD&T Callouts

To show and hide the callouts, from the **Graphic Items** toolbar, click the **Show GD&T** icon ( ). If you want the callouts to always appear in newly created measurement routines, from the PC-DMIS Settings Editor, in the **Options** section, set the `ShowCADGDT` entry to **TRUE**.

Showing or Hiding CAD GD&T Notes

To show or hide GD&T notes, from the **Graphic Items** toolbar, click the **Show GD&T Notes** icon ( ).

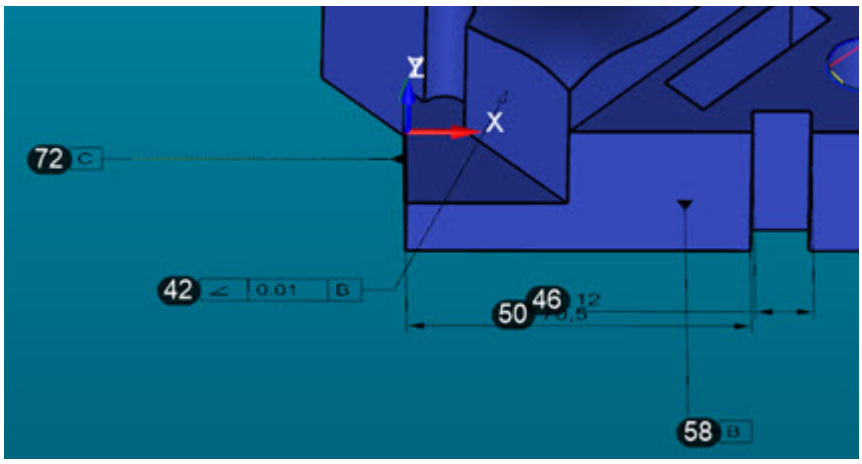
GD&T notes contain information embedded directly within the CAD model.

You cannot import them as commands into the Edit window.

Showing or Hiding CAD GD&T Characteristic IDs

You can use the **Characteristic IDs** icon ( ) on the **Graphic Items** toolbar to show and hide CAD GD&T characteristic IDs on CAD GD&T callouts. A characteristic ID on a callout is a unique identifier assigned to the CAD GD&T by the native CAD software. You can only see a characteristic ID if its corresponding CAD GD&T is also visible.

For information on the **Characteristic IDs** icon on the **Graphic Items** toolbar, see "Graphic Items Toolbar" in the "Using Toolbars" chapter of the PC-DMIS Core documentation.



Example of Characteristic IDs

You can modify the font size with the **Graphic Font** on the **Font Setup** dialog box. The font **Color** on that dialog box controls the background color of the characteristic ID. The foreground color automatically uses black or white to maintain good contrast.

> For information on the **Font Setup** dialog box, see "Customizing User Interface Fonts" in the "Setting Your Preferences" chapter.

You can use an Excel sheet to map different values to each characteristic ID and show those instead. To do this, use the **Balloon ID** tab on the **CAD and Graphic Setup** dialog box.

> For information on the **Balloon ID** tab, see the "Balloon ID Tab" topic.

> Don't confuse these characteristic IDs with the characteristic IDs PC-DMIS uses when you import a CAD GD&T callout. That type of characteristic ID comes from the native CAD model's PMI. For additional information on characteristic IDs in feature names, see "Use Characteristic ID Naming" in the "Setting Your Preferences" chapter.

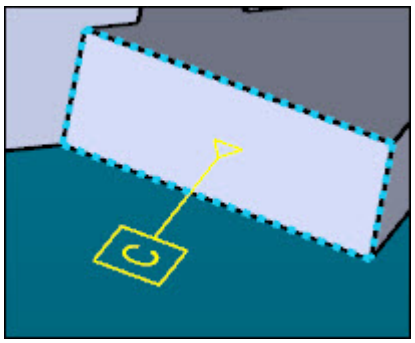
Viewing Associated CAD Elements

If you have the **CAD Information** dialog box open (**View | CAD Information**) and you click on a CAD GD&T callout, the software highlights the associated CAD element.

Inside the **CAD Information** dialog box, you can then click on links inside of curly braces to zoom in on specific CAD elements.

For more information on the **CAD Information** dialog box, see "Viewing CAD Information".

Another way to view associated CAD elements is to press Shift and position your mouse pointer over the GD&T callout. PC-DMIS highlights both the callout and all CAD elements linked to the callout:



Example of highlighting the surface for datum C

Associating CAD GD&T Callouts with CAD Elements

When you import a part with embedded CAD GD&T callouts, PC-DMIS does its best to link it to the correct CAD element. If PC-DMIS links something incorrectly, you can fix it with the **Associate CAD elements** shortcut menu.

To access this menu, from the **Graphic Modes** toolbar, click the **GD&T Selection Mode (from CAD)** button ( ). Then right-click on a CAD GD&T callout and choose **Associate CAD Elements** to open the **Associate CAD Elements** dialog box.

This dialog box shows the current associated CAD elements in the **Elements** list:

Associate CAD Elements dialog box with two associated elements

You can click on any element to zoom to that element in the Graphic Display window.

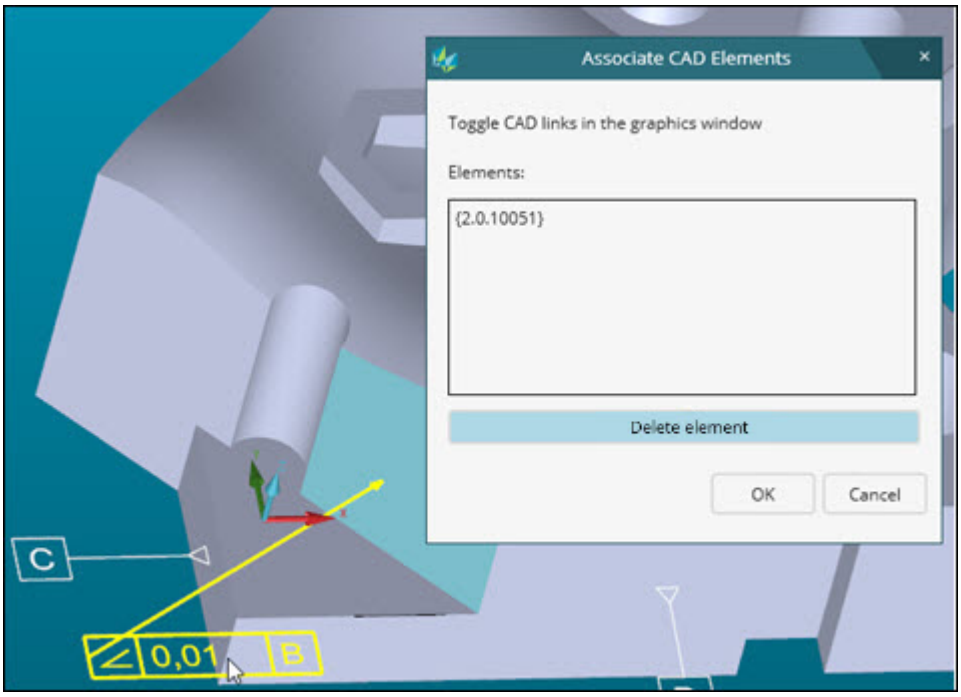
To remove an association:

1. Select the **Associate CAD elements** shortcut menu to see the current association in the **Associate CAD Elements** dialog box.
2. From the **Elements** list, select the element.
3. Click the **Delete element** button to remove the item from the **Elements** list.

4. Click **OK** to confirm the change.

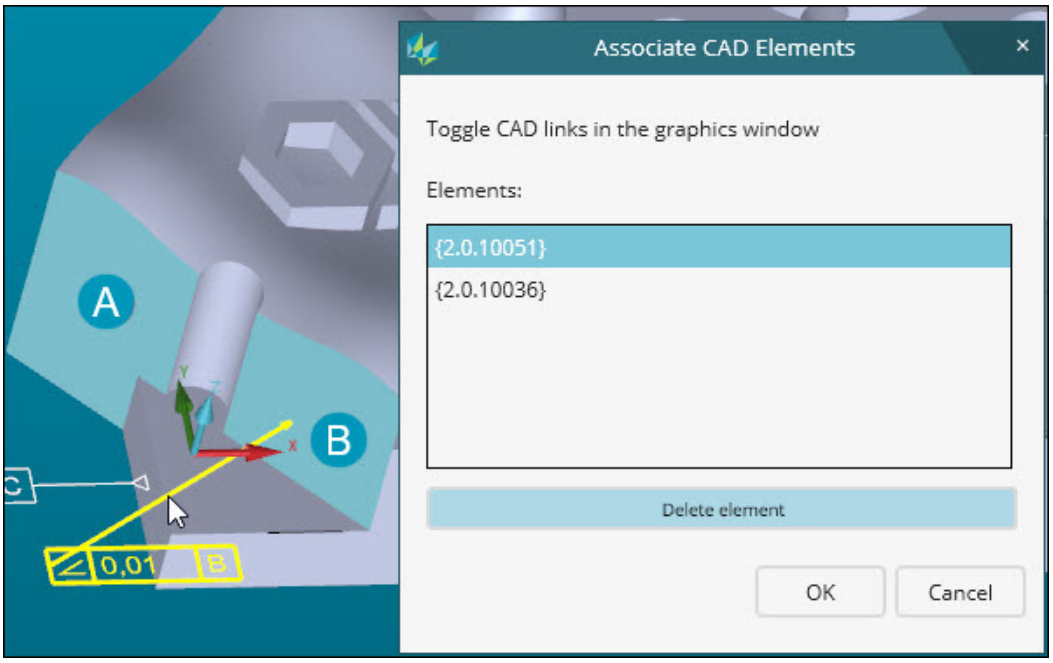
To add a new association:

1. Choose the **Associate CAD elements** shortcut menu to see the current association in the **Associate CAD Elements** dialog box.



GD&T callout initially associated to a single element

2. With the dialog box open, from the CAD model, click on a CAD element. PC-DMIS highlights the selected CAD element and adds it into the **Elements** list.



GD&T callout now associated to both elements (A and B)

3. Click **OK** to confirm the change.

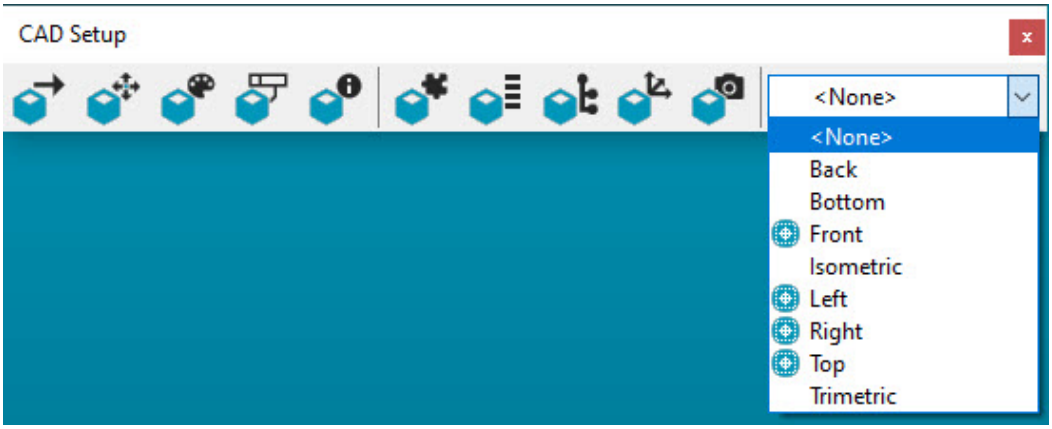
Associating CAD GD&T Callouts with Existing PC-DMIS Features

Similar to creating and removing associations between GD&T callouts and CAD elements, you can also add or remove an association between CAD GD&T callouts and existing features from your measurement routine. This can be helpful if you do not want to use the default Auto Feature method to measure a feature. For example, you may want to create constructions instead.

To associate a CAD GD&T callout with an existing PC-DMIS feature, do this:

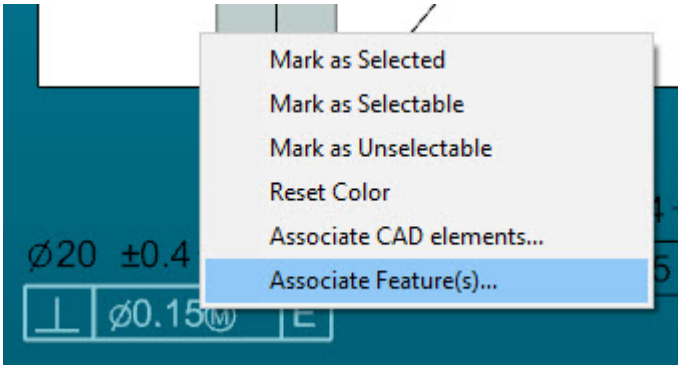
1. From the **Graphic Modes** toolbar, click the **GD&T Selection Mode (from CAD)** button ( ).

2. From the **CAD Setup** toolbar (**View | Toolbars | CAD Setup**), select the view associated with the features from the **CAD Model View** list. The view you select must have the PMI displayed as shown.



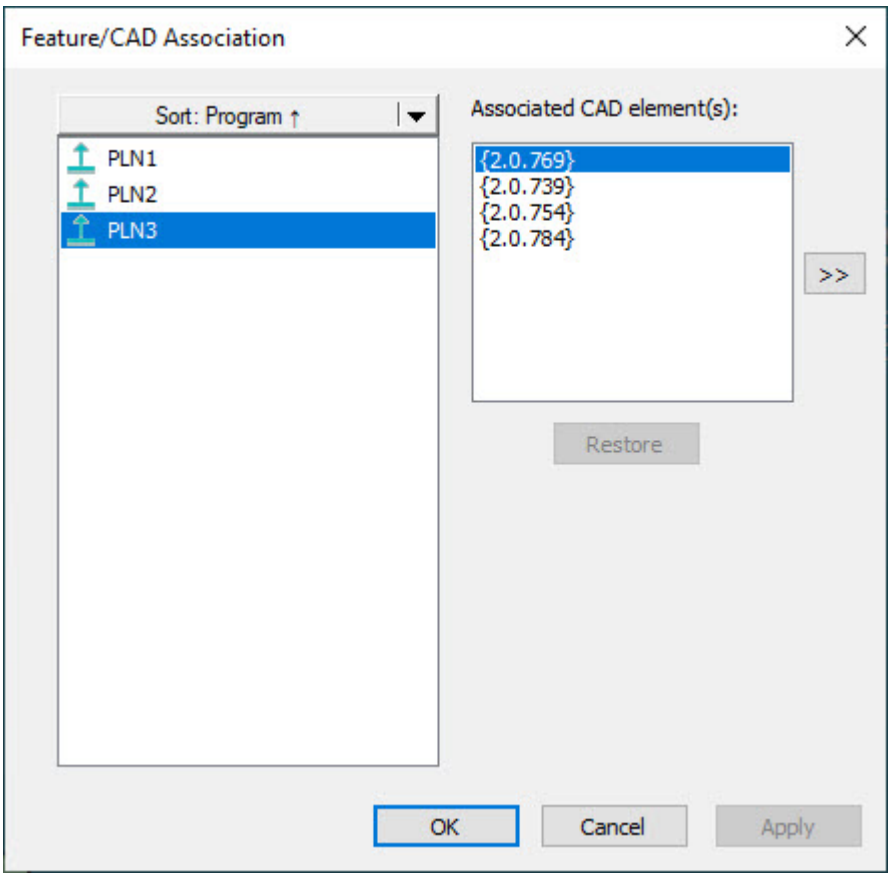
Example of view options with PMI in CAD and displayed (Front, Left, Right, and Top)

3. Right-click on a CAD GD&T callout in the Graphic Display window and select the **Associate Feature(s)** option.



PMI menu with the Associate Feature(s) option selected

4. The **Feature/CAD Association** dialog box opens. Select a feature from the **Features** list on the left to view its associated CAD feature elements in the **Associated CAD element(s)** list on the right. PC-DMIS highlights all associated feature elements in the Graphic Display window.



Feature/CAD Association dialog box

5. Click **OK** to save your changes.

To remove an association between a CAD GD&T Callout and a feature, do this:

1. Right-click on a CAD GD&T callout in the Graphic Display window and select the **Associate Feature(s)** option.

2. Select a feature from the Feature list on the left to view the associated feature elements in the **Associated CAD element(s)** list on the right. PC-DMIS highlights all the elements associated with the selected feature on the CAD model in the Graphic Display window.

3. Click any of the highlighted features in the Graphic Display window to temporarily remove them from the **Associated CAD element(s)** list. This also removes their association with the selected feature in the **Feature** list causing PC-DMIS to no longer highlight the selected feature in the Graphic Display window.

   You can also select the feature element from the **Associated CAD element(s)** list and click the **>>** button to remove the association.

   Click the **Restore** button to insert the feature element back into the **Associated CAD element(s)** list. This restores its association with the selected feature and PC-DMIS highlights the element in the Graphic Display window again.

4. Click **OK** to save any changes you made.

Once you click **OK**, any associations you removed are permanent. If you need to recreate the association, you must perform the steps to associate a CAD GD&T callout with a CAD feature and select the feature element that you removed.

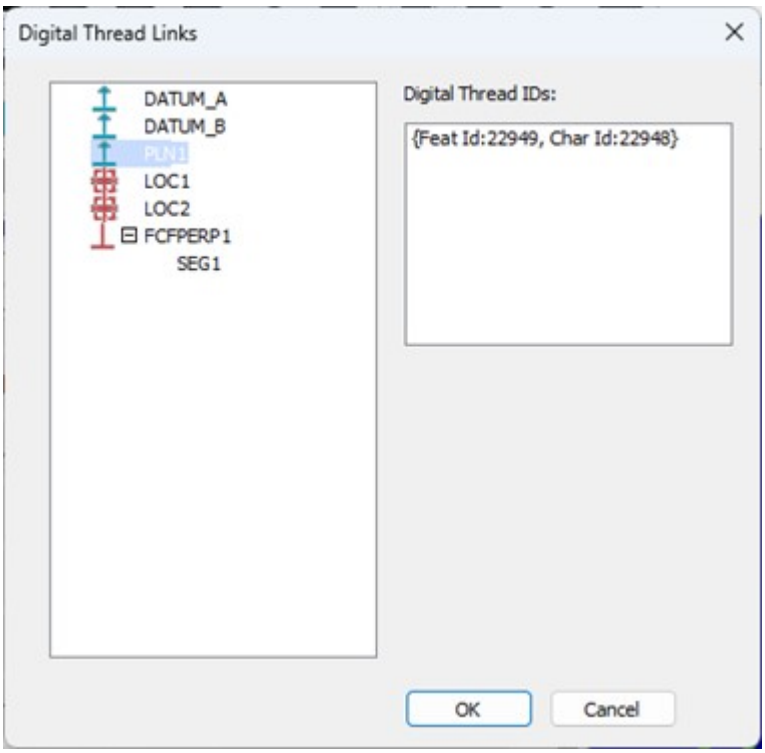
Once you associate a feature with a CAD surface, PC-DMIS recognizes it when it consumes the GD&T callout and uses the pre-existing, associated feature rather than inserting a new feature into the measurement routine.

Viewing GD&T Digital Thread IDs

When you import a CAD file, PC-DMIS reads the digital thread IDs (Feature ID and Character ID) from the CAD data and stores them internally. When you create features and dimensions using the associated PMI, PC-DMIS inserts these IDs into the corresponding PC-DMIS commands. As a result, when you export the data, the digital thread IDs are preserved and included in the output.

Digital thread IDs are available for CAD data imported from a .qif file, or from a STEP file that contains UUIDs for the features and characteristics. PC-DMIS includes digital thread ID information only in QIF report.

To view the associated digital thread IDs for a feature or dimension, select **Operation | Graphic Display Window | GD&T Selection | View GD&T Digital Thread Ids** to open the **Digital Thread Links** dialog box. From the **Features** list on the left, select a feature or dimension to view its associated digital thread IDs in the **Digital Thread IDs** area.



Example showing the digital thread IDs (feature ID and character ID) for a selected plane feature.

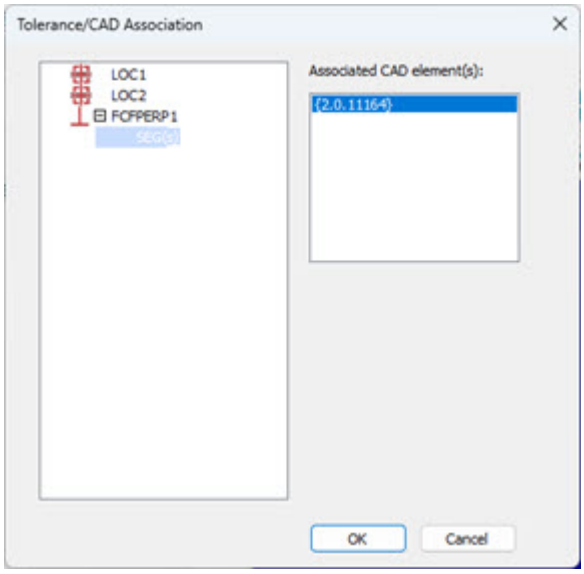
Associating GD&T Digital Thread IDs with Dimensions

You can associate GD&T digital thread IDs with dimensions that you create manually. To associate GD&T digital thread IDs with dimensions, follow these steps:

> If the dimension references measured or constructed features, make sure you associate those features with the required CAD elements before you associate the dimension. For information on how to associate features, see "Associating CAD GD&T Callouts with Existing PC-DMIS Features" in the "Working with CAD GD&T Callouts" topic in the PC-DMIS Core documentation.

1. From the **Graphic Modes** toolbar, click the **GD&T Selection Mode (from CAD)** button ( ).
2. Right-click on a CAD GD&T callout in the Graphic Display window and select **Associate Tolerance(s)** to open the **Tolerance/CAD Association** dialog box.
3. Select a dimension from the **Dimension** list and then click the CAD GD&T callout in the Graphic Display window that you want to associate with the selected dimension.

   PC-DMIS displays the associated CAD element in the **Associated CAD element(s)** area.



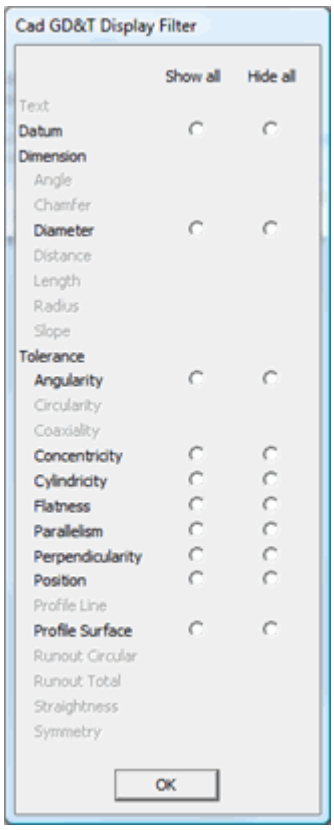
4. Click **OK** to save your changes.

Moving CAD GD&T Callouts

To reposition a CAD GD&T callout, from the **Graphic Modes** toolbar, turn on Text Box Mode ( ), and then drag the callout to a new location.

Filtering CAD GD&T Callouts

1. From the **Graphic Modes** toolbar, turn on Text Box Mode ( ).
2. Right-click on a GD&T callout, and from the shortcut menu, select **CAD GD&T Display Filter** to show the **CAD GD&T Display Filter** dialog box. This dialog box shows a list of CAD GD&T information and two columns of option buttons (**Show all** and **Hide all**).
3. For each item you want to show or hide, click **Show all** or **Hide all** to immediately toggle the visibility state of that GD&T callout.



4. Click **OK** to close the dialog box and store the current filtered state until accessed and modified later.

Showing or Hiding Individual Callouts

From the **Graphic Modes** toolbar, turn on Text Box Mode ( ), right-click on a GD&T callout, and select one of the Show or Hide ID menu items.

Importing CAD GD&T Callouts

PC-DMIS can import selected GD&T callouts as dynamically-generated FCF dimensions or datum definitions.

1. From the **Graphic Items** toolbar, click the **Show GD&T** icon ( ), to ensure that the GD&T callouts are visible.

> You can hide all the label types, so that they don't fill up the screen during the import process. To do this, from the **Graphic Items** toolbar, click the **Show All Label IDs** icon ( ).

2. Do one of the following:

- *Select* **Operation | Graphic Display Window | GD&T Selection | Select All GD&T**. This option selects all embedded CAD GD&T callouts and converts them into Edit window commands.
- *Select* **Operation | Graphic Display Window | GD&T Selection | Select GD&T in Current View**. This option selects only the embedded CAD GD&T callouts that are visible and converts them into Edit window commands.
- *Select* **Operation | Graphic Display Window | GD&T Selection | Select GD&T in Active CAD Model View**. This option creates GD&T dimensions from the embedded CAD GD&T callouts stored inside an active CAD model view. This menu remains grayed out until you make a view active. To make a view active, from the **CAD Model Views** dialog box (**Edit | Graphic Display window | CAD Model Views**), from the list of views, double-click on a model view. For information on CAD Model Views, see "Working with CAD Model Views".

- Enable the **GD&T Selection Mode (from CAD)** ( ) icon on the **Graphic Modes** toolbar. Then *click on a single item*. In the Edit window, PC-DMIS generates all necessary commands that make up the specific datum definition or FCF dimension. These commands are temporary. PC-DMIS removes them if you click **Cancel**. From the **Geometric Tolerance Command** dialog box that appears, make any changes to the FCF dimension. Click **OK** to accept the dimension and make referenced temporary commands permanent. Temporary commands that are not referenced are removed.

  Even if PC-DMIS is not in **GD&T Selection Mode (from CAD)**, as long as PC-DMIS is in a supported mode and the callout is visible, you can press Shift and click on the callout to import it. Supported modes include Translate Mode, 2D Rotate Mode, and 3D Rotate Mode.

- Enable the **GD&T Selection Mode (from CAD)** ( ) icon on the **Graphic Modes** toolbar. Then *box-select multiple callouts*. In the Edit window, PC-DMIS generates all necessary commands that make up the selected datum definitions or FCF dimensions.

If the selected CAD GD&T callouts don't have geometry features, they are incomplete. You cannot import incomplete GD&T callouts into the measurement routine. PC-DMIS shows incomplete or invalid callouts in the **Invalid GD&T** tab of the **CAD GD&T Selection Information** dialog box.

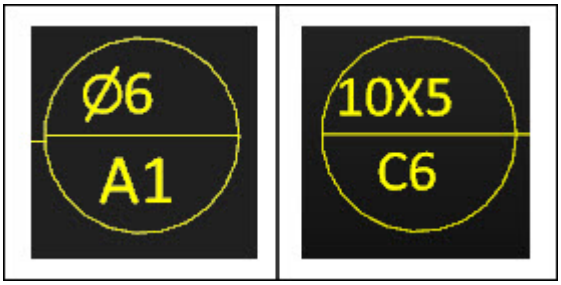
If the measurement routine's GD&T standard does not support the selected CAD GD&T callout, PC-DMIS adds an operator comment stating that the geometric tolerance command was not created. It then lists the IDs of the features that it did add to the routine. For information on the routine's GD&T standard, see the "Setup Options: Geometric Tolerances Tab" topic in the "Setting Your Preferences" chapter.

If you hold down Shift before you click on or box-select the CAD GD&T callout item, the software generates the FCF datum definitions and commands in the Edit window without showing the **Geometric Tolerance Command** dialog box.

By default, if you hold down Shift and click (or double-click) on a GD&T callout item, PC-DMIS displays the measurement strategy widget. You can use this widget to change a feature's main parameters. For more information on this widget, see "Using the Measurement Strategy Widget" in the "Creating Auto Features" chapter.

Datum Target Areas

You can import rectangular and circular datum target areas if they are aligned to an axis. The following picture shows some sample datum target areas:



The left shows a circular area, while the right sample shows a rectangular area

When you import a datum target area, the software creates the following in the Edit window:

- A High Point feature
- A workplane that corresponds to the target area surface normal

Regions of Interest (ROI)

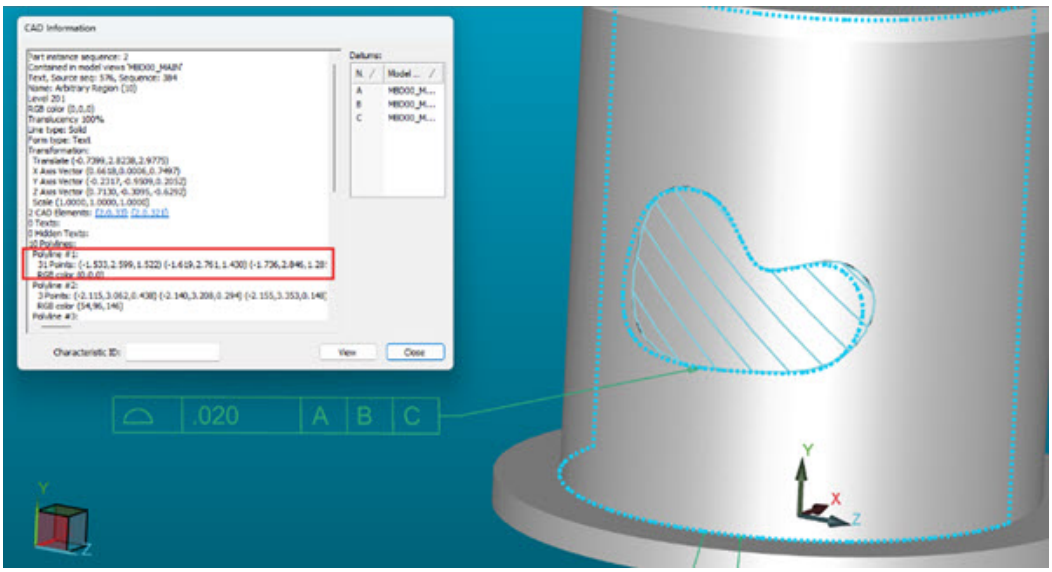


Image showing a surface profile within a specified region of interest, the associated CAD elements, and the CAD Information dialog, with the polyline information highlighted

You can import a surface profile tolerance with a restricted region of interest, if the region's outer boundary is defined as a CAD polyline. The associated CAD elements must be the polyline and the underlying CAD surface. PC-DMIS uses the polyline points as the boundary points of a Patch scan, limiting the extent of measurement to the region within the polyline.

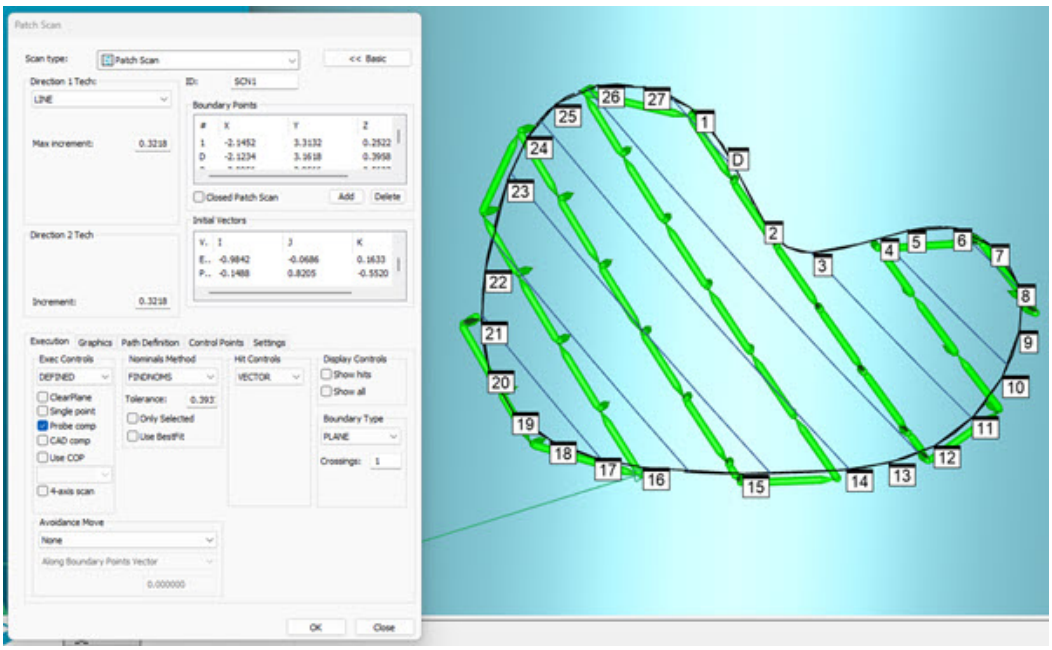


Image showing the Patch scan's boundary points and the path lines of the generated scan

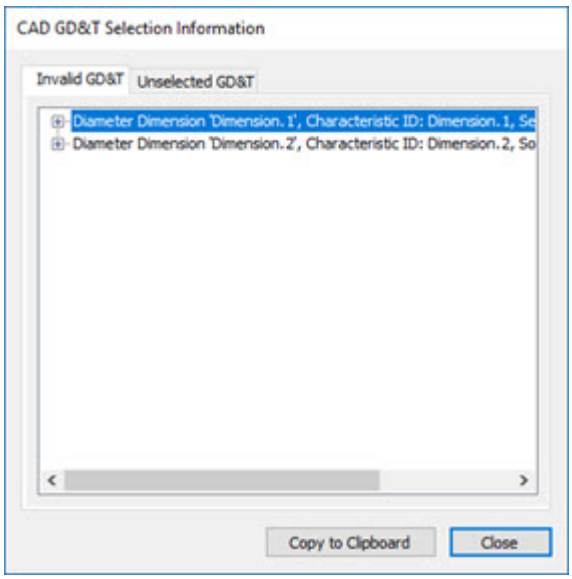
Use Characteristic ID Naming

When you import callouts, you can have the features use their native characteristic IDs instead of PC-DMIS's default feature naming. For more information see "Use Characteristic ID Naming" in the "Setting Your Preferences" chapter.

Viewing Invalid GD&T Callouts

The **Operation | Graphic Display Window | GD&T Selection | View Invalid GD&T for Selection** menu option opens the **Invalid GD&T** tab on the **CAD GD&T Selection Information** dialog box. This tab also appears after you import CAD GD&T callouts as Edit window commands and some callouts were incomplete or invalid.

This tab analyzes all the CAD GD&T callouts to locate invalid callouts. Invalid callouts are those that the software cannot solve for selection. The dialog box then displays these callouts in a list.



CAD GD&T Selection Information dialog box - Invalid GD&T tab

You can expand an object to see a description of why the software cannot solve the object for selection.

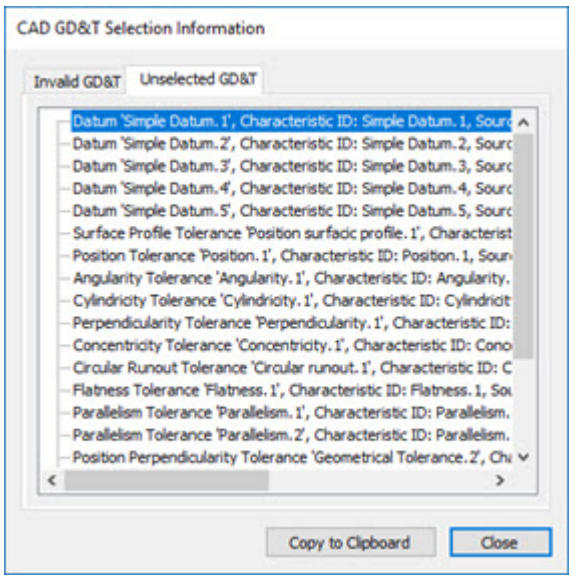
You can click on an object to highlight the object in the Graphic Display window. You can double-click on an object to highlight and scale the object. These make it easier to locate.

The **Copy to clipboard** button copies the text of the contents to the Clipboard.

> This option cannot detect all the reasons why the software cannot solve an object for selection. This means objects may not appear in this dialog box that it cannot solve.

Viewing Unselected GD&T Callouts

The **Operation | Graphic Display Window | GD&T Selection | View GD&T Not Selected** menu option opens the **Unselected GD&T** tab on the **CAD GD&T Selection Information** dialog box.



CAD GD&T Selection Information dialog box - Unselected GD&T tab

This tab analyzes all the CAD GD&T callouts to locate unselected callouts. Unselected callouts are those that you have not selected using GD&T selection. That is, these callouts don't have corresponding measurement routine commands.

Once you import them with GD&T selection, they no longer appear in this tab. For information on GD&T selection, see the "Importing CAD GD&T Callouts" subtopic above.

You can click on an object to highlight the object in the Graphic Display window. You can double-click on an object to highlight and scale the object. These make it easier to locate.

The **Copy to clipboard** button copies the text of the contents to the Clipboard.

Setting CAD GD&T Callout Colors

You can set the colors the software uses for valid and invalid CAD GD&T callouts in the Graphic Display window. For information on how to do this, see "CAD GD&T Tab" under "Setting Up CAD and Graphics".

Marking CAD GD&T Callouts with Colors for Selected, Selectable, or Unselectable

You can set CAD GD&T Callouts to different colors to better communicate the state of the current callout (selected, selectable, or unselectable).

For more information, see "CAD GD&T Callouts Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.

Excluding CAD GD&T Callouts

You can use .exclude files to exclude items that PC-DMIS would normally accept as selectable CAD GD&T callouts. One reason to do this is to only display callouts that apply to a specific measurement device.

1. Choose **Edit | Graphic Display Window | CAD GD&T** to access the **CAD GD&T** tab on the **CAD and Graphic Setup** dialog box.

2. From the **Exclude** area, click **Generate** to create an .exclude file based on the available CAD GD&T callouts.
3. Click **Edit** to open the .exclude file in a text editor.
4. For any item that you want to exclude, remove the single quote character at the front of the line.
5. Mark the **Use exclude file** check box.
    - In the **Exclude** area, click **Apply** if you want to color the excluded CAD GD&T callouts.
    - Click **Hide excluded data**, if you want to hide the items you chose to exclude.
6. Click **OK**.

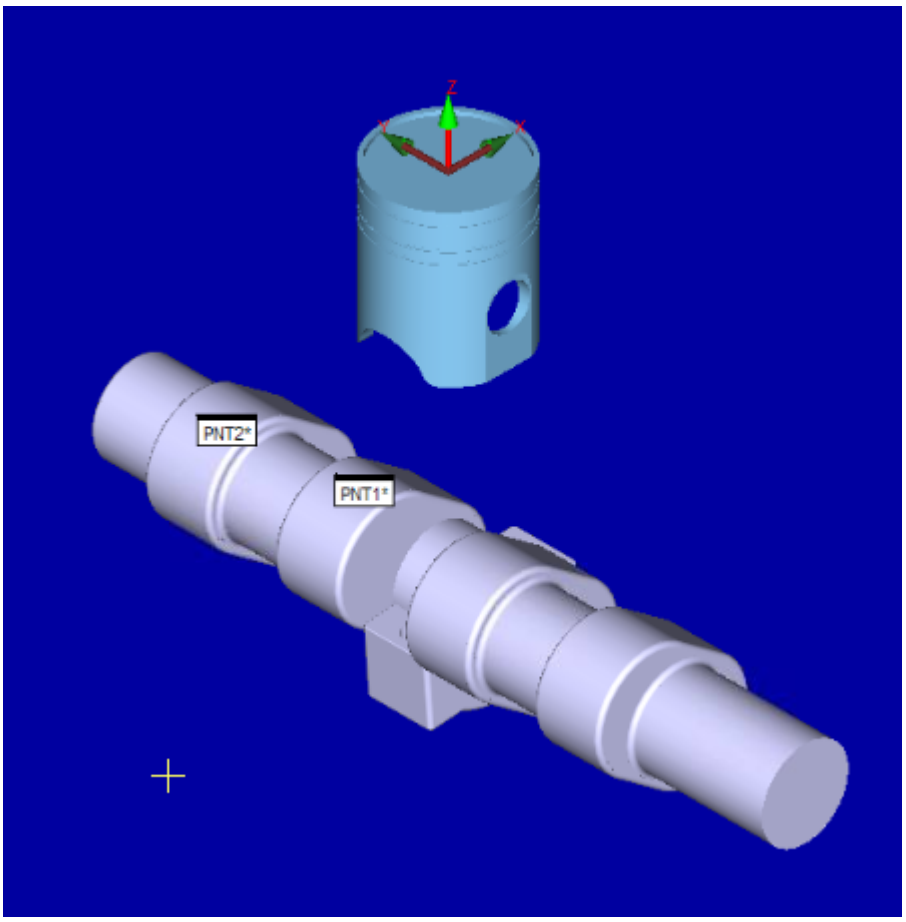
For information on .exclude files, see the "CAD GD&T Tab" topic in this chapter.

Instead of using the **Generate** button to create the .exclude file, you can also manually create your own .exclude file. To do this:

1. Create a text file with the same name as your measurement routine. Give the text file the **.exclude** filename extension.
2. Use a text editor, and type the Characteristic IDs to exclude on separate lines. You only need to type the IDs. If an ID has a space in it, put quotes around the ID. For example, an ID of A 1 should be "A 1".
3. Save your changes, and place the .exclude file in the same directory as the measurement routine.

Working with Assemblies of Parts

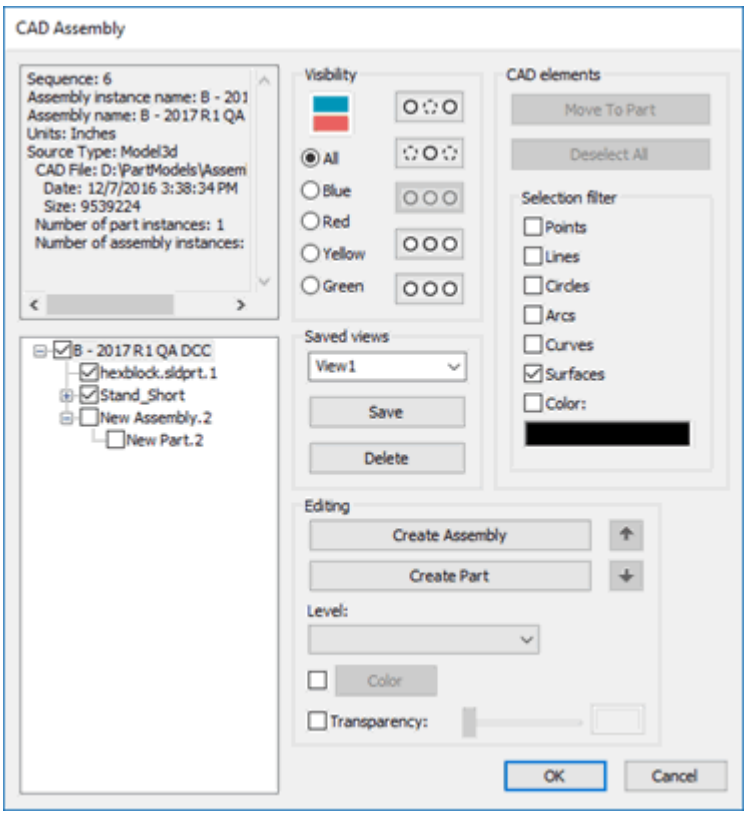
A CAD Assembly consists of two or more imported part models merged together in the Graphic Display window. For example, the graphic below shows an assembly of two visible parts.



A Sample CAD Assembly

You can have multiple parts and even multiple assemblies together in a single .cad file.

To work with assemblies, open the **CAD Assembly** dialog box. From the **CAD** toolbar, select the **Assembly** icon , or select **Edit | Graphic Display Window | CAD Assembly**.

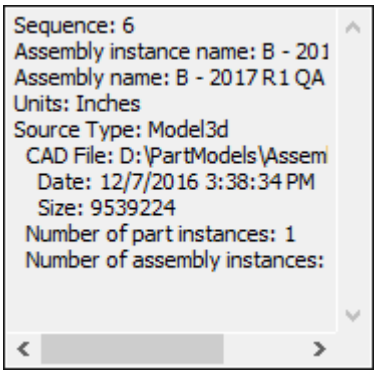


CAD Assembly Dialog Box

The **CAD Assembly** dialog box contains a list of all imported and merged parts, assemblies, and CAD section cuts used in the Graphic Display window. You can use this dialog box to view, edit, and manage assemblies of parts. You can also resize this dialog box, and the software remembers the last used size and position the next time it opens.

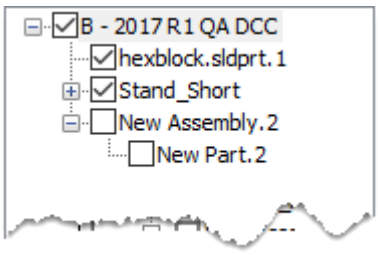
This dialog box contains these options:

Information Box



The gray box at the top of the dialog box displays information about the item currently selected on the Assembly Tree View. The information changes depending on the item selected.

Assembly Tree View



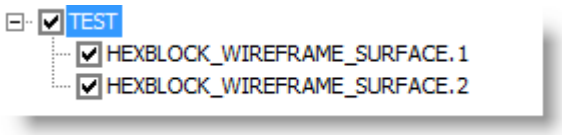
The left side of the dialog box has a tree view that represents the CAD assembly structure. Each item in the tree view represents an assembly component.

- If you select a component from the tree view, PC-DMIS displays information about that item in the gray box above the list.
- If you double-click a component in the assembly, PC-DMIS selects and highlights the component in the Graphic Display window. Conversely, with the **CAD Assembly** dialog box open, if you press and hold the Alt key and then left-click a CAD object in the Graphic Display window, PC-DMIS selects its corresponding component in the tree view. This helps to identify which CAD objects belong to which assembly component.

Each component has its own check box to show or hide that component immediately within the Graphic Display window.

- If you mark a check box, PC-DMIS shows that component.
- If you clear a check box, PC-DMIS hides that component (and any children components, regardless of their visibility state).
- Once you mark a check box, you can use the buttons in the **Visibility** area to change the visibility state of that component or of its sibling or children components.

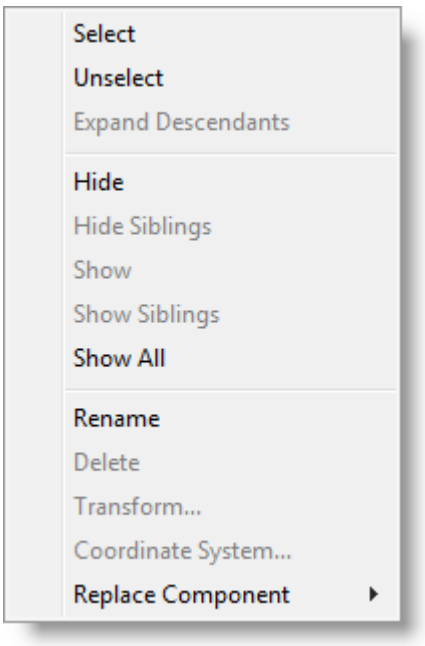
Your original imported image name has the number 1 appended to its name. Any CAD transformation that creates a new instance of this same CAD file are then incremented to have a unique number. See "Transforming a CAD Model".



Mirrored CAD File names with unique numbers inside a CAD Assembly Tree View

For information on SectionCutCurve entries, see "Creating and Viewing Section Cut Curves".

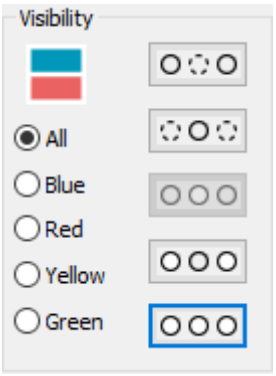
A shortcut menu appears when you right-click in the tree view. The shortcut menu contains these items:



- **Select** - This item functions as if you had double-clicked on the assembly component.
- **Unselect** - This item deselects the assembly component in the Graphic Display window.
- **Expand Descendants** - This item expands an assembly to show a list of child parts making up the assembly.
- **Hide** - This item hides the selected component.
- **Hide Siblings** - This item hides all of the components except the selected component at the same level on the assembly tree.
- **Show** - This item shows the selected component.
- **Show Siblings** - This item shows all of the components except the selected component at the same level on the assembly tree.
- **Show All** - This item shows all the components on the assembly tree.
- **Rename -** This item renames the selected component in the assembly.
- **Delete** - This item deletes the selected component from the assembly and removes it from the Graphic Display window. You cannot delete the root tree view component.
- **Transform** - This item displays the **CAD Transform** dialog box. This dialog box allows you to transform (translate, scale, and rotate) a single component in the assembly. You cannot transform the root tree view component. For more information, see "Transforming a CAD Model".

- **Coordinate System** - This item displays the **CAD Coordinate System** dialog box. This dialog box lets you create and manage different coordinate systems. For more information, see "Working with CAD Coordinate Systems".
- **Replace Component** - This item allows the selected assembly component to be replaced with another CAD model. This model can either be an imported CAD model or a model connected with the DCI. The submenus have the same structure as the **File | Import** and **File | Direct CAD Interface** menu options respectively. The new model replaces whatever model is associated with the selected node. The new model on the node retains any transformation that was applied to the previous model. For information on how to import model files, see "Importing CAD Data or Feature Data" in the "Using Advanced File Options" chapter.

Visibility



The **Visibility** area provides option buttons and icon buttons that allow you to run visibility operations in order to change the hidden state of assembly component groups.

The colored icon at the top of this area shows your current screen layout. It corresponds with the icon used in the **Layout** area of the **View Setup** dialog box. See "Setting Up the Screen View".

The **Blue**, **Red**, **Yellow**, and **Green** options limit the visibility operation to only that "view" of the Graphic Display window. The **All** option applies the visibility operation to all views.

These command buttons have pictures on them that depict their action:

Hide - This button hides the selected component.

Hide Siblings - This button hides all of the components except the selected component at the same level on the assembly tree.

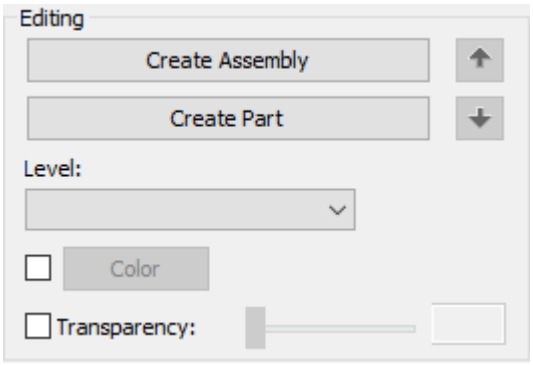
Show - This button shows the selected component.

Show Siblings - This button shows all of the components except the selected component at the same level on the assembly tree.

Show All - This button shows all the components on the assembly tree.

Only the hide and show properties are applied to individual views. The **Color** and **Transparency** settings in the **Editing** area are always applied to all views.

Editing



The **Editing** area lets you edit the currently selected component from the **Assembly Tree View**.

Create Assembly - This button creates a new assembly node in the currently selected assembly. Initially, the node is empty, but you can then move other assembly components into it by using the up and down arrow buttons ( ) on those components.

Create Part - This button creates a new part node in the currently selected assembly. Initially, the node is empty, but you can move CAD objects into it by using the Move to Part button in the CAD Elements area.

Up Arrow - This button moves the part or assembly component in the tree view up into the assembly just above it in the list.

Down Arrow - This button moves the part or assembly component in the tree view down into the assembly just below it in the list.

Color - This check box enables the **Color** button and displays the **Color** dialog box. This dialog box lets you apply a color to the selected tree view component. This check box also enables or disables the display of the selected color. Once you choose the initial color, you can click the **Color** button thereafter to modify the color used. This overrides any colors set for CAD elements using the **Edit CAD Elements** dialog box until the assembly component no longer uses a color or is removed from the **CAD Assembly** dialog box. For more information, see "Editing CAD".

Level - This list lets you assign the CAD objects within the selected assembly component to a predefined CAD level. This action overrides any CAD level assignments that were set using the **CAD Levels** dialog box until the assembly component is removed from the level

or from the assembly within the **CAD Assembly** dialog box. For more information, see "Working with CAD Levels".

Transparency - This check box applies the selected transparency percentage to the selected tree view component. You can drag the slider or change the value manually in the box to change the transparency percentage.

You cannot perform some edit operations on the root-level component (usually the part name used in your measurement routine) or DCI components. Those operations that you cannot use are disabled when you select these types of components.

Save Views

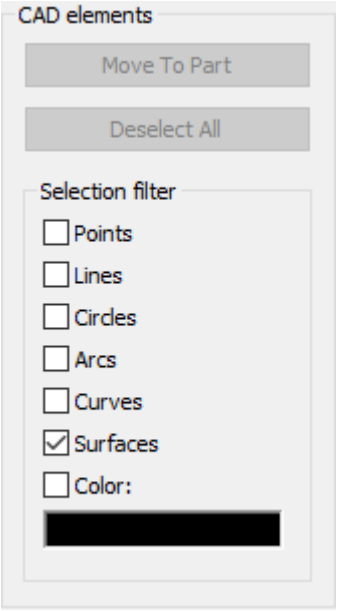
The **Saved Views** area lets you manage different visibility states of the **Assembly Tree View**.

Save - This button saves the current visibility state, giving it the name specified in the current **Saved Views** box. You must click the **OK** button to permanently save your view. PC-DMIS stores saved views in the .cad file associated with the measurement routine.

Delete - This button deletes the stored visibility state of the name selected from the **Saved Views** box.

To load a stored visibility state, select it from the **Saved Views** list. The check boxes in the **Assembly Tree View** are immediately modified according to the loaded view.

CAD Elements



The **CAD Elements** area lets you select CAD objects and move them to a different part. You can select or deselect CAD objects within the Graphic Display window. To do that, click on them individually, or for multiple CAD objects, drag a box around them.

Move To Part - This button moves the selected CAD objects to the currently selected part in the **Assembly Tree View**.

Deselect All - This button deselects all previously selected **CAD** objects.

Selection Filter - This area lets you filter the CAD objects that PC-DMIS can select. If an item is selected, PC-DMIS has the ability to select that type of CAD object when you click or box-select CAD objects in the Graphic Display window.

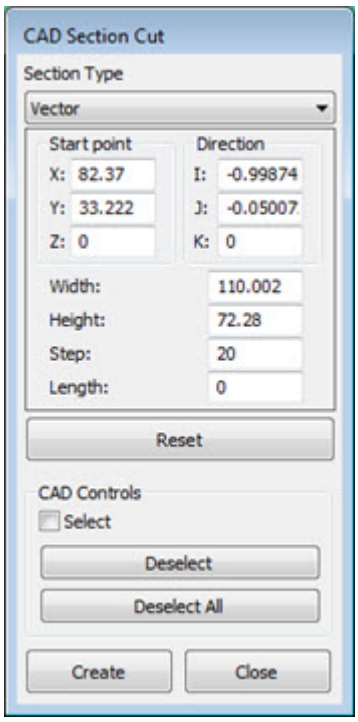
You can also filter the selection by the color of the CAD object. If you select the **Color** check box, the color of the next CAD object that you select in the Graphic Display window is then used as the filter color. For subsequent selections, you can only select CAD elements with that color.

Creating a CAD Section Cut

The **CAD Section Cut** function creates a CAD curve polyline defined by the contour of the CAD model.

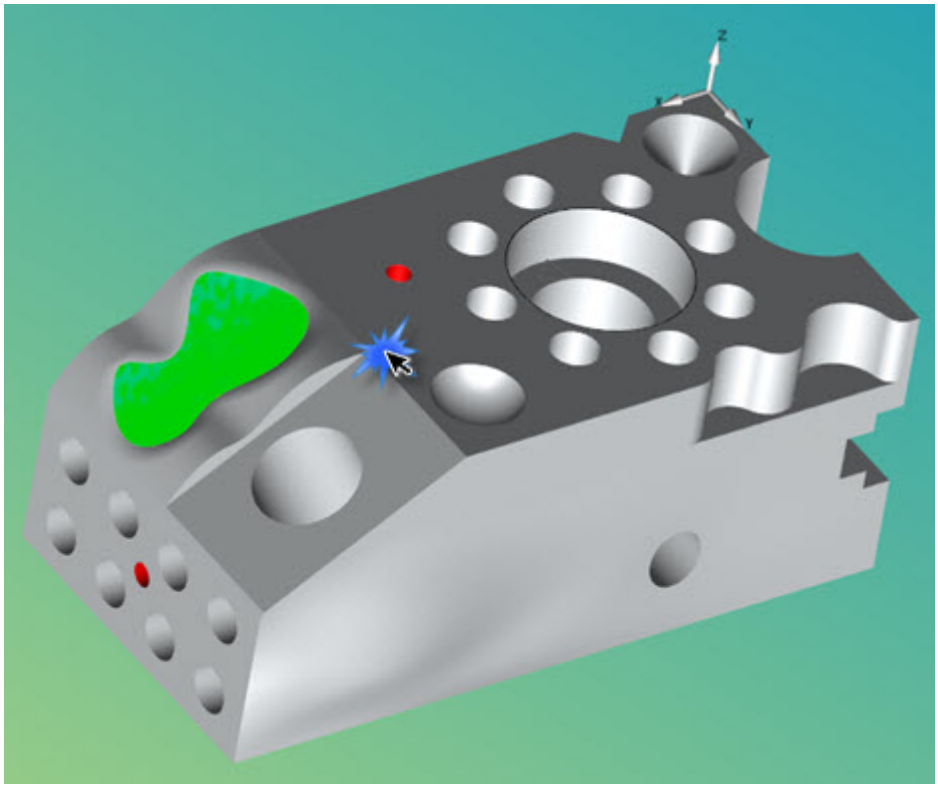
To create a CAD Section Cut:

1. From the main menu, select **Edit | Graphic Display Window | CAD Section Cut** to open the **CAD Section Cut** dialog box.

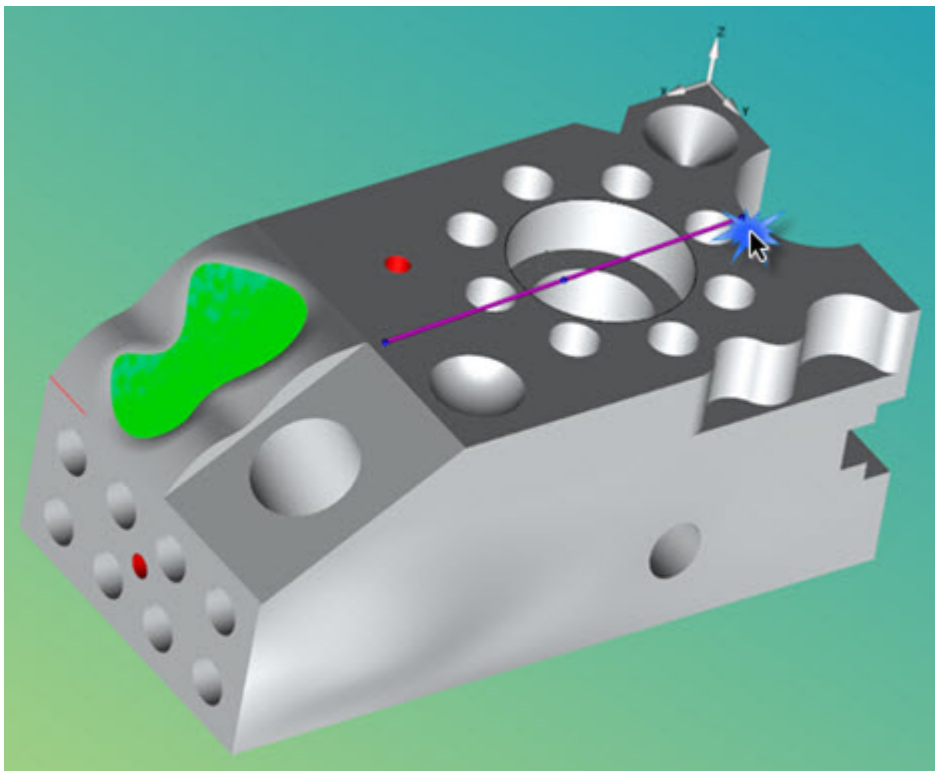


2. Select the type of section cut you want to define from the **Section Type** list. For details on these options, see the topic "Cross Section".

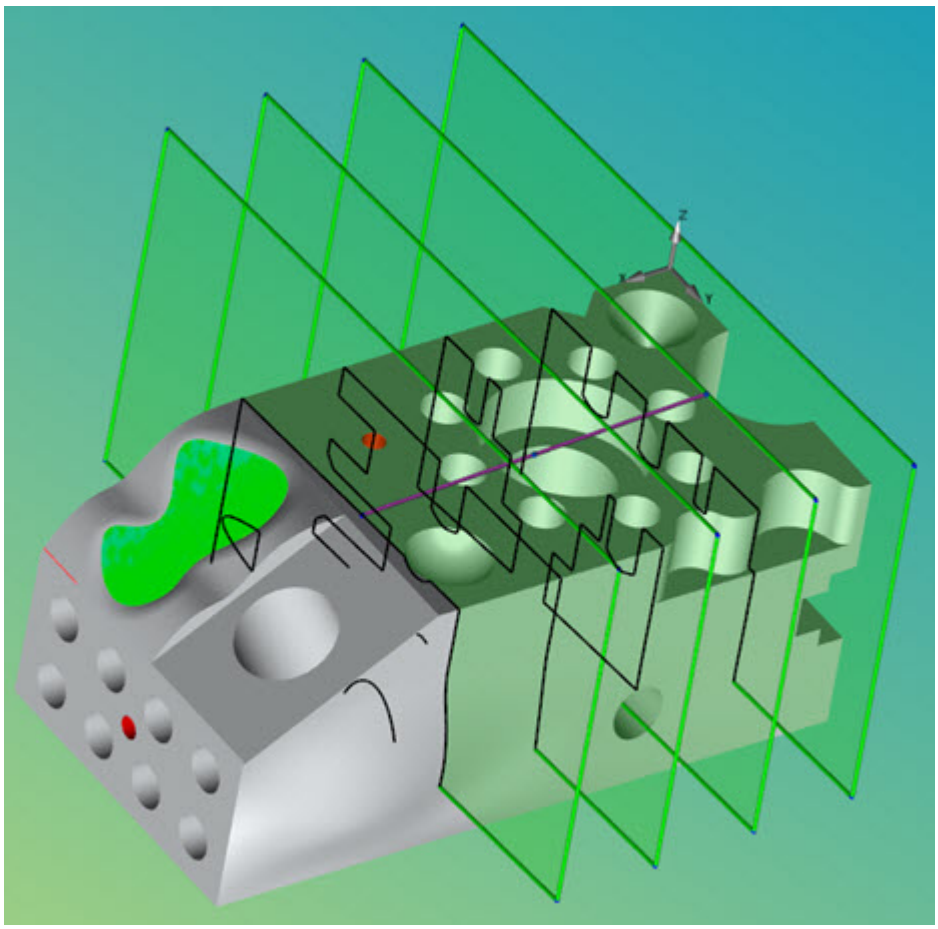
3. Click the surface where the section cut is to start.



4. Move your cursor to where the section cut is to end and click.

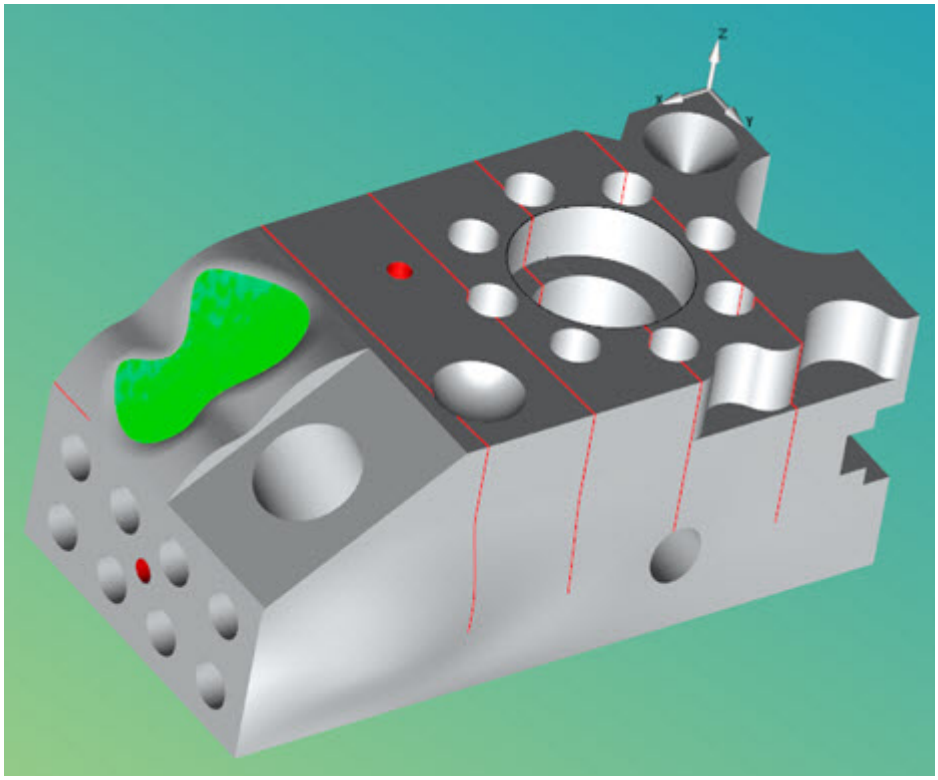


5. The CAD Section Cut is generated.



From the **CAD Section Cut** dialog box, edit the properties. Click and drag any of the section cut handles in the Graphic Display window to manipulate the properties.

6. Click **Create** to finalize the CAD section cuts.

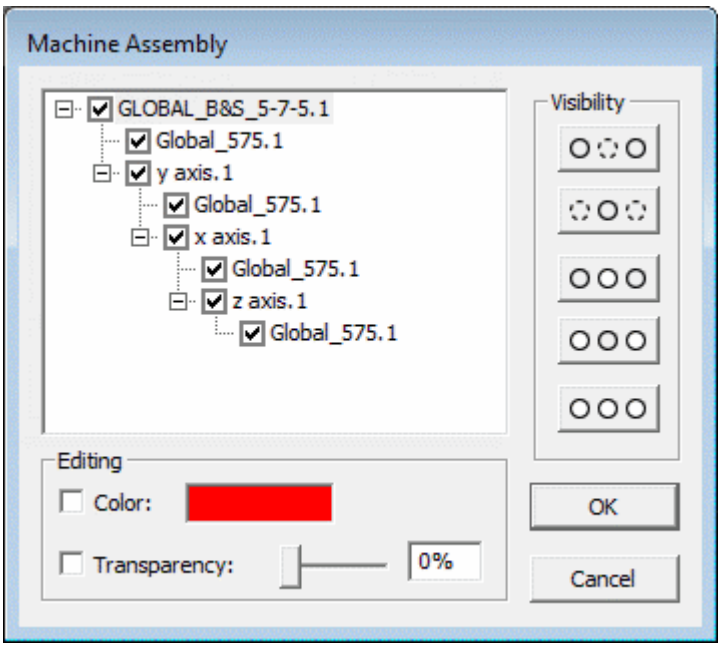


Once created, the CAD section cuts can be viewed in the **CAD Assembly** dialog box (**Edit | Graphic Display Window | CAD Assembly**).

7. Click **Close** when done.

Working with a Machine Assembly

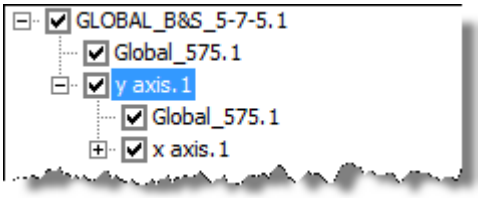
The **Edit | Graphic Display Window | Machine Assembly** displays the **Machine Assembly** dialog box. This gives you the tools to show and hide different components of your defined machine in the Graphic Display window.



Machine Assembly Dialog Box

This dialog box contains these options:

Assembly Tree View



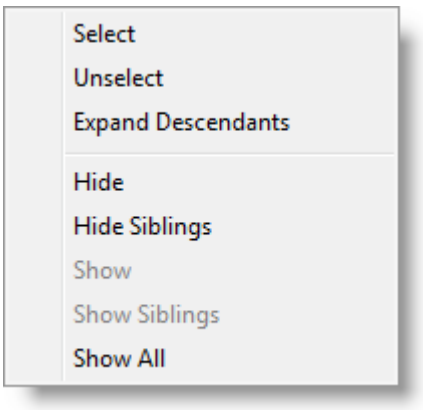
The tree view represents the machine's assembly structure. Each item in the tree view represents an assembly component.

- If you double-click on a component in the assembly, PC-DMIS selects and highlights that CAD object in the Graphic Display window. Conversely, with the dialog box open, if you select a CAD object in the Graphic Display window, PC-DMIS selects its corresponding component in the tree view. This makes it easy to identify which CAD objects belong to which assembly component.

Each component has its own check box to show or hide that component immediately within the Graphic Display window.

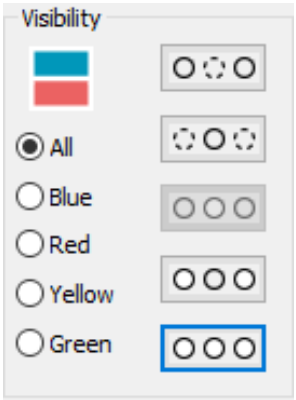
- If you mark a check box, PC-DMIS shows that component.
- If you clear a check box, PC-DMIS hides that component (and any children components, regardless of their visibility state).
- Once you mark a check box, you can also change the visibility state of that component or of its sibling or children components by using the buttons in the **Visibility** area.

A shortcut menu appears when you right-click in the tree view. It contains these items:



- **Select** - This functions as if you had double-clicked on the assembly component.
- **Unselect** - This deselects the assembly component in the Graphic Display window.
- **Expand Descendants** - This expands an assembly to show a list of child parts making up the assembly.
- **Hide** - This hides the selected component.
- **Hide Siblings** - This hides all of the components except the selected component at the same level on the assembly tree.
- **Show** - This shows the selected component.
- **Show Siblings** - This shows all of the components except the selected component at the same level on the assembly tree.
- **Show All** - This shows all the components on the assembly tree.

Visibility

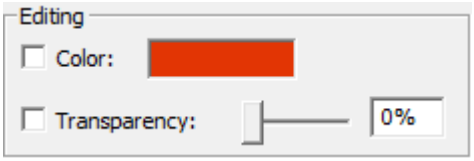


The **Visibility** area provides convenient buttons to change the hidden state of assembly component groups. Select the Hide button to hide the selected component.

- **Hide** - This hides the selected component.
- **Hide Siblings** - This hides all of the components except the selected component at the same level on the assembly tree.

- **Show** - This shows the selected component.
- **Show Siblings** - This shows all of the components except the selected component at the same level on the assembly tree.
- **Show All** - This shows all of the components on the assembly tree.

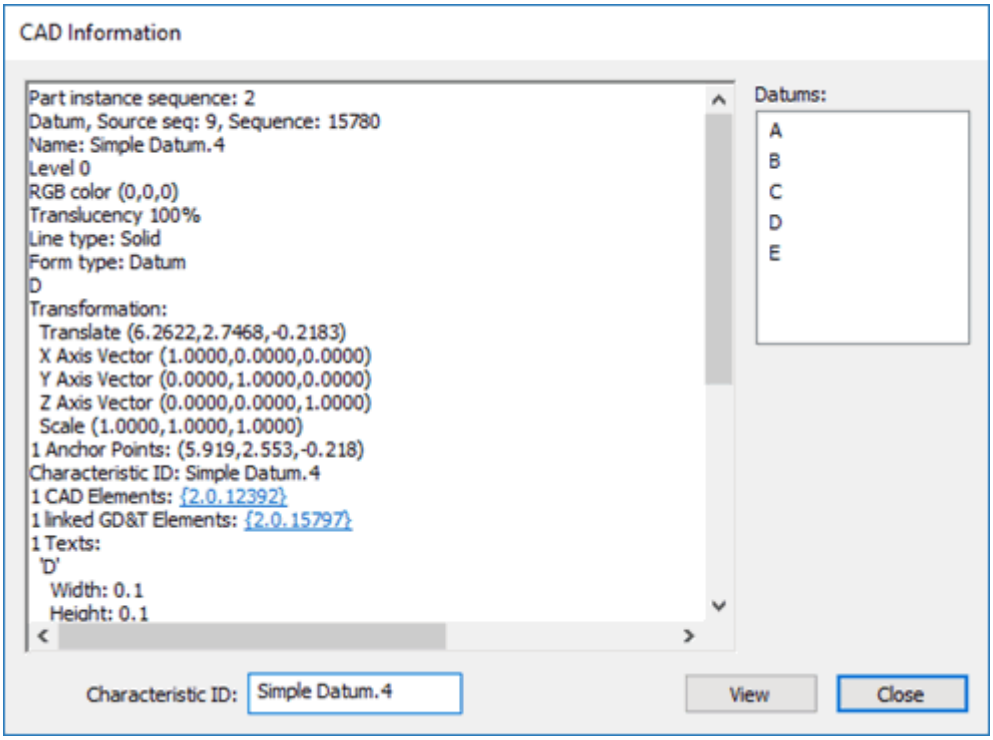
Editing



The **Editing** area lets you edit the currently selected component from the **Assembly Tree View**.

- **Color** - This check box enables the **Color** button and displays the **Color** dialog box. This dialog box lets you apply a color to the selected tree view component. This check box also enables or disables the display of the selected color. Once you choose the initial color, you can click the **Color** button thereafter to modify the color used.
- **Transparency** - This check box applies the selected transparency percentage to the selected tree view component. You can drag the slider or change the value manually in the box to change the transparency percentage.

Viewing CAD Information



CAD Information Display Box

The **View | CAD Information** menu option opens the **CAD Information** dialog box. This dialog box shows detailed CAD information for any CAD element you select from the Graphic Display window.

If you click on a CAD GD&T Callout, the following occurs:

- The software highlights the associated CAD element in the Graphic Display window.
- The CAD information in the dialog box includes curly braces with a link that zooms to that CAD element.

Datums - This list shows all the datums in the CAD model. You can click on a datum to have the Graphic Display window highlight it and rotate to it.

Characteristic ID - This ID box shows the Characteristic ID for a CAD GD&T object when you click on that CAD GD&T object in the Graphic Display window.

You can also use this box to do case-insensitive searches of CAD GD&T objects. To perform this search, type an ID in the box, and press Tab to have PC-DMIS show the CAD GD&T object with that ID.

- PC-DMIS also uses partial matches. For a partial match, PC-DMIS shows the first object with that ID in its Characteristic ID. You can press Enter to find the next partial match. You can keep pressing Enter to cycle through all available matches.

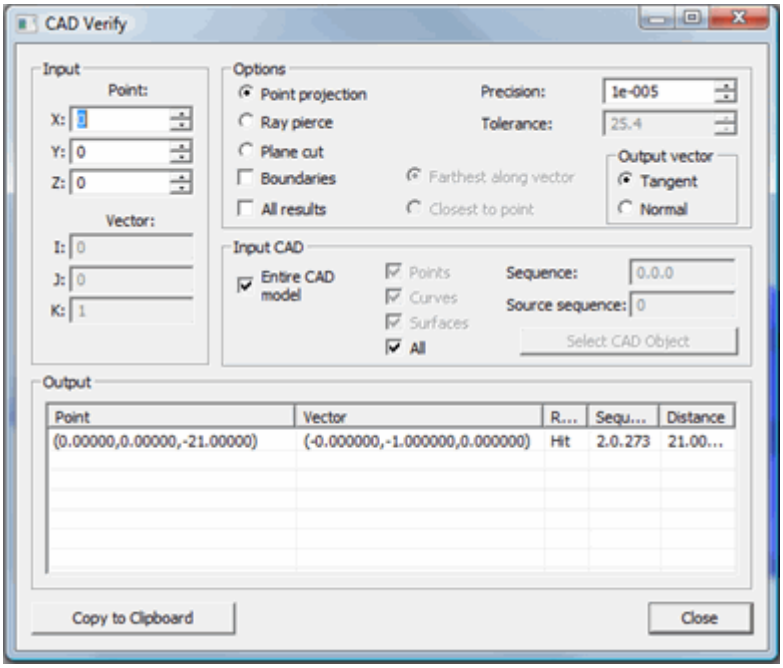
- If there is no matching CAD GD&T object with that ID, the value in the ID reverts to what it was.

You can manually revert an ID to its previous value if you press Escape before the box loses focus. The box loses focus whenever you click on something else you can interact with or if you press Tab or Enter.

Suppose you have some CAD GD&T objects, and two of them have Characteristic IDs of 23 and 125. If you type 2 in this box, PC-DMIS shows the object with 23 in its ID. In this example, both 23 and 125 are partial matches for 2, but 23 is earlier in the list of objects. You can then press Enter to have PC-DMIS show the next partial match of 125.

View - This button causes the selected datum to flash a few times in the Graphic Display window.

Verifying CAD Information



CAD Verify Dialog Box

You can use the **View | CAD Verify** menu item to verify that the CAD model is accurate based on the CAD native coordinate system. PC-DMIS displays a **CAD Verify** dialog box that allows you to perform different operations on the CAD model imported into the Graphic Display window. You can easily resize the **CAD Verify** dialog box to a larger size as needed.

The dialog box contains the following areas and items:

Input area

This area specifies the **XYZ** position and the **IJK** vector associated with it which are used to verify the CAD model. PC-DMIS disables the **IJK Vector** boxes if the **Point projection** option is selected.

Options area

This area determines what operation PC-DMIS performs on the CAD model. You can also specify options that control the behavior of the operations and the resulting output. The input point and vector in these options refer to the information entered into the **Input** area.

Point Projection - This projects the input point onto the CAD model. A *closest distance* algorithm is used to project the point.

Ray pierce - This option pierces the CAD model using a line. The input point and vector define the line.

Plane cut - This option intersects the CAD model using a plane. The input point defines a point on the plane and the vector specifies the plane's normal vector. For surfaces, only the surface boundaries are intersected with the plane.

Boundaries - This check box determines whether or not surface boundaries are used. If you select this check box and the CAD entity is a surface, only the surface boundaries are used. This does not affect curve geometry.

All results - This check box determines whether or not to get results for all the CAD entities in the model. If you don't select this check box, only the "best" result is displayed. The best result depends on the operation selected.

- For **Point projection**, the best result is the CAD point closest to the input point.
- For **Ray pierce**, the best result is either the intersection point farthest along the ray vector or closest to the input point. You can specify which intersection point is displayed. See "Farthest Along Vector" below.
- For **Plane cut**, the best result is the intersection point closest to the input point.

Farthest Along Vector - This option displays the intersection point farthest along the ray vector.

Closest to Point - This option displays the intersection point closest to the input point.

Resolution - This box controls the precision of the result. The result remains within the resolution value. The minimum resolution is 0.0000001.

For Direct CAD Interface models, the precision only affects the number of displayed decimal places. The actual CAD system controls the precision of the operations and cannot be modified from within PC-DMIS.

Tolerance -This box defines a tolerance value to limit the display to only those items that fall within the specified limits.

- If you select **Point projection** and **All results**, you can specify a **Tolerance** value. PC-DMIS then displays all CAD entities within the tolerance distance.
- If you select **Ray pierce** and **All results**, you can specify a **Tolerance**. For curve geometry and surface boundaries, PC-DMIS displays any curve within the tolerance distance of the ray vector.

Output Vector - This area controls the displayed vector for surface boundaries and curves. (For surface points within its boundaries, the displayed vector is always the surface normal.)

- Select **Tangent** to display the curve point's tangent vector.
- Select **Normal** to display the curve point's normal vector. For curves, the normal vector is the inverse of the second-derivative vector. For surfaces, the normal vector is simply the surface normal.

Input CAD area

This area lets you choose the CAD entities you want tested.

Entire CAD Model - This check box determines whether PC-DMIS tests the entire CAD model or only selected CAD entities.

- If you select this check box, PC-DMIS enables other check boxes that allow you to define which entity types to test in the entire CAD Model: **Points**, **Curves**, **Surfaces**, or **All**.
- If you clear this check box, only specific CAD entities are tested. See "Sequence/ Source Sequence" below.

Points - This check box tests for all points on the entire CAD model.

Curves - This check box tests for all curves on the entire CAD model.

Surfaces - This check box tests for all surfaces on the entire CAD model.

All - This check box tests for all points, curves, and surfaces on the entire CAD model.

Sequence / Source Sequence - These boxes let you define a single CAD entity. You can fill in these boxes and then click the **Select CAD Object** button. PC-DMIS turns the item red and causes it to flash a few times.

Alternately, you can test multiple CAD entities by selecting them one at a time from the Graphic Display window, or by box-selecting a group of entities.

Output area

The **Output** area contains the results of the verification in a table format with points making up the rows. PC-DMIS displays all the points in the Graphic Display window. Selected points from this list are highlighted in the Graphic Display window. To update the information in the **Output** area, select a new option and press TAB.

This list describes the column headings in the **Output** area:

Point - This column shows the resulting CAD point from the input point and the CAD.

Vector - This column shows the resulting CAD vector from the input point and the CAD.

Result - This can be either **Hit** or **Miss**.

For **Point projection**,

- **Hit** means that the projection point was on the interior of the CAD entity.
- **Miss** means that the projection point was on the extremities of the CAD entity. For curves, the extremities are the end-points. For surfaces, the extremities are the boundaries.

For **Ray pierce**,

- **Hit** means that the ray vector directly intersected the CAD entity.
- **Miss** means that the ray vector passed close to, but did not directly intersect, the CAD entity.

For **Plane cut**,

- **Hit** means that the plane directly intersected the CAD entity.
- **Miss** means that the plane passed close to, but did not directly intersect, the CAD entity.

Sequence - This column shows which CAD entity the point is on. The sequence is a unique identifier assigned to each CAD entity.

Distance - This column shows the distance between the input point and the output point.

Copy to Clipboard

This button copies the **Output** area's results to the Windows Clipboard. If you selected specific points, only information for those points is copied. Otherwise, the entire output is copied.

Using Screen Captures of the Graphic Display Window

You can take screen captures of the Graphic Display window and then send them to the Clipboard, send them to the report, or save them to your computer. You can also add a captured image into the PC-DMIS Edit window as an external object. For more information, see "Inserting External Objects" in the "Adding External Elements" chapter.

Sending Screen Captures to the Clipboard

The **Operation | Graphic Display Window | Screen Capture To | Clipboard** menu option captures the Graphic Display window and copies the screen capture to the clipboard. The image remains on the clipboard until another screen is captured, or you close the measurement routine.

To view an image captured to the clipboard, paste it into a supporting application such as Microsoft Paint or Microsoft Word.

Sending Screen Captures to the Report

See "Screen Captures" in the "Using the Edit Window" chapter.

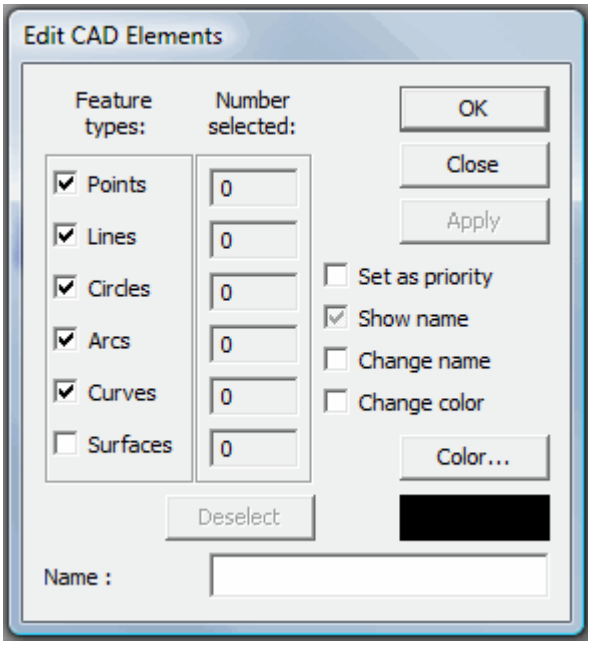
Sending Screen Captures to a File

The **Operation | Graphic Display Window | Screen Capture To | File** menu option captures the Graphic Display window and opens a **Save As** dialog box so you can save the screen capture as a bitmap file to a folder on your computer.

Editing Screen Captures

By itself, PC-DMIS does not have the ability to edit or format a captured image. However, you can use any image editor software to edit or format your screen captures.

Editing CAD Elements



Edit CAD Dialog Box

The **Edit CAD Elements** dialog box (**Edit | Graphic Display Window | CAD Elements**) allows you to alter the priority, element name, display of name, or color for different CAD elements in the CAD file.

To use this dialog box:

1. From the **Feature types** area, select the desired CAD element feature types to change.
2. From the Graphic Display window, select the CAD elements to change.
3. To select multiple CAD elements, box-select the desired CAD elements. When you release the mouse, PC-DMIS highlights the current selected elements and indicates the number of features in the **Number Selected** box. You can choose additional elements in the same manner. If needed, click the **Deselect** button to clear the screen of any highlighted elements.
4. Once the desired elements are selected and highlighted, mark the check box of the operation you want to perform:
    - **Set as priority**
    - **Show name**
    - **Change name**
    - **Change color**

5. See the "Dialog Box Description" subtopic below for any additional instructions and information on the individual operations.

6. Click **Apply** or **OK**.

In order for this option to work, you must create CAD levels. See "Setting Up the Screen View" for more information.

Dialog Box Description

The dialog box contains these items:

Feature types - Determines what feature types to edit. The available types are:

- Points
- Lines
- Circles
- Arcs
- Curves
- Surfaces

Number selected - Shows the number of selected CAD elements for each type.

Deselect - Deselects any highlighted CAD elements.

Name - Specifies a name for the selected CAD elements.

Set as priority - Determines the surfaces PC-DMIS checks first when calculating nominal values. For more information see "About Priority Surfaces" below.

Show name - Shows or hides the CAD element feature name (if provided by the CAD designer) associated with the selected CAD elements.

Change name - If marked, this changes the name of the selected CAD elements to the name provided in the **Name** box.

Change color - Draws the selected CAD data in the color shown in the box near the **Color** button once you click **Apply** or **OK**. To change the current color, select the **Color** button.

Color - Defines a color to apply to selected CAD elements by displaying a basic **Color** dialog box.

Once you click **OK** on this dialog box, it becomes the selected color in the **Edit CAD Elements** dialog box.

About Priority Surfaces

You can use the **Set as Priority** check box on the **Edit CAD Elements** dialog box to define a prioritized set of CAD surfaces for Find Noms (find nominals) processing. These are called "priority surfaces".

Prioritizing surfaces speeds up the calculation of nominal values as it allows PC-DMIS to determine which surfaces to check first. There is no limit to the number of surfaces that you can select. The order that you select the surfaces determines the order of the search.

Whenever you open the **Edit CAD Elements** dialog box and select the **Set as Priority** check box, PC-DMIS automatically selects the previously stored set of surfaces.

Setting Priority Surfaces

1. Select **Edit | Graphic Display Window | CAD Elements**.
2. If the **Set as priority** check-box is not marked, mark it now. PC-DMIS highlights any surfaces that are already prioritized.
3. Select surfaces in the Graphic Display window to add or remove them to this list.
4. Click **Apply** or **OK** button to update the measurement routine with the changes made to the list of priority surfaces.

Storing a New Set of Priority Surfaces

1. Open the **Edit CAD Elements** dialog box (**Edit | Graphic Display Window | CAD Elements**).

2. Select the **Set as Priority** check box. If you've already defined a set of surfaces, PC-DMIS selects them in the Graphic Display window.
3. With the **Set as Priority** check box selected, click the **Deselect** button and then click the **Apply** button. This essentially tells PC-DMIS to clear any stored surfaces.
4. Deselect the **Set as Priority** check box.
5. Select the new set of surfaces to store.
6. Select the **Set as Priority** check box.
7. Click the **Apply** button.
8. Click **OK**.

Box-Selecting and Priority Surfaces

When you box-select with the **Auto Features** dialog box open, PC-DMIS selects all CAD surfaces of the feature type inside the box.

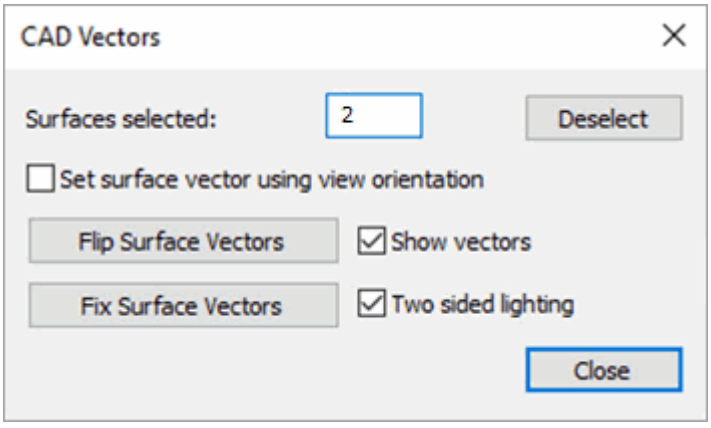
To limit which surfaces PC-DMIS selects inside the box, you can specify the priority surfaces. That is, if you've defined priority surfaces using the **Edit CAD Elements** dialog box, when you box-select from a surface Auto feature, PC-DMIS considers only the priority surfaces for that feature.

PC-DMIS uses priority surfaces for other functionalities, including Find Noms.

For more information on how to box-select to create features, see "Box Selecting to Create Multiple Auto Features" in the "Creating Auto Features" chapter.

Editing CAD Vectors

The **Edit | Graphic Display Window | CAD Vectors** menu item opens the **CAD Vectors** dialog box. With this dialog box, you can display and manipulate surface vectors.



CAD Vectors Dialog Box

Surfaces selected

With the dialog box open, click on a CAD surface in the Graphic Display window to select or deselect that surface. You can also drag a box across a portion of your part to box-select several surfaces at once. PC-DMIS highlights any selected surface and shows the number of selected surfaces in the **Surfaces selected** box.

Deselect

The **Deselect** button clears all selected CAD surfaces.

Set surface vector using view orientation

When you select this check box, the software flips the normal vector to make it point towards you as you select CAD surfaces.

For example, deselect the **Two sided lighting** check box, select the **Set surface vector using view orientation** and then click on a dark-colored CAD surface in the Graphic Display window. The software flips the surface normal towards you and away from the CAD model.

Flip Surface Vectors

The **Flip Surface Vectors** button flips your surface vectors, which causes the vector arrows to point in the opposite direction. PC-DMIS then clears the selected surfaces, and then redraws them with the flipped surface vector the next time you select them.

Show vectors

If you select the **Show vectors** check box, PC-DMIS displays a green arrow on each selected CAD surface to represent its vector.

Fix Surface Vectors

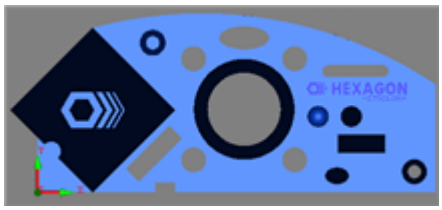
Some surfaces, when you initially select them, may have improper vectors. A proper vector points away from the part.

The **Fix Surface Vectors** button uses internal mathematical algorithms to correct selected surface vectors, which automatically causes the vector arrows to point in the proper direction. PC-DMIS then clears the selected surfaces. The next time you select them, they will appear with the corrected surface vector.

To fix the surface vectors of the entire CAD model, follow these steps:

1. Select the **Show Graphic Window as Solid** icon ( ) from the **Graphic View** toolbar to display your CAD model in Solid view.
2. Select the **Edit | Graphic Display Window | Lighting, Materials** menu item. The **CAD and Graphic Setup** dialog box opens.
3. Select the **Lighting** tab.
4. Clear the **Two sided lighting** check box.

5. Click the **OK** button. In the Graphic Display window, notice how some of the CAD surfaces appear dark. This is a visual indication that the surface normal for that surface is pointing in the wrong direction.



6. Select **Edit | Graphic Display Window | CAD Vectors**. The **CAD Vectors** dialog box opens.

7. Click the **Fix Surface Vectors** button. If the algorithm can fix all of the surface vectors, PC-DMIS changes the dark surfaces to the part's usual color, signifying that the surface vectors now point in the proper direction outside the part. If any of the surfaces remain with a dark shade, you can select those surfaces and manually flip their vectors by using the **Flip Surface Vectors** button.

To fix only specific surfaces on the model, follow the above procedure, but before you click **Fix Surface Vectors**, select the specific surfaces.

Two sided lighting

This check box determines whether the software lights the front and back faces of surfaces. For details on how to change CAD and graphics lighting settings, see the "Lighting Tab" topic in the "Setting Up CAD and Graphics" section of this documentation.

Close

This button closes the dialog box and clears any selected vectors.

Selecting Features Using the Graphic Display Window

You can use the Graphic Display window to select features and elements in the Graphic Display window (for example, to print, create coordinate systems, construct new features, calculate dimensions, and so on).

You can select features in any of these five basic ways:

- Use the ID(s)
- Use meta-character matching
- Select the last ID(s)
- Box-select ID(s)

- Select features in Online mode

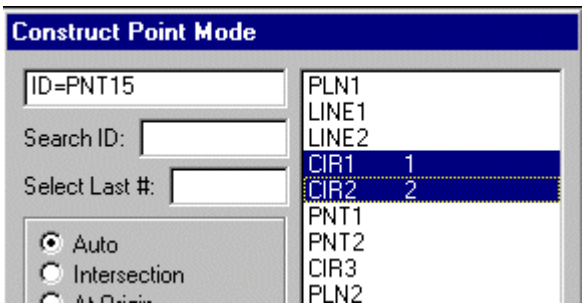
Selected features are indicated by the current highlight color. To deselect a selected (or highlighted) feature, click the selected feature once. To re-select it, click the feature again.

Selecting Features Using the ID(s)

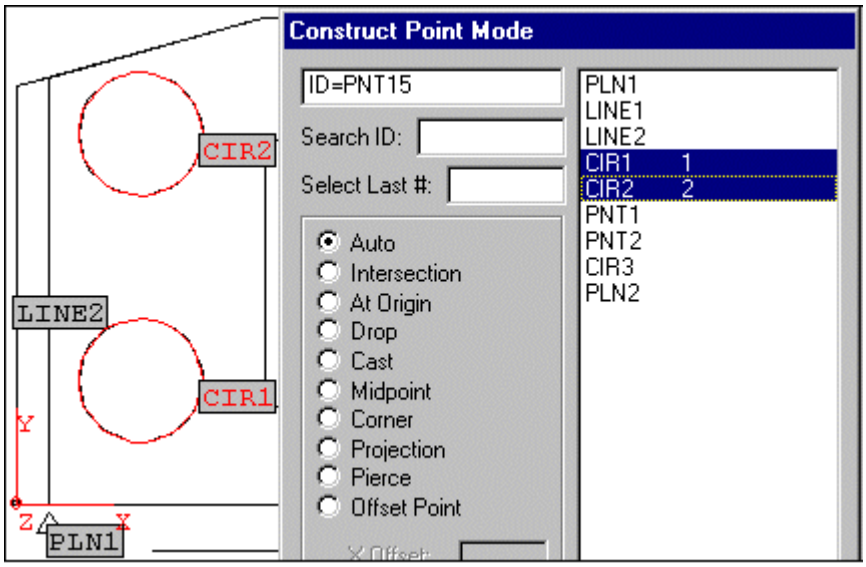
This selects the desired feature using the ID.

1. Type the ID(s) assigned to the feature(s) in the **Search ID** box.

2. Click on the desired ID from the list of features. PC-DMIS automatically assigns each feature a number in order of selection indicated to the right of the feature ID.



3. While in the Graphic Display window, move the cursor over the feature desired and click the left mouse button. (Note that the feature you selected in the Graphic Display window is now highlighted (or selected) in the feature window.)



Selecting Features Using Meta-Character Matching

This selects features using a meta-character. Meta-characters are characters that act as wildcard characters for other alphanumeric characters. There are two meta-characters available in PC-DMIS. They are the:

1. Asterisk (*)
2. Question mark (?)

Both are described below in greater detail.

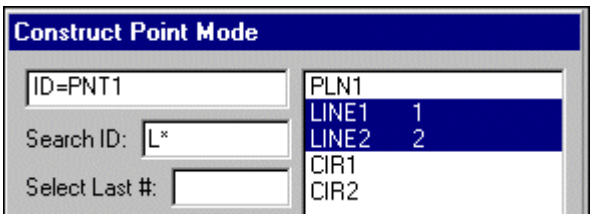
The Asterisk (*) Meta Character

***** The asterisk meta-character (*) matches, or takes the place, of any character or characters in a search.

For example, imagine these features are available in the feature list:

- PLN1

- LINE1
- LINE2
- CIR1
- CIR2



Asterisk (*) meta-character

To select all the line features (LINE1 and LINE2), in the **Search ID** box type **L***, and then press the Tab key. PC-DMIS selects all features beginning with "L".

To use the asterisk (*) meta-character in your feature selection:

1. Place your cursor in the **Search ID** box.
2. Type your search criteria using the asterisk (*).
3. Press the Tab key.

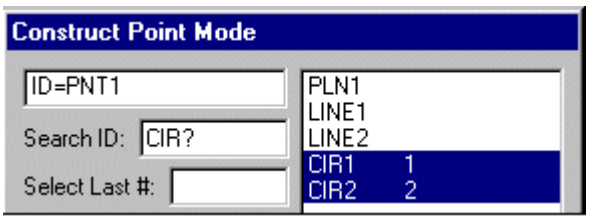
PC-DMIS selects the features that meet the search criteria.

You can use the asterisk (*) meta-character multiple times in any one search. You can also use it with the question mark (?) meta-character.

The Question Mark (?) Meta Character

? - The question mark meta-character (?) acts in the same way as the asterisk (*), except that the question mark meta-character matches *only 1* alphanumeric character. For example, suppose in your list of features, you have the following:

- PLN1
- LINE1
- LINE2
- CIR1
- CIR2



Question mark (?) meta-character

For example, if you want to select all the circle features (CIR1 and CIR2) using the question mark (?) meta-character, in the **Search ID** box, type **CIR?** and press the Tab key. PC-DMIS looks for all features that meet the search criteria, which in this case, is a feature with an ID 4 characters long with the first 3 characters being "CIR".

To use the question mark (?) meta-character:

1. Place your cursor in the **Search ID** box.
2. Type search criteria using the question mark (?).
3. Press Tab.

PC-DMIS selects the features that meet the search criteria.

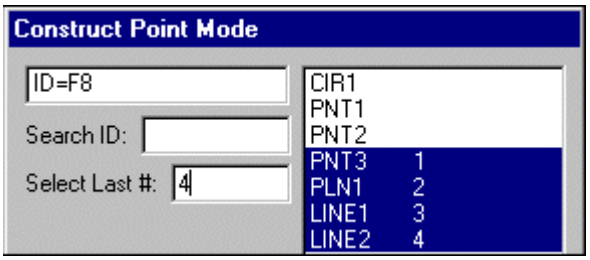
You can use the question mark (?) meta-character multiple times in any one search. You can also use it with the asterisk (*) meta-character.

Selecting the Last ID(s)

There are several alternative methods for selecting features in the feature list box, located in many dialog boxes. This instructs PC-DMIS to use the last 'number' of features in the current operation.

To select the last number of features:

1. From the dialog box, place your cursor in the **Select Last #** box.
2. Type the *last* number of features to be used. For example, to create a line from the last four features measured type **4** in the box.

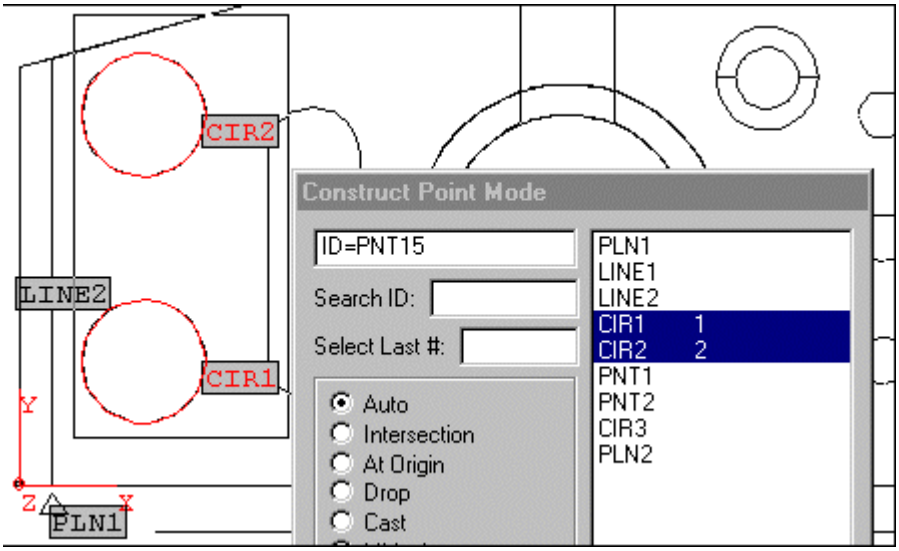


Select Last ID

Box-Selecting ID(s)

Another way to select features is to use the left mouse button to draw a "box" around the features. This is called "box-selecting". To do this:

1. Place the mouse pointer at one corner of where the box is to be drawn, and hold down the left mouse button.
2. Drag the mouse pointer to the opposite corner of the "box". PC-DMIS draws the box outline on the screen as you drag the mouse.
3. When the "box" is satisfactory, release the left mouse button. PC-DMIS highlights all the elements in the box, and lists them in the feature list box.



Box-Selecting CIR1 and CIR2

This option also allows you to modify the contents of the box as indicated in the list. To either add or delete a feature from the set, click on the feature to be changed (either in the Graphic Display window, or the dialog box's list of features).

Overview of Box-Selecting Sheet Metal Feature IDs

The ability of PC-DMIS to "box-select" feature identifications now allows a union of sheet metal features between two selected groups. This functionality makes the selection of multiple three-dimensional objects much easier. Additionally, if any CAD objects are box-selected and then a CAD object is chosen without box selection, those objects that were box-selected become permanent selections in addition to the single selection.

For specific information on how to use this capability, see "Box-Selecting to Create Multiple Auto Features" in the "Creating Auto Features" chapter.

Selecting Features in Online Mode

With PC-DMIS in Online mode, use the active tip as a pointer, and trigger the probe when the tip is close to the feature needed.

Editing a Feature ID

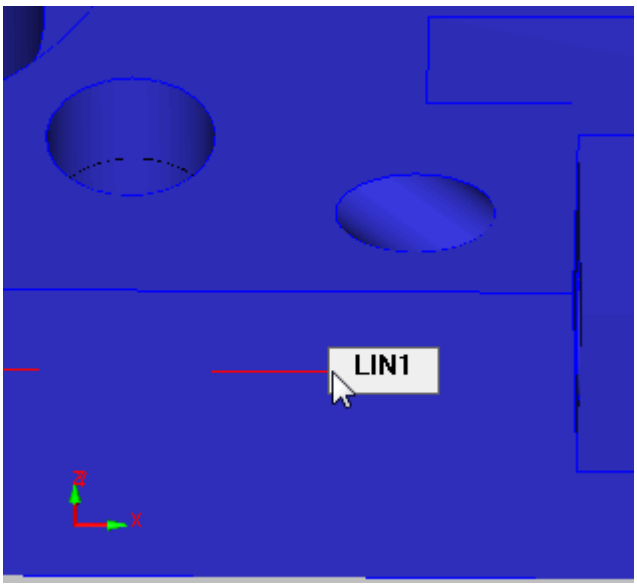
Edit ID Dialog Box

To change a feature's identification, double-click on the desired feature ID in the feature list. PC-DMIS displays the **Edit ID** dialog box. This dialog box allows you to rename the selected feature's ID.

Never use any of the mathematical characters (-, +, /, or *) in an ID. This causes problems when you attempt to use the ID inside PC-DMIS expressions. For the full list of ID rules, see "Setup Options: ID Setup tab" in the "Setting Your Preferences" chapter.

Identifying a Feature by Using Tooltips

You can also identify a feature without having to show feature IDs. Instead, PC-DMIS can display a small tooltip that appears when you briefly hover your mouse pointer over the feature inside the Graphic Display window. The tooltip remains visible until you move the mouse pointer off of the tooltip.



Sample Tooltip of a Line Feature

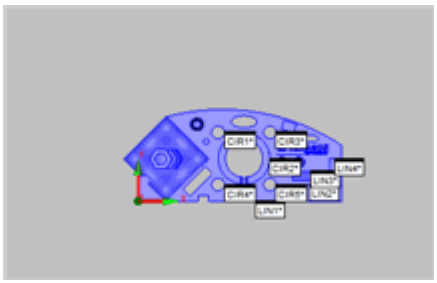
You may find this useful when you have a large measurement routine, and you want to keep your feature ID labels turned off to improve performance, yet you still need a way to quickly identify a feature.

- Click on the tooltip to toggle the feature's ID label in the current view.
- Right-mouse click to display a shortcut menu with the same options available when you right-click on a feature inside Text Box Mode.

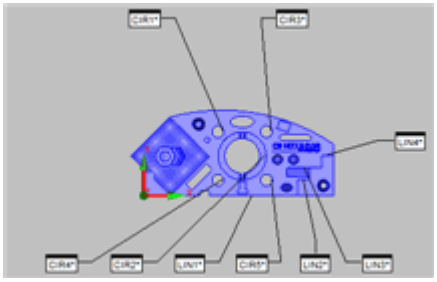
These tooltips are not available when PC-DMIS performs path operations (Animate Path, Edit Path, and other path operations), when in Quick Fixture Mode, or in any mode where you use the mouse button or keyboard buttons at the same time (such as during a Pan, Zoom, or Rotate operation).

Automatically Positioning Feature ID Labels

PC-DMIS provides you with these methods to automatically position feature ID labels in the Graphic Display window with leader lines so that they point to the features they reference instead of resting directly on the feature. This moves the labels to the edge of your CAD view so that you can more easily see the part or feature.



Example that shows the ID labels without automatic positioning.



Example that shows ID labels with automatic positioning.

Method 1 - Use the Setup Options Dialog Box

Access the **Setup Options** dialog box (**Edit | Preferences | Setup**), and from the list of check boxes on the **General** tab, enable the **Automatic label positioning** check box. This method keeps repositioning your labels whenever you perform pan, zoom, or rotate operations on the part. It only functions with the main CAD view if you have multiple split views.

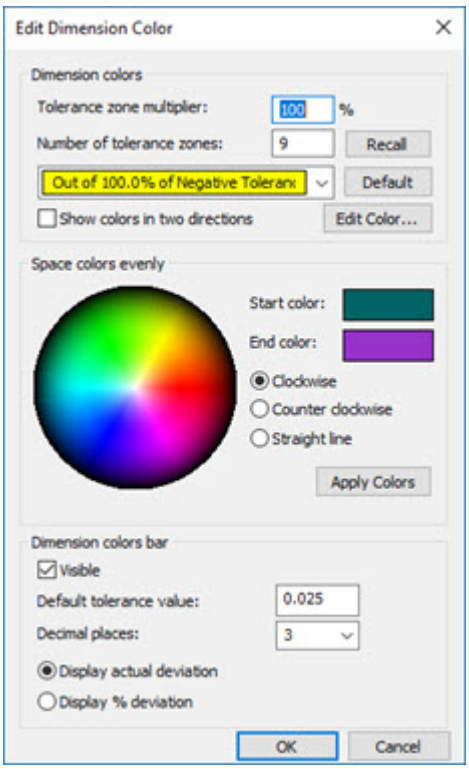
Method 2 - Use the Feature ID Label Shortcut Menu

Right-click on a feature ID label, and from the shortcut menu, select **Automatic Label Positioning**. This method, different from method 1, works in the current CAD view, not just the main CAD view. Also, it only positions the labels once. So, if you perform pan, zoom, or rotate operations, the labels are not repositioned.

Editing Dimension Colors

Select the **Edit | Graphic Display Window | Dimension Colors** menu option to open the **Edit Dimension Colors** dialog box. This dialog box defines the colors and tolerance zones for your entire tolerance band.

You can also right-click the **Dimension Colors Bar** that is visible in the Graphic Display window to open the **Edit Dimension Colors** dialog box.



Edit Dimension Colors Dialog Box

Defining Dimension Colors

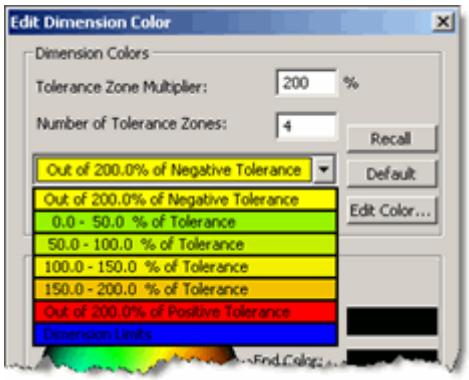
1. Set the **Tolerance zone multiplier** value as needed.
2. Type a value in the **Number of tolerance zones** box to define the number of tolerance zones.
3. If desired, select the **Show colors in two directions** check box to extend your color range to the -Tol value.
4. Pick the colors for the tolerance zones in one of two ways:
    - *Method 1* - Select each tolerance zone one at a time from the tolerance zone drop-down list, and click the **Edit** button to set a specific color for each zone.
    - *Method 2* - Use the color wheel to select the start and end tolerance zone colors, and allow PC-DMIS to define colors for the other zones.
5. Click **Apply Colors**.
6. Modify the options for the **Dimension colors bar** as desired.
7. Click **OK**.

Dialog Box Description

Tolerance zone multiplier - This value increases the negative and positive tolerance by the percentage specified. This value provides you with greater control over the tolerance range for your tolerance zone. This allows your range of colors to extend into the out-of-tolerance zone and lets you visually see how much something is out of tolerance.

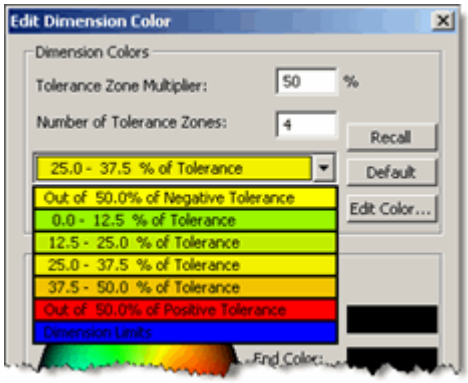
If this option is set to 200%, the range of tolerance zones is 2.0*(-Tol) through 2.0*(+Tol).

For example, if you give this option a value of 200%, the tolerance zone doubles its normal range:



Example that shows the Tolerance Zone Multiplier set to 200%.

If you type 50%, the zone is half its normal range:



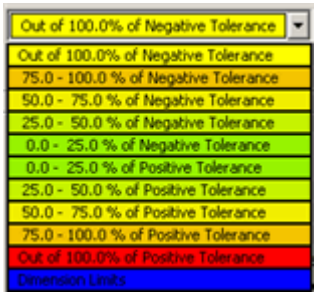
Example that shows the Tolerance Zone Multiplier set to 50%.

The default value is 100%.

Number of tolerance zones - This value determines the number of tolerance zones for your measurement routine. Tolerance zones divide your entire tolerance band by this number to create the different zones. Each tolerance zone has a unique color associated with it.

Tolerance zones - This list contains all the tolerance zones. You can select a specific zone from this list to manipulate its color in detail. The **Dimension Limits** item from this list is used to create a border at the absolute positive or negative limit of the feature tolerance. The color used for this corresponds to the color used to draw the tolerance band.

Show colors in two directions - This option determines whether or not your tolerance zones for the current measurement routine display the same color range in two directions, one heading towards the **Positive Out of Tolerance** range and the other heading towards the **Negative Out of Tolerance** range. This effectively expands the lower dimension color range to –Tol instead of zero.



Example showing tolerances displayed in two directions.

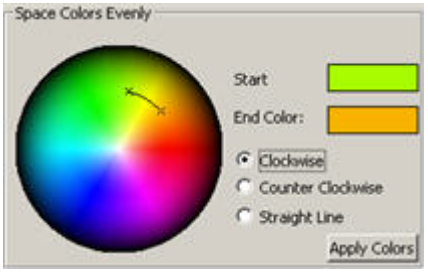
You can then use the **Edit** button to define the negative dimension colors and positive dimension colors separately.

Recall - This button returns the colors to the default setting.

Default - This button overrides the previous default color settings with the current values.

Edit Color - This button opens a **Color** dialog box so that you can change the color associated with the currently-selected tolerance zone.

Color Wheel - The color wheel provides you with a quick way to define colors for all your tolerance zones. It lets you pick the colors for the first and last tolerance zones. A small x appears on the color wheel for each color selected. The remaining tolerance zone colors are spaced evenly between the start and end colors. The direction of the spacing depends upon the option selected.



Example showing the Color Wheel with selected start and end colors.

Clockwise - This option sets the end color to have the same brightness as the start color and spaces the colors in a clockwise direction.

Counter clockwise - This option sets the end color to have the same brightness as the start color, but spaces the colors in the counter-clockwise direction.

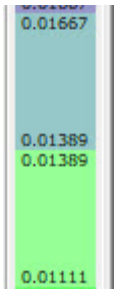
Straight line - This option spaces the colors from the start color to the end color, regardless of the brightness, in a straight line direction.

Apply Colors - This button applies any color changes you made without closing the dialog box, which allows you to immediately test your color selection.

Visible - This check box lets you show or hide the **Dimension Colors Bar** once you click **OK**. For information on the **Dimension Colors Bar**, see the "Using the Dimension Colors Window" topic in the "Using Other Windows, Editors, and Tools" chapter.

Default tolerance value - This box lets you change the default tolerance value for the **Dimension Colors Bar**.

Decimal places - This value changes the number of decimal places displayed in the **Dimension Colors Bar** in the Graphic Display window.



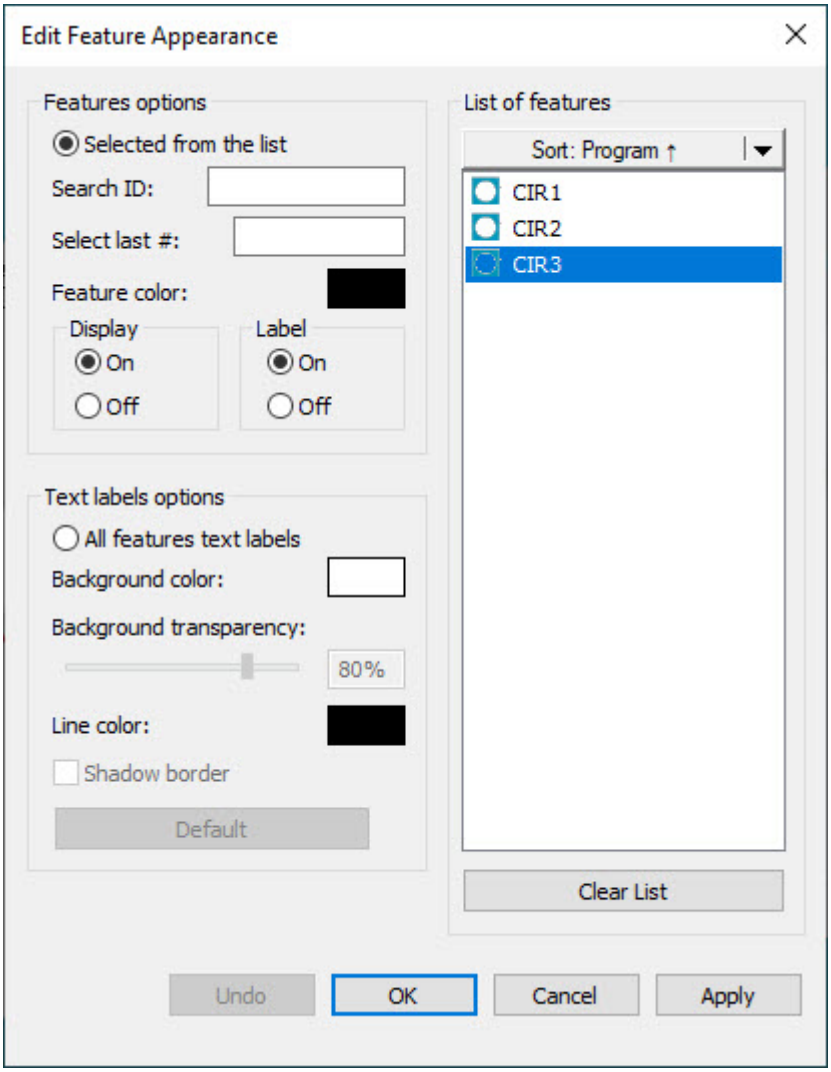
Example of the Dimension Colors Bar with five decimal places.

Note that this value does not affect the decimal places used by the **Dimension Color Key** object used in reporting. The decimal places for the Dimension Color Key object in reporting are hard coded to three decimal places.

Display actual deviation - This option displays tolerances as the actual deviation in the **Dimension Colors Bar**.

Display % deviation - This option displays tolerances as a percentage of the deviation in the **Dimension Colors Bar** for the current measurement routine.

Editing Feature Appearance

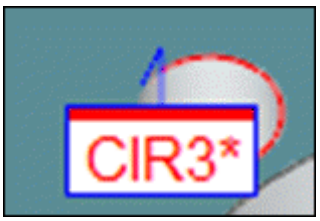


Edit Feature Appearance Dialog Box

Select the **Edit | Graphic Display Window | Feature Appearance** menu option to open the **Edit Feature Appearance** dialog box. You can use this dialog box to alter feature identifications (termed "feature IDs") within a measurement routine, feature colors, and feature ID labels.

The following example shows how to modify a legacy feature ID label and a feature with the **Edit Feature Appearance** dialog box. Various colors demonstrate the different parts of the label.

- Red - A modified **Feature color**. By default, the colored bar at the top of the ID label also changes to match the color of the circle.
- White - The default **Background color**.
- Blue - A modified **Line color**.



Example showing a feature ID label and a feature with a modified appearance.

Editing Feature IDs

Double-click the desired feature in the list box to edit a previously-assigned feature name. The software opens the **Edit ID** dialog box which you can use to type a new identification.

Edit ID Dialog Box

You can also change feature IDs in the Edit window. Highlight the desired ID, and type a new identification. If you type an ID that another feature already uses, PC-DMIS displays a message and keeps the previous ID.

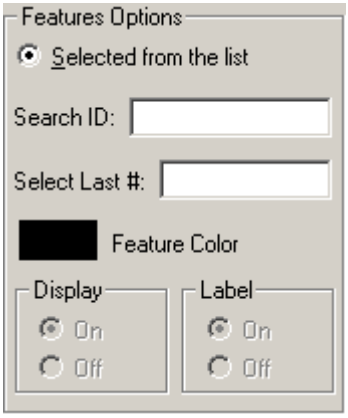
Never use any of the mathematical characters (-, +, /, or *) in an ID. This causes problems when you attempt to use the ID inside PC-DMIS expressions. For the full list of ID rules, see "Setup Options: ID Setup tab" in the "Setting Your Preferences" chapter.

This dialog box contains two main areas:

- **Features options** - You can use this area to modify the display of the part's features.
- **Text labels options** - You can use this area to modify the feature ID labels in the Graphic Display window.

To change the graphic options for features or feature ID labels, select the desired options from the dialog box, click **Apply** and then **OK**.

Features Options Area



The **Features Options** area allows you to change the color of selected part features and whether or not selected features or feature ID labels are displayed in the Graphic Display window. To use the items in this area, you must first select one or more items from the **List of Features** and then click the **Selected from the list** option. This enables the other items in this area.

The **Search ID** and **Select Last #** boxes are discussed in the "Search ID" and "Select Last #" topics in the "Navigating the User Interface" chapter.

The **Feature Color** option allows you to set the color for a specified feature's identification. This option works in the same manner as the **Edit Color** option (see the **Edit CAD** dialog box found in "Editing CAD").

To Change the Color of a Feature:

1. Select the feature to be altered from the **List of Features**.
2. Select the **Feature Color** option. PC-DMIS automatically displays the **Color** dialog box.
3. Click on the desired color, or define a new one using the **Custom Color** box.
4. Click the **OK** button. PC-DMIS returns to the **Edit Feature Appearance** dialog box. The new color is displayed in the **Feature Color** box.
5. Click the **Apply** button and PC-DMIS automatically changes the color of the feature in the Graphic Display window. The top border of the feature ID label also changes to match the feature color.

The **Display** area contains **On** and **Off** options that control the display of features in the Graphic Display window. This option is useful when you want to focus on only a portion of geometrical features the part contains at a time.

To Determine the Display of Selected Features:

1. Select the feature or features that you want to turn on or off.

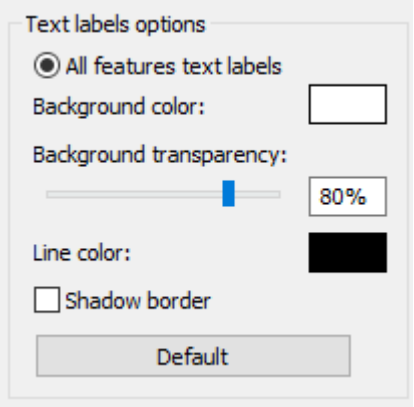
2. From the **Display** area, select either the **ON** or the **OFF** option.
3. Click the **Apply** button. PC-DMIS displays or conceals the features.
4. If the changes made are to your satisfaction, click the **OK** button. PC-DMIS closes the dialog box and saves the changes.

The **Label** area contains **On** and **Off** options that control the display of feature IDs in the Graphic Display window. This option is useful when a portion of the part drawing becomes cluttered with several feature IDs. You can hide Specific IDs, allowing other IDs to be more easily viewed.

To Determine the Display of Selected Feature ID Labels:

1. Select the feature label or labels that you want to turn on or off.
2. From the **Label** area, select either the **ON** or the **OFF** option.
3. Click the **Apply** button. PC-DMIS displays (or conceals) the feature labels.
4. If the changes made are to your satisfaction, click the **OK** button. PC-DMIS closes the dialog box and saves the changes.

Text Label Options Area



The **Text Label Options** area lets you set the graphic options for all feature ID labels. To use the features in this area, you must first select the **All features text labels** option. PC-DMIS automatically selects all the features from the **List of Features**.

Background Color
This box opens a **Color** dialog box so that you can change the label's background color.

Background Transparency
This slider or the box to its right adjusts the transparency level of the label's background color. The value can be from 0 (opaque) to 100 (completely transparent).

Line Color
This box opens a **Color** dialog box so that you can change the label's border color.

Shadow border
This check box displays a small shadow under the label.

Example Feature ID labels set to a pink background color with 50% transparency

You can set the default graphic options for future labels. To do this, select the desired graphic options and then click the **Default** button.

You can also change these items by right-clicking on a feature's ID label and selecting the **Change Appearance** menu option from the resulting shortcut menu. See "Feature Shortcut Menu" in "Using Shortcut Keys and Shortcut Menus".

Showing, Animating, and Moving Path Lines

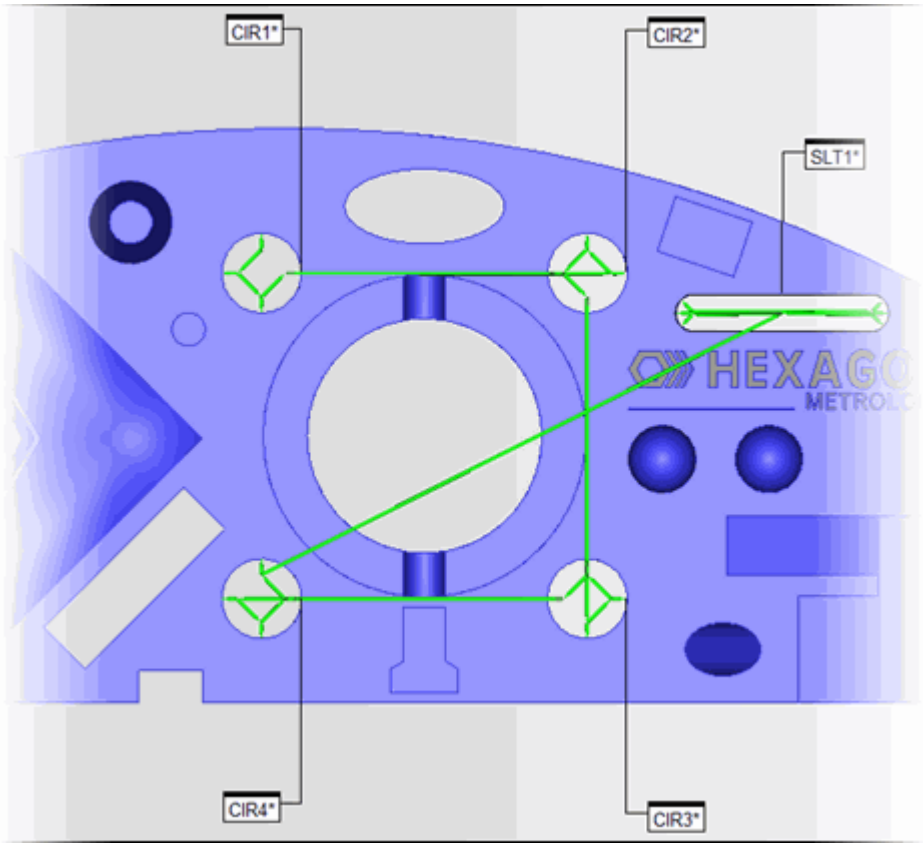
PC-DMIS can generate colored lines on the CAD model in the Graphic Display window that show the path your probe will take during execution as it measures marked features. These lines are called path lines. Path lines help you preview the path your probe will take and troubleshoot possible collision areas.

The software only generates the probe's path lines for features that follow a `MODE/DCC` command. If your measurement routine doesn't have a `MODE/DCC` command, you won't see any path lines.

The **Animate Path**, **Regenerate Path**, **Optimize Path**, and **Collision Detection** options are not available for Portable devices.

Viewing All Path Lines

To view path lines, select **View | Path Lines**. PC-DMIS checks each command and draws the path lines accordingly. You can press Esc to cancel this process at any time.



This example shows the probe's route drawn on the part's graphical image

Viewing Path Lines from a Cursor's Position

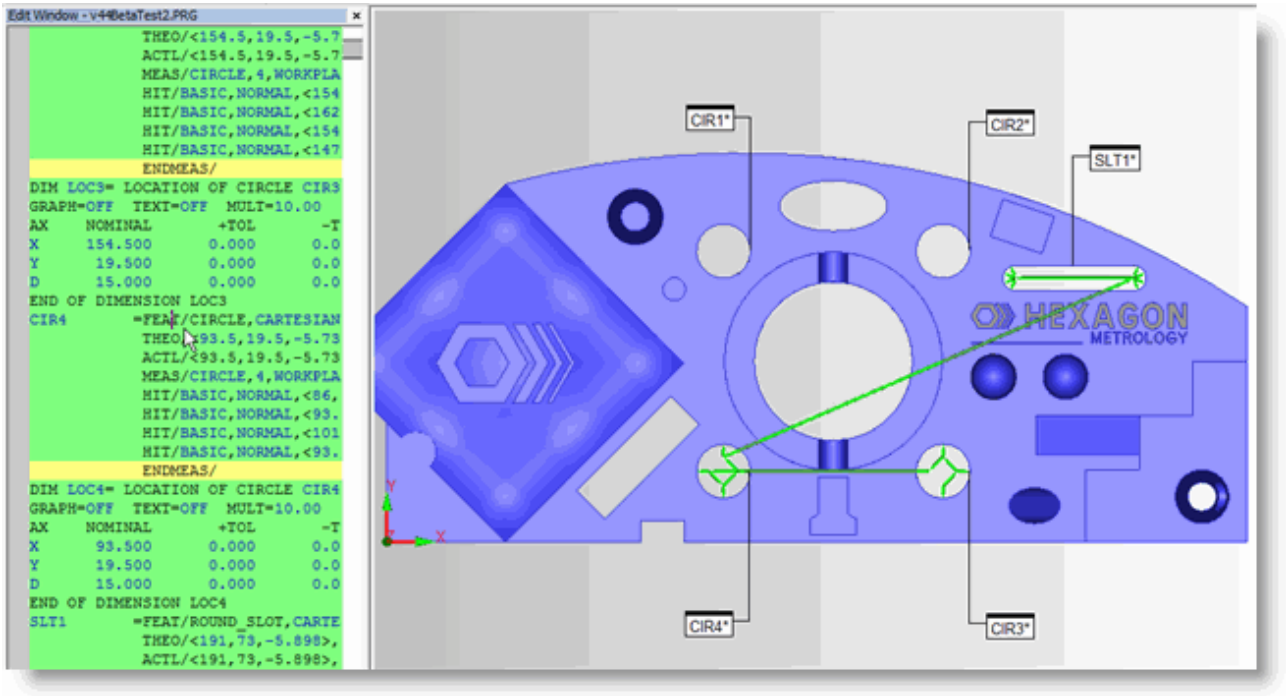
This only works if the Edit window is in Command Mode.

You can select **View | Path Lines from Cursor** to limit the displayed path lines to focus on the following:

- The feature where the cursor resides.
- The feature that precedes it.
- The feature that follows it.

You may find this most useful when stepping through a measurement routine.

For example, suppose your measurement routine has these features in this order: CIR1, CIR2, CIR3, CIR4, and SLT1. If you clicked on item CIR4 in Command Mode, the Graphic Display window draws path lines for CIR3, CIR4, and SLT1.



Example showing path lines drawn for the clicked feature, CIR4, its preceding feature, CIR3, and its following feature, SLT1

To modify the number of features that display their path lines within **Path Lines from Cursor** mode, edit the value in the **Create Path Feature Count** box located in the **Animation** tab of the **Setup Options** dialog box. The default value is 1, which means that PC-DMIS draws path lines for one feature preceding the current feature and one feature following the current feature. See the "Setup Options: Animation tab" topic in the "Setting Your Preferences" chapter.

The **Path Lines from Cursor** mode does not function with **Auto Insert Moves** operation (**Operation | Graphic Display Window | Clearance Moves | Auto Insert Moves**). If you perform an **Auto Insert Moves** operation, PC-DMIS reverts to displaying all the path lines. See "Inserting Clearance Moves Automatically" in the "Inserting Move Commands" chapter.

Viewing Path Lines for a Range of Selected Items

Another way of limiting the displayed path lines is to select a range of one or more features in the Edit window and show path lines used for only those selected features.

To do this:

1. In the Edit window, select the range of features.
    - If in Command Mode, click and drag the selection or click on the first feature, press Shift and then click on the second feature to establish the range of features.
    - If in Summary Mode, click on the first feature, press Shift and then click on the second feature to establish the range of features.
    - In either mode, you can press Ctrl and click to select non-sequential features.

2. Right-click in the Edit window. A shortcut menu appears.
3. From the shortcut menu, choose **Path | Path Lines**. PC-DMIS draws the path lines on the screen for the selected features. A check box appears next to the shortcut menu item.

If you want to select a different range of features, you need to right-click in the Edit window and, from the shortcut menu, clear the **Path | Path Lines** menu item. You can then re-select the **Path Lines** shortcut menu item for the new selection.

Changing a Path Line's Size

To change the size of your path lines:

1. Choose **Edit | Graphic Display Window | Display Symbols** to open the **CAD and Graphic Setup** dialog box.
2. Click the **Symbols tab**.
3. Modify the **Fixed Size** value.

During execution or collision detection operations, or during a part translation or rotation, PC-DMIS draws the path lines as simple lines (without the defined diameter) in order to speed up these operations.

Changing a Path Line's Color

By default, the software shows path lines in a green color.

To change the color:

1. Access the **Setup Options** dialog box
2. Click the **Animation** tab.
3. From the **Path Lines Color** box, choose a color.

See the "Setup Options: Animation tab" topic in the "Setting Your Preferences" chapter.

You can use the **Path** toolbar on the **QuickMeasure** toolbar (**View | Toolbars | QuickMeasure**) to work with common path line options. For more information, see the "CMM QuickMeasure Toolbar" topic in the PC-DMIS CMM documentation.

Animating the Path

The **Animate Path** option is not available for Portable devices.

The **Operation | Graphic Display Window | Animate Path** menu option animates the path of the probe on the part. The **Animate Path** option only simulates the movement of the probe. It does not execute the measurement routine.

To make **Animate Path** available for selection:

1. Mark the desired features.
2. Select the **View | Path Lines** menu item. The **Animate Path** option is then available for selection.

   The software only generates the probe's path lines for features that follow a `MODE/DCC` command. If your measurement routine doesn't have a `MODE/DCC` command, you won't see any path lines.
3. Select the **Animate Path** menu item. PC-DMIS displays the **Execution** dialog box and automatically begins animating the path with a simulated probe tip.

   - To stop the animation at any time, press the Esc key or click **Stop** on the **Execution** dialog box.
   - To continue a stopped animation, click **Continue** .
   - You can also press the Enter key to toggle between stop and continue when animating the path.
   - You can click on a path line to select that feature in the Edit window. This also provides an opportunity to add a move point along that path line. For more information on how to use move points to move a path line, see "Moving Path Lines".

Regenerating the Path

The **Operation | Graphic Display Window | Regenerate Path** menu option erases the current path lines and re-draws them.

You may want to regenerate your path when you've made changes to a measurement routine, and you want your path lines to reflect the current state of the measurement routine.

Notes

- **Regenerate Path** is only accessible *after* you mark features in the Edit window *and* you select the **View | Path Lines** option.

  The software only generates the probe's path lines for features that follow a `MODE/DCC` command. If your measurement routine doesn't have a `MODE/DCC` command, you won't see any path lines.

- **Regenerate Path** is not available for Portable devices.
- You can press Esc to cancel the regeneration at any time.
- If you insert a move before a PH9 wrist change, the location of the wrist change does not move until you choose **Regenerate Path**.

Optimizing the Path

For information on how to create an optimized path for your probe from the **Optimize Path Workflow** dialog box (**Edit | Optimize Path** or **Operation | Graphic Display Window | Optimize Path**), see the "Optimizing the Path" topic in the "Using Inspection Plans in PC-DMIS" appendix.

Detecting Collisions

The **Collision Detection** option is not available for Portable devices.

The **Operation | Graphic Display Window | Collision Detection** menu option (CD) displays the **Collision Detection** dialog box. The CD module detects collisions between the probe and CAD surfaces. It ignores all CAD curves, lines, and points. As a result, only CAD files that fully describe the part using surfaces are suitable for collision detection. (The CD algorithm doesn't use the actual surface definitions, but rather the surface tessellations [or graphical approximations], which are also used to render solid views using the OpenGL graphics language.)

Changing Animation Speeds: If you want to modify your offline animation speeds, see the **Execution** area on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). In addition, see the "Executing and Debugging Measurement Routines Offline" topic in the "Working in Offline Mode" chapter.

Viewing Collisions for a Range of Selected Items

You can also limit the collision detection items by selecting a range of one or more features in the Edit window, and perform the collision detection on only those selected features.

To do this:

1. In the Edit window, select the range of features. Either click and drag the selection (if in Command Mode), or click on the first feature, press the Shift key on your keyboard, and then click the second feature to establish the range of features. (If in Summary Mode, you can also press the Ctrl key instead of the Shift key on your keyboard.)

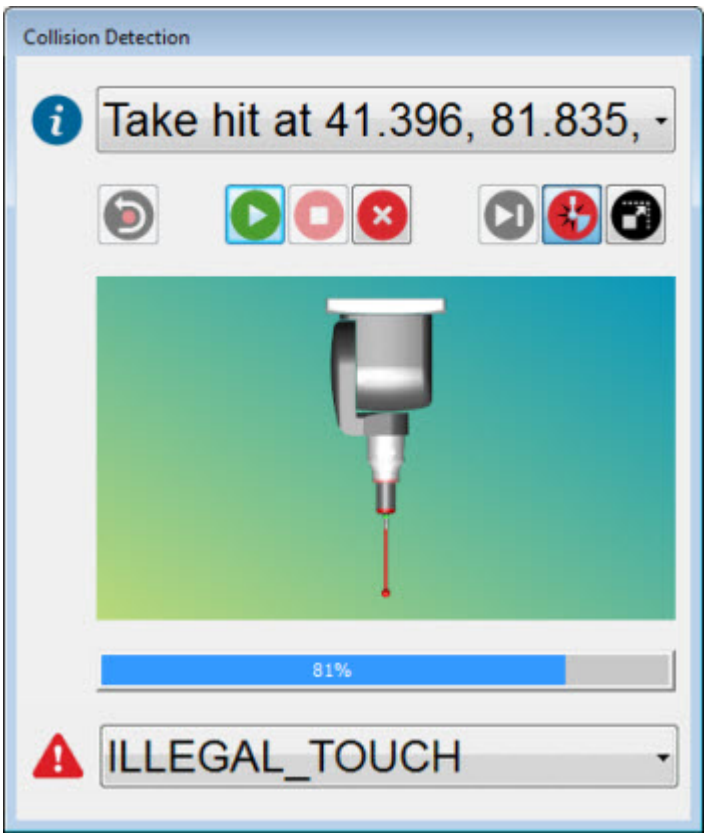
2. Right-click in the Edit window. A shortcut menu appears.

3. Choose **Selected Items Collision Detection**. PC-DMIS automatically performs the collision detection on those features and draws path lines as PC-DMIS processes the features. A check box then appears next to the shortcut menu item.

If you want to immediately select a different range of features, you need to right-click in the Edit window, clear the **Selected Items Collision Detection** first, and then re-select it for the new selection.

Collision Detection Dialog Box Options

Collisions are shown in the **Collision Detection** dialog box (**Operation** | **Graphic Display Window** | **Collision Detection**).



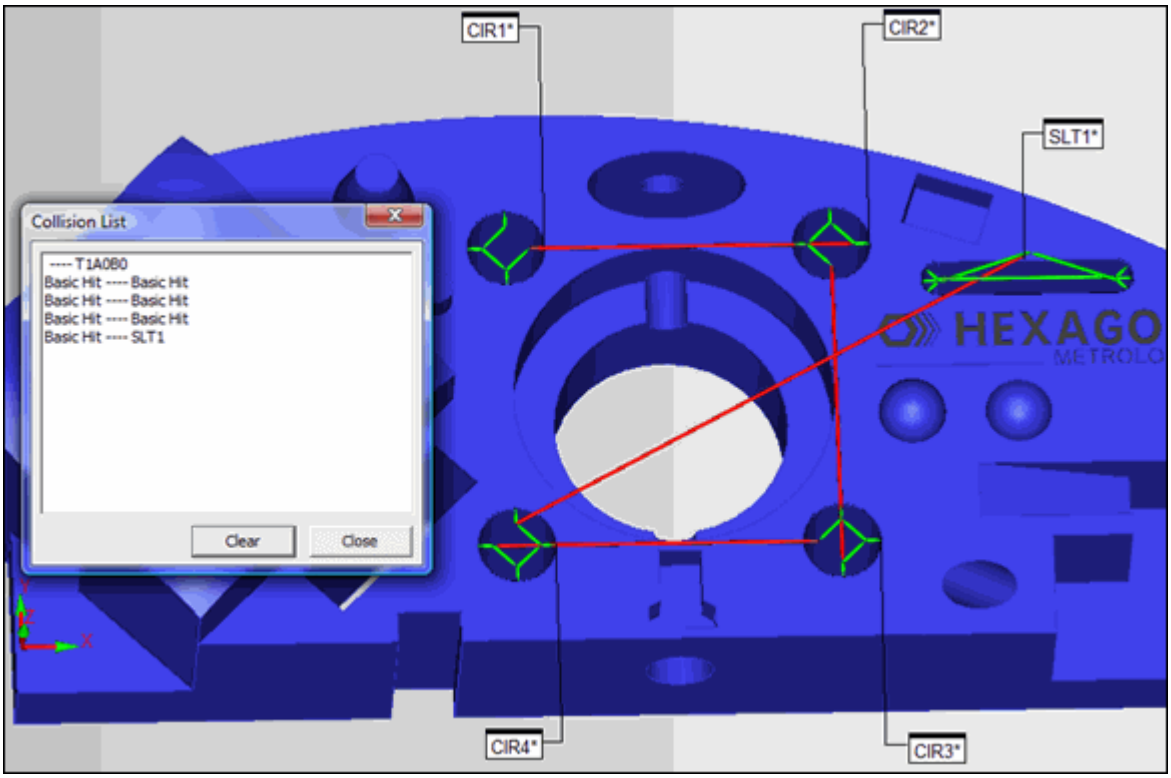
Collision Detection Dialog Box

Most of the items in this dialog box function like those in the **Execution** dialog box. For information on those items, see "Using the Execution Dialog Box" in the "Using Advanced File Options" chapter.

The following items only apply to collision detection:

- **Stop On Collision** - If selected (highlighted), the probe animation collision detection stops if a collision occurs.
- **Probe Display** - Immediately below the icons on the dialog box in the probe display area. The probe is shown using a green color. If a collision occurs, the portion of the probe that collides is shown in red. The same color scheme is used to draw the probe on the screen, as well as in the **Collision Detection** dialog box.
    - You can enlarge or shrink the drawing of the probe just as you would the part on the Graphic Display window by right-clicking above or below an imaginary horizontal line.
    - You can 3D rotate the drawing of the probe by holding down Ctrl key on your keyboard and the right-mouse button while dragging the mouse.
- **Scale to Fit** - Shrinks or enlarges the drawing of the probe so that it fits back into the probe display area of the dialog box.

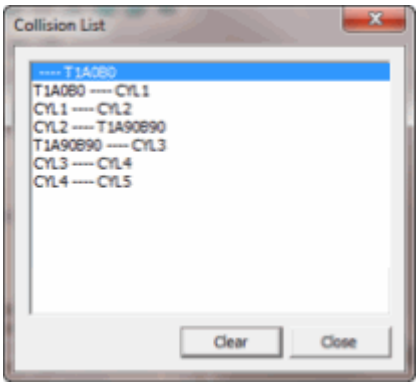
When PC-DMIS has finished running Collision Detection, it redraws the path lines in the Graphic Display window. PC-DMIS indicates where collisions have occurred by drawing affected path line segments in red (RGB color of 255,0,0). In addition, PC-DMIS displays a **Collision List** dialog box helping you to quickly locate collision problems in your measurement routine.



Example of the Graphic Display window showing the path lines and collisions (red lines).

Displaying a Collision List

The **Collision List** dialog box is displayed as soon as you select **Operation | Graphic Display Window | Collision Detection** menu option to start collision detection. This is helpful when collision detection needs to be stopped, identified collisions can be reviewed up to that point.



Collision List Dialog Box

This dialog box displays a list of collisions for the measurement routine. The Graphic Display window also highlights those path lines in red. If the Edit window is in Command Mode, you can click on an item on the **Collision List** dialog box to locate the command in the Edit window where collision was detected. You can then modify the measurement routine to fix the collision problem.

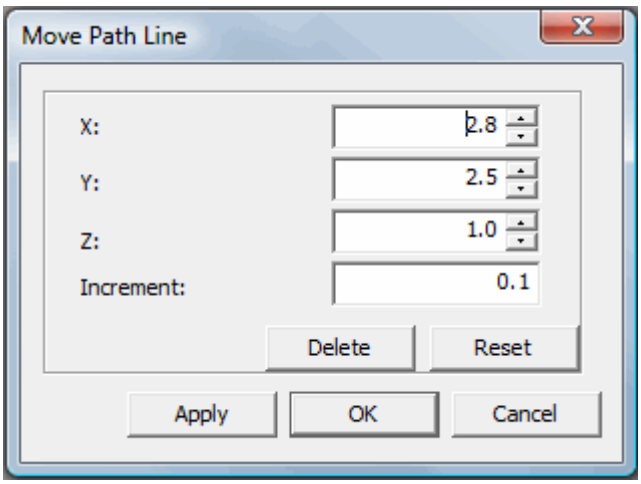
If you want to remove one or more items from the **Collision List** dialog box, select the item or items, and click the **Clear** button. You may find this useful if you want to remove an item you've already corrected or if you want to focus on a subset of a large list of detected collisions.

Moving Path Lines

Once you enable path lines and see them on your screen, you can change the route the probe follows as it moves between features. To do this, place PC-DMIS in Translate Mode and click on a highlighted path line. PC-DMIS splits the path line near where you clicked and adds a `MOVE/POINT` command at that location. Use the **Move Path Line** dialog box to adjust the move point location.

This method only works on the path lines *between* features. You can use a different method to reposition path lines *within* Auto features. For information, see "Show Hit Targets" in the "Creating Auto Features" chapter.

If you click on a path line between features and the **Move Path Line** dialog box still doesn't open, you may need to enable this functionality. To do this, from the **Setup Options** dialog box (**Edit | Preferences | Setup**), on the **Animation** tab, mark the **Path Line Moving Enabled** check box. For more information, see "Setup Options: Animation tab" in the "Setting Your Preferences" chapter.



Move Path Line Dialog Box

The dialog box contains **X**, **Y**, and **Z** axis boxes, allowing you to type a specific move point location, or you can use the **Increment** value with the small up and down arrows next to each axis box.

- When you click the up arrow, PC-DMIS adds the increment value to the current value.
- When you click the down arrow, PC-DMIS subtracts the increment value from the current value.

The selected path line is adjusted automatically in the Graphic Display window.

- **Delete** - This button deletes the selected move point. This button only becomes active when you click on or near a move point located along the path line.
- **Reset** - This button sets the path line back to normal as long as you haven't clicked **OK** or **Apply**.
- **Apply** - This button functions the same as the **OK** button except that the dialog box remains open, which allows you to continue to work with additional path lines.
- **OK** - This button accepts your changes and inserts a `MOVE/POINT` command into the Edit window at the appropriate location to reflect the adjusted path line and then closes the dialog box.

If you need to adjust the move point later, simply click on the point on the path line. The **Move Path Line** dialog box opens again, which allows you to change the values for the `MOVE/POINT` . Alternately, you can press F9 on the `MOVE/POINT` command inside the Edit window, and then use the **Move Point** dialog box to change the values.

It's often handy to rotate the part while you adjust path lines. The **Move Path Line** dialog box permits you to do 2D and 3D rotate operations with the standard Alt + right-click drag and Ctrl + right-click drag methods, respectively.

The only way to remove a path line modification is to delete the inserted `MOVE/POINT` command from the Edit window.

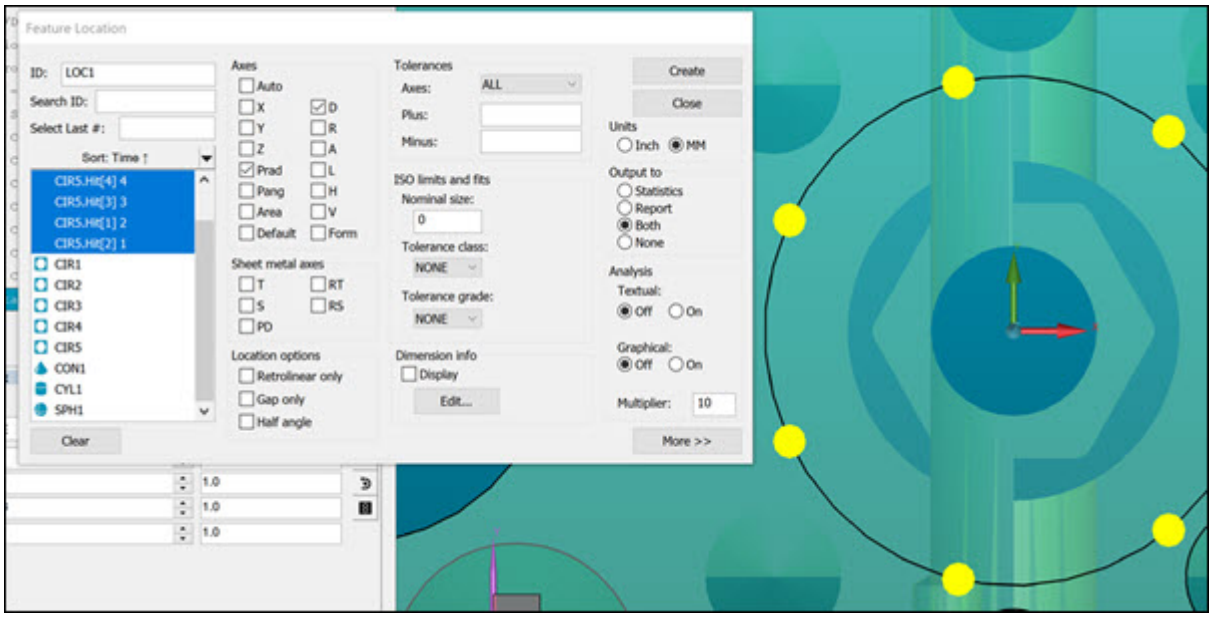
Graphical Hit Selection Method

The Graphical Hit Selection method allows you to select existing feature or scan hits in the Graphic Display window, and then use them as inputs for Alignments, Constructions, and Dimensions.

This provides an alternate way to select inputs for those types of commands.

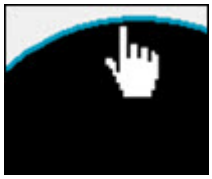
Once you have created the feature or scan hits:

1. Open the relevant dialog box to create your Alignment, Construction, or Dimension.
2. Select the feature or features from the Graphic Display window or from the **Feature List**. When you select the items from the **Feature List**, the software highlights them in the Graphic Display window.



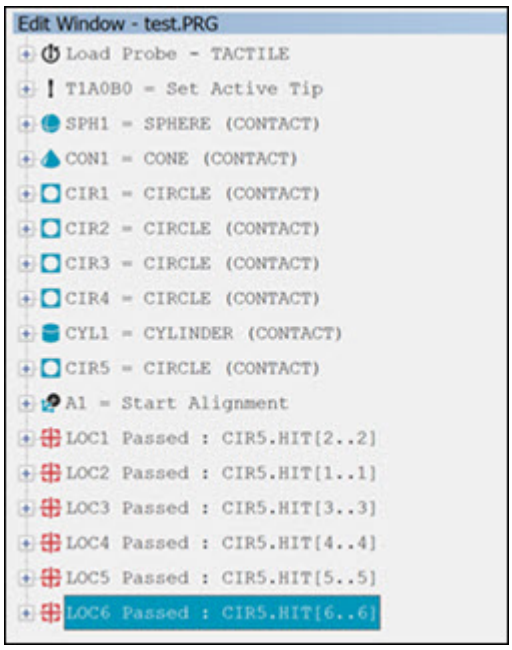
Dimension example using the Graphical Hit Selection method, selecting 6 circles from the Feature List

After you select the feature or scan, the cursor changes to show the hits you can select.



Once you select the item from the Graphic Display window, the software adds it to the **Feature List** of the opened dialog box.

3. Click **Create** on the dialog box to add the command to the Edit window's measurement routine.



Notes on hit properties:

Hits are drawn in black.

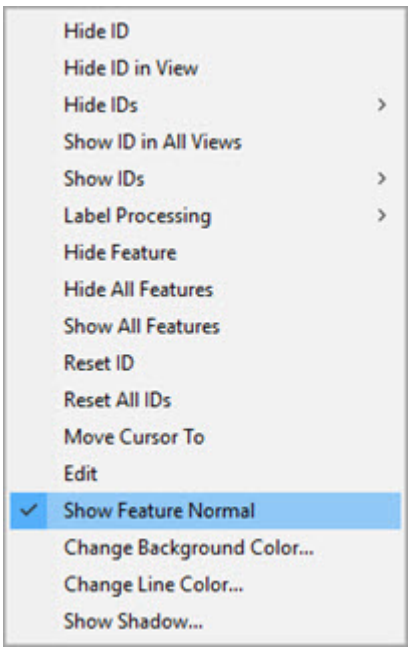
You can define or view the hit selection color from the **Colors** tab on the **CAD & Graphics Setup** dialog box (**Edit | Graphic Display Window | Screen Color**).

You can define or view how hits are drawn in the Graphic Display window from the **Point Symbols** section of the **Symbols** tab on the **CAD & Graphics Setup** dialog box (**Edit | Graphic Display Window | Display Symbols**).

Show Feature Normal Option

To assist in the selection of hits, the **Show Feature Normal** option is available. You can find it on the right-click menu in the Graphic Display window.

To access it, right-click any feature label in the Graphic Display window. The pop-up menu displays a list of options, including the **Show Feature Normal** option.



Graphic Display pop-up menu with the Show Feature Normal option selected

When you select the **Show Feature Normal** option, the software performs these operations:

- Saves the state of all CAD views
- Rotates and translates the feature to normal in the selected view
- Zooms to fit on the selected feature in the selected view
- Flashes the feature in all views

The software displays the CAD model with the transparency value as defined by the **Enable transparency** setting. You can find this setting in the **Transparency** section of the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | Lighting, Materials**).

De-selecting the **Show Feature Normal** option returns all views to the states they were in before the option was last enabled.

Alignment Example

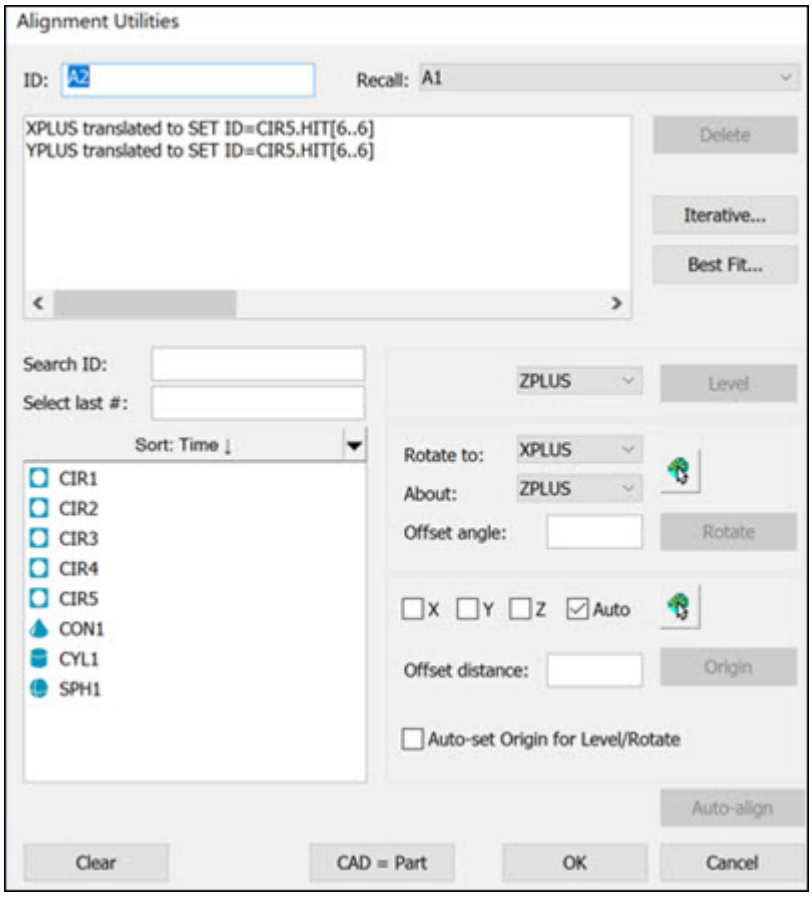
Use the Graphical Hit Selection method to select feature and scan hits as inputs when you create an alignment.

> **!** You cannot use feature or scan hits as inputs for Best Fit and Iterative alignments.

To select feature or scan hits with the Graphical Hit Selection method:

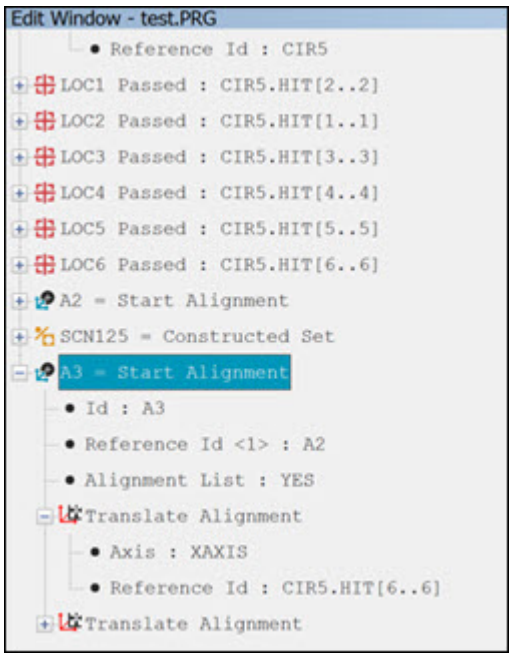
1. Create the feature or scan hits.

2. Click **New Alignment** from the **Alignments** toolbar (**View | Toolbars | Alignment**) or from the menu (**Insert | Alignment | New**). PC-DMIS displays the **Alignment Utilities** dialog box.



3. Select the items to use as inputs from the **Feature List** in the dialog box or select them from the Graphic Display window. For details, see "Graphical Hit Selection Method".

4. After you select all the items, click **Create**. The software displays the new alignment in the measurement routine.



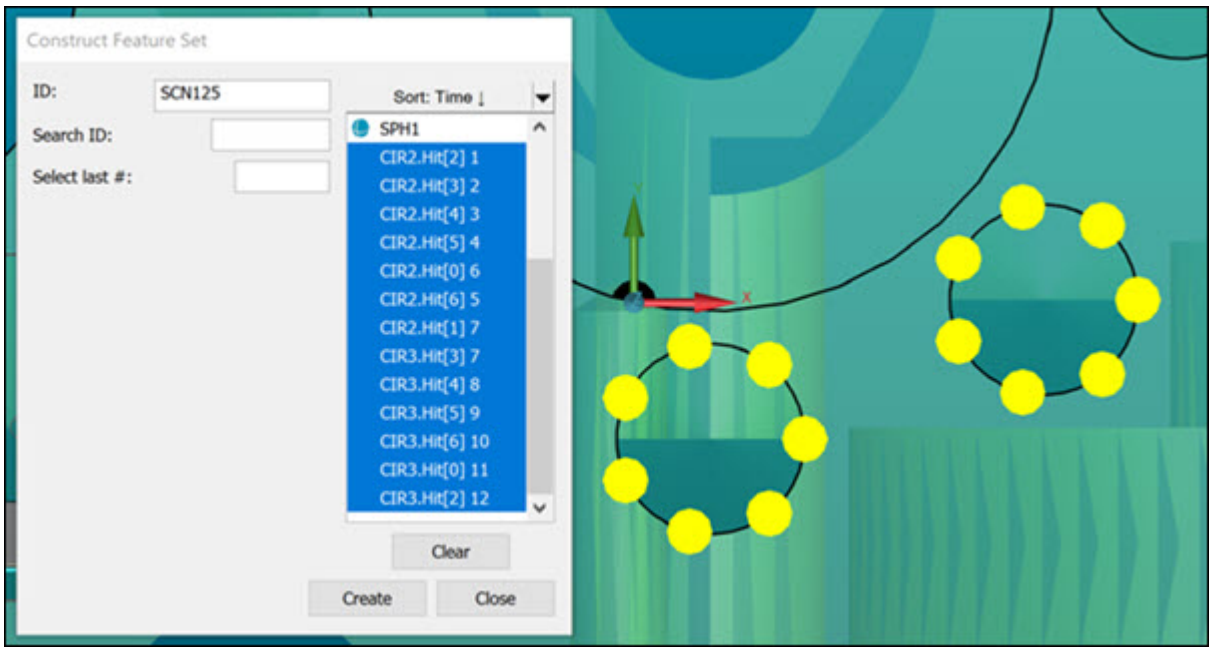
Construction Example

Use the Graphical Hit Selection method to select feature and scan hits as inputs when you create a construction.

Hits can only be selected individually. Multiple hit selection is not currently supported.

To select feature or scan hits with the Graphical Hit Selection method:

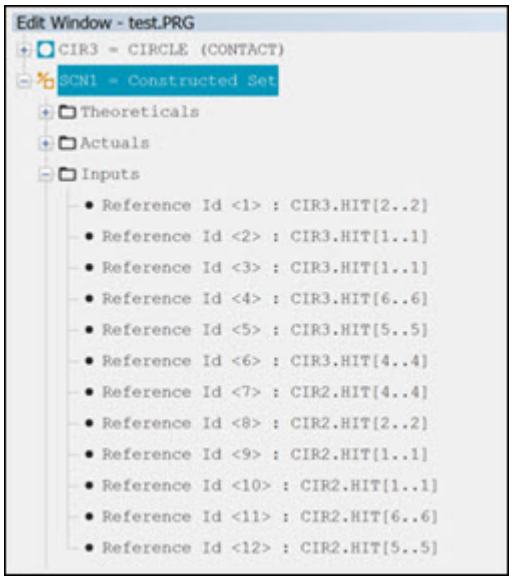
1. Create the feature or scan hits.
2. Click the construction type (for example, the **Constructed Feature Set** option) from the **Constructed Features** toolbar (**View | Toolbars | Constructed Features**) or from the menu (**Insert | Feature | Constructed | Set**). PC-DMIS displays the construction's dialog box.



Construction dialog box with features selected and highlighted in the Graphic Display window.

3. Select the items to use as inputs from the **Feature List** in the dialog box or select them from the Graphic Display window. For details, see the "Graphical Hit Selection Method" topic in the PC-DMIS Core documentation.

4. After you select all the items, click **Create**. The software displays the new construction in the measurement routine.



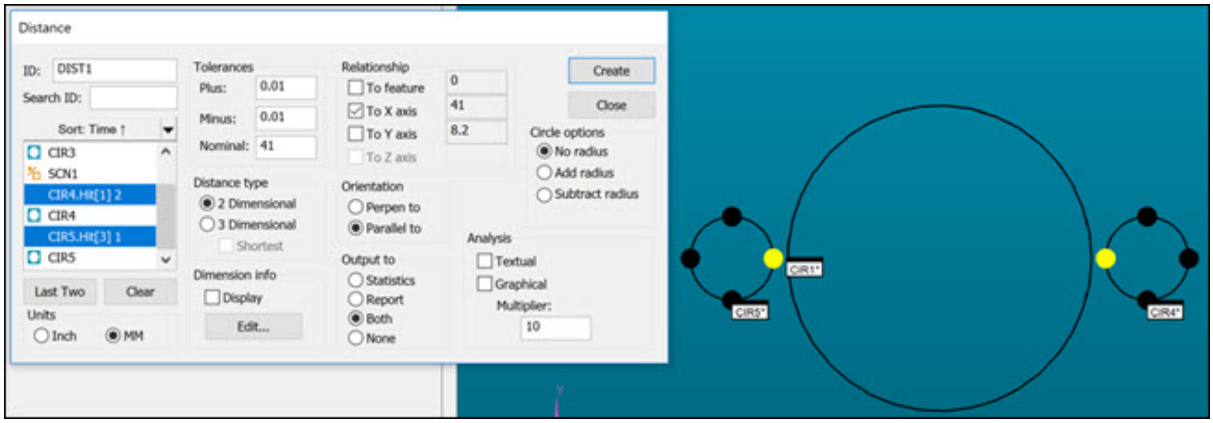
Dimension Example

Use the Graphical Hit Selection method to select feature and scan hits as inputs when you create a dimension.

Dimension types where points are not valid inputs will not display feature or scan hits.

To select feature or scan hits with the Graphical Hit Selection method:

1. Create the feature or scan hits.
2. Click the dimension type (for example, the **Distance** option) from the **Dimension** toolbar (**View | Toolbars | Dimension**) or from the menu (**Insert | Dimension**). PC-DMIS displays the dimension's dialog box.

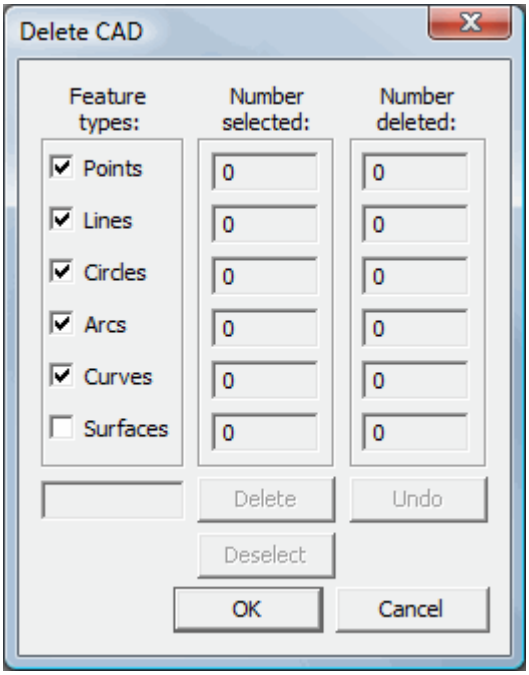


Dimension Distance dialog box with features selected and highlighted in the Graphic Display window

Dimension types where points are not valid inputs will not display Feature or Scan Hits.

3. Select the items to use as inputs from the **Feature List** in the dialog box, or select them from the Graphic Display window. For details, see "Graphical Hit Selection Method".
4. After you select all the items, click **Create**. The software displays the new dimension in the measurement routine.

Deleting CAD Elements



Delete CAD Dialog Box

The **Edit | Delete | CAD Elements** menu option displays the **Delete CAD** dialog box.

You can use this dialog box to permanently remove the CAD elements of selected features from the CAD model in the Graphic Display window.

This can help simplify an imported CAD file before building a measurement routine. For example, a CAD file may contain text or other descriptive information that has no impact on the inspection of the part.

To use this dialog box:

1. From the **Feature types** area, mark the CAD element feature types you want to delete.
2. From the Graphic Display window, press Ctrl + Click (or box-select) to highlight the CAD elements you want to remove.
3. Click **Delete** or press the Delete key to preview the deletion. At this point, the items are only temporarily removed, and you can click **Undo** to restore the temporary deletion.
4. Click **OK** to permanently delete the selected CAD elements.

Dialog Box Description

The dialog box contains these items:

Feature types - Determines what feature types to delete. The available types are:

- Points
- Lines
- Circles
- Arcs
- Curves
- Surfaces

Number selected - Shows the number of selected CAD elements for each type.

Deselect - Deselects any highlighted CAD elements.

Number deleted - Displays the number of features that have been deleted. The numbers in these fields change, depending on the features selected in the **Feature types** area.

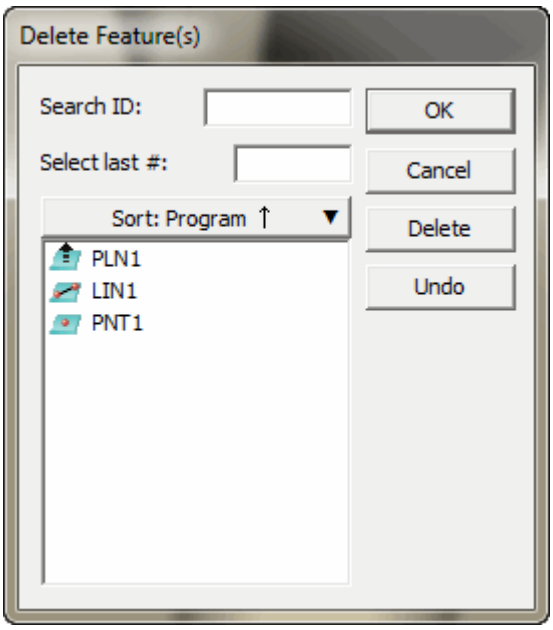
Delete - Deletes selected nominal features. If this dialog box is open, you can also press the Delete key to delete the CAD elements. Until you click **OK**, this is only temporary.

Undo - Restores the recently deleted nominal features. This only works for items that are temporarily deleted. To restore permanently deleted CAD elements, you need to re-import the entire model file.

Deleting Features

PC-DMIS allows you to delete existing features in two ways:

- You can select features in the Edit window and press the Delete key on your keyboard.
- You can use the **Delete Features** dialog box, discussed below.



Delete Features Dialog Box

The **Delete Features** dialog box (**Edit | Delete | Features**) permanently removes measured, auto, or constructed features from the measurement routine. You should use this option when you want to remove a number of unnecessary features. The **Delete Features** dialog box only lists those features above the current cursor position in the Edit window.

Select the features to delete, and then click **Delete** to mark them for deletion. Click **OK** to delete items marked for deletion.

Search ID - Selects matching features after you press Tab.

Select last # - Selects a number of features from the end of the list.

Delete - Marks the selected items for deletion by placing a text fragment of "del" next to selected features in the list. This deletion is not permanent until you click **OK**.

Undo - Restore all the features that are marked for deletion after clicking **Delete**. This button does not restore deleted dimensions once you click **OK**.

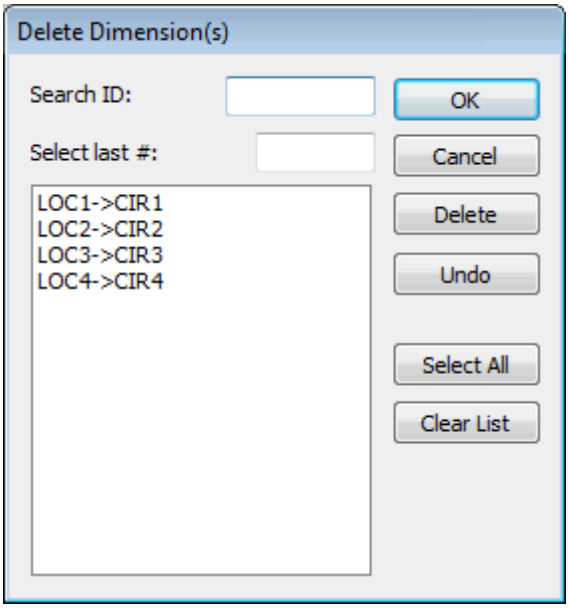
When measured features are removed from the Graphic Display window, PC-DMIS automatically removes any dimensions or datums associated with them from the measurement routine.

Deleting Dimensions

PC-DMIS allows you to delete existing dimensions in two ways:

- You can select dimensions in the Edit window and press the Delete key on your keyboard.

- You can use the **Delete Dimensions** dialog box, discussed below.



Delete Dimensions Dialog Box

The **Delete Dimensions** dialog box (**Edit | Delete | Dimension**) lets you permanently remove any dimensions from the measurement routine. Use this option when you need to remove a number of unnecessary dimensions. The **Delete Dimensions** dialog box only lists those features above the current cursor position in the Edit window.

Select the dimensions to delete, and then click **Delete** to mark them for deletion. Click **OK** to delete items marked for deletion.

Search ID - This box selects matching dimensions after you press Tab.

Select last # - This box selects a number of dimensions from the end of the list.

Delete - This button places a text fragment of "del" next to selected dimensions in the list to mark them for deletion. You can select all the dimensions from the list by clicking **Select All**. This deletion is not permanent until you click **OK**.

Undo - This button restore all the dimensions that are marked for deletion after clicking **Delete**. This button does not restore deleted dimensions once you click **OK**.

Select All - This button selects all the features in the list. For

Clear List - This button clears any selected features from the list.

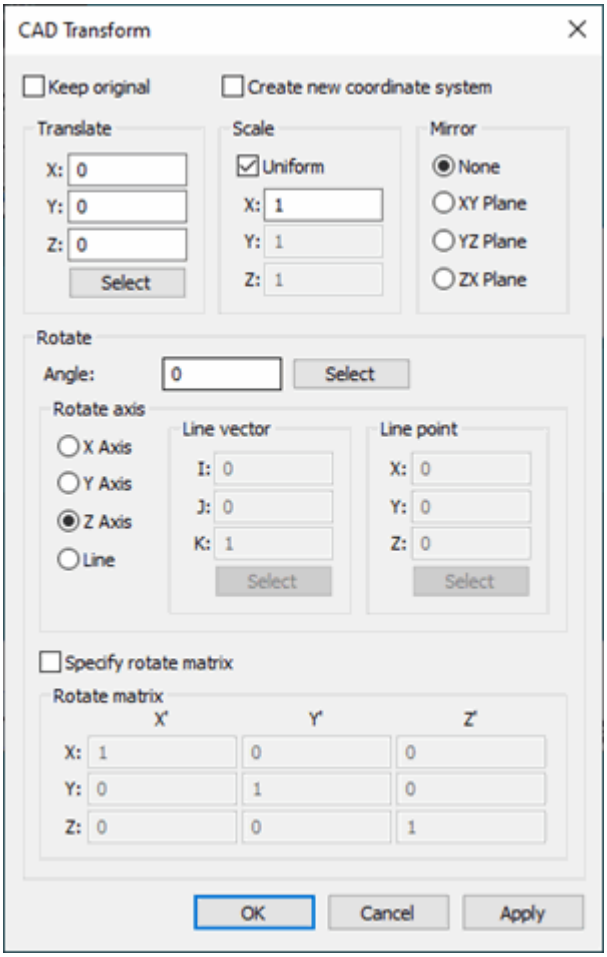
Transforming a CAD Model

You can transform (translate, scale, and rotate) your CAD model. Also, if desired, you can keep a copy of the original, unmodified CAD model as well as create a new coordinate system for the transformed model.

To transform your CAD model, open the **CAD Transform** dialog box (**Operation | Graphic Display Window | Transform**). Use the items in the dialog box as needed, and then click **OK** or **Apply**.

You can also click the **CAD Transform** button from the **CAD Setup** toolbar or from the **CAD** dropdown toolbar on the **QuickMeasure** toolbar to open the **CAD Transform** dialog box.

Performing a CAD transform operation does not transform any existing learned features. This operation essentially renders them useless. You may want to perform a **File | Save As** operation to create a backup of your measurement routine and CAD model before you proceed.



CAD Transform Dialog Box

Keep original - This check box lets you keep a copy of the original, unmodified CAD model.

Create new coordinate system - This check box lets you create a new coordinate system from the newly-translated CAD model. For more information, see "Working with CAD Coordinate Systems".

Translate - This area defines the XYZ offsets to translate the model. You can type the specific location to which you want PC-DMIS to move the coordinate system. If you do not know the coordinates, you can use the **Select** button to select a specific CAD entity to which the coordinate system moves. See "Transforming by Selecting" below.

Scale - This area defines how the CAD model is scaled. This can be useful for fixing models that are not scaled correctly due to improper measurement-unit identification. For example, if the model is sized for millimeter units but you want it scaled for inch units, you can scale the model by 25.4.

The **Uniform** check box uniformly scales the model. If you want to scale a selected axis of the model, clear the **Uniform** check box. Leave the value as 1 for axes that you do not want scaled, and change the axis that you want scaled. You can also scale axes by negative values. This is useful if you want to mirror an axis. In this case, enter -1 for that axis.

Mirror - In this section, you can mirror the CAD view of a part. Mirroring provides the same functionality as using **Scale** with -1 specified for the mirroring axis. Mirroring your CAD model is particularly useful when measuring automotive parts that have identical left- and right-hand pieces. If CAD information is available for the right side of a part, you can mirror the appropriate axis, and create a CAD view of the left side of the part.

The **Mirror** option does *not* create a new measurement routine in the mirrored image. If you want to mirror your measurement routine instead, see the "Mirror" topic in the "Using Basic File Options" chapter, and follow the instructions.

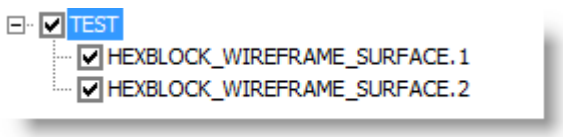
To mirror the CAD data, do the following:

1. Select the axis you want to mirror. If you want to keep your old CAD data so that after mirroring you have CAD data for both the symmetrical halves, select the **Keep original** check box.

2. Click **Apply** or **OK**. PC-DMIS mirrors the CAD drawing in the specified axis and displays the image in the Graphic Display window.

Mirrored Elements in CAD Assemblies

If you open the **CAD Assembly** dialog box, you can see that your original imported image name has the number 1 appended to its name. Any CAD transformation that creates a new instance of the same CAD file (such as a mirror operation) is then incremented to have a unique number. See "Working with Assemblies of Parts".



Example showing a mirrored CAD file names with unique numbers inside the CAD Assembly tree view.

Hidden Assembly Components During a Mirror Operation

If you hide elements of your CAD assembly prior to mirroring your CAD data, hidden components are still mirrored, but their mirrored counterparts also continue to remain hidden in the Graphic Display window until their visibility state is modified in the **Assembly Tree View** of the **CAD Assembly** dialog box.

Rotate - This area controls how the CAD model is rotated. Type the angle you want to rotate the model by in the **Angle** box. You can type the specific location to which you want PC-DMIS to move the coordinate system. If you do not know the coordinates, you can use the **Select** button to select a specific CAD entity to which the coordinate system moves. See "Transforming by Selecting" below.

Rotate axis - This area defines the line about which the CAD model is rotated. The model rotates about this line by the specified angle. The direction of rotation follows the right-hand rule. You can use one of the coordinate-system axes as the line to rotate about by selecting the appropriate **X**, **Y**, or **Z** axis option button. If you do not want to rotate about one of the coordinate system axes, you can rotate about an arbitrary line by selecting the **Line** option button. This enables the **Line vector** and **Line point** areas. Fill these areas out to determine the point and vector that comprise the arbitrary line.

Right-Hand Rule: If you extend the thumb on your right hand in the direction of the line vector, and curl your other fingers into your palm, your fingers indicate the direction of positive angle rotation.

Rotate matrix - As you determine your CAD model's new transformation, this area automatically gets filled out with the values to use in a 3 X 3 matrix. This 3 X 3 matrix rotates the CAD model. Usually, you won't need to fill out anything in this area as it is generally for informational purposes only.

For the Advanced User:

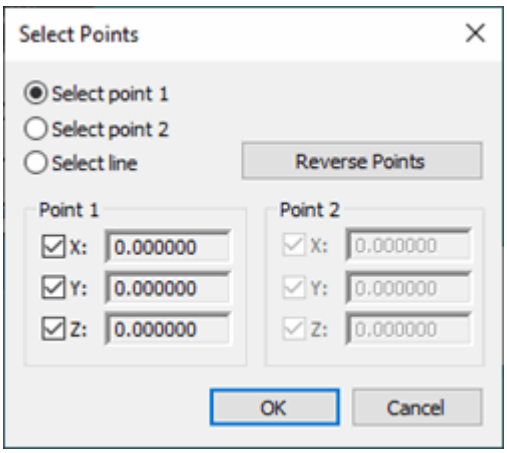
You can select the **Specify rotate matrix** check box to type the values for the rotation matrix manually. The columns specify the axes for the rotation. These restrictions apply:

- Each axis of the matrix must be orthogonal to the other two axes. Thus, each axis pair must form a 90-degree angle.
- Each axis must be unit length. That is, the length of the axis must be one.

When you apply the transformation, if either of these restrictions is not met, a message appears and indicates the problem, and PC-DMIS automatically corrects the rotation matrix.

Transforming by Selecting

When you click the **Select** button, PC-DMIS displays the **Select Points** dialog box.



Select Points Dialog Box

Instead of typing an offset value, you can use this dialog box to select an offset by picking a CAD entity from the Graphic Display window.

Select point 1 - This option defines the translation location. With this option selected, click on a desired CAD entity. This anchors the point to that location.

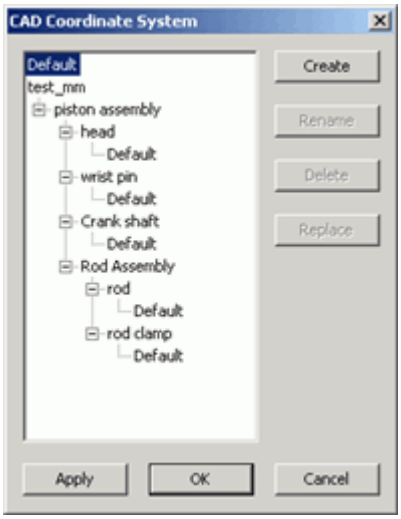
Select point 2 - This option specifies the angle with respect to point 1 and the axis of rotation. With this option selected, click on a second CAD entity on your CAD model to define the angle.

Select line - Instead of selecting two points for your translation, this option lets you select a single line. PC-DMIS then sets the values of the **Point 1** and **Point 2** areas to match the start and end points of the selected line.

Reverse Points - This button switches the XYZ values of Point 1 with the XYZ values of Point 2.

Point 1 and Point 2 - These areas define the XYZ center point of the CAD entity selected with the **Select point 1** and **Select point 2** options. Use the check boxes in these areas to selectively update the X, Y, or Z value of the point, which enables you to specify points where there is no actual geometry for you to click on. For example, for Point 1, suppose you wanted the X and Y value of one point but the Z value of a different point. To do this, clear the **Z** check box and then select one point. Then clear the **X**, **Y** check boxes, select the **Z** check box, and select the other point.

Working with CAD Coordinate Systems



CAD Coordinate System Dialog Box

The **Edit | Graphic Display Window | CAD Coordinate System** menu option displays the **CAD Coordinate System** dialog box. The **CAD Coordinate System** dialog box lets you create or select new coordinate systems for your CAD model.

The **Coordinate System** list on the left side of the dialog box shows all the coordinate systems in the CAD model. The coordinate systems listed first are the global coordinate systems. The **Default** coordinate system is always listed.

If the CAD model is an assembly of component parts then following the global coordinate systems, PC-DMIS lists all the coordinate systems contained in that assembly. These are listed in a tree view structure. To expand the tree view, click on a plus symbol (+). PC-DMIS displays the component parts' assembly coordinate systems. The **Default** coordinate system is always listed for each part component in the assembly. This coordinate system defines the part's default local coordinate system.

When you first open the **CAD Coordinate System** dialog box, PC-DMIS checks to see if the current active coordinate system matches a coordinate system in the list:

- If it finds a match then PC-DMIS selects it from the list.
- If it doesn't match any of the coordinate systems then PC-DMIS uses the **Default** coordinate system at the top of the list. This can happen if you transformed the CAD without creating a coordinate system (for more information, see the "Transforming a CAD Model" topic).

To Select and Use Coordinate Systems

To use an existing coordinate system:

1. Select the coordinate system from the **Coordinate System** list. PC-DMIS shows that coordinate system in the Graphic Display window. If you select an assembly component, PC-DMIS selects the first coordinate system in that component instead.

2. Click **Apply** or **OK**. The selected coordinate system becomes the new active coordinate system, and PC-DMIS redraws the CAD to reflect its new position.

To Create a Coordinate System

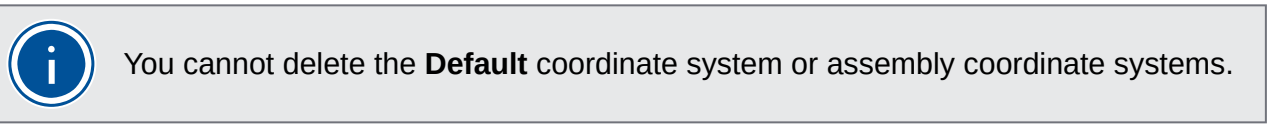
To create a coordinate system at the current CAD model's position, click the **Create** button. You can use the **CAD Transform** dialog box to change the CAD model's position. For more information on how to transform a CAD model, see the "Transforming a CAD Model" topic.

To Rename a Coordinate System

To rename a coordinate system, select the coordinate system from the list, and click the **Rename** button. Type the new name.

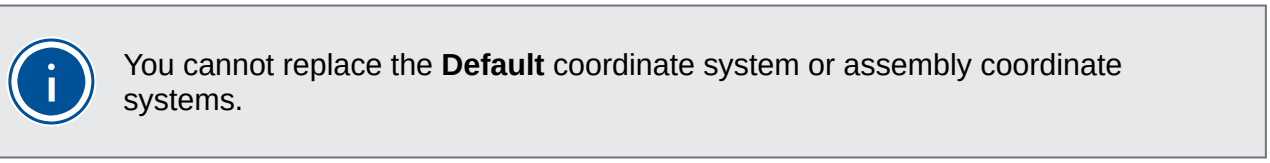
To Delete a Coordinate System

To delete a coordinate system, select the coordinate system from the list, and click the **Delete** button. PC-DMIS deletes the selected coordinate system.

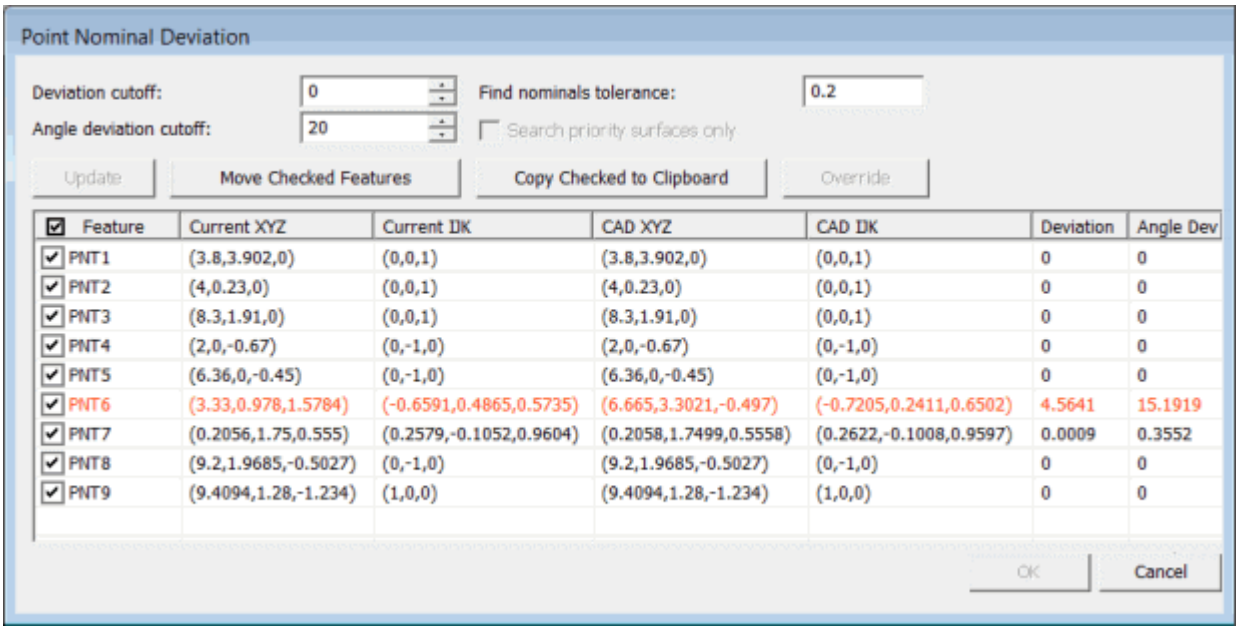


To Replace a Coordinate System

To replace or overwrite an existing coordinate system with the current CAD model's position, select a coordinate system from the list, and click the **Replace** button.



Checking and Fixing Point Nominal Deviation



Point Nominal Deviation Dialog Box

The **Edit | Graphic Display Window | Point Nominal Deviation** menu option displays the **Point Nominal Deviation** dialog box. Use this re-sizable dialog box to compare certain point measurements against a CAD model to check for changes or updates made to the CAD model. Essentially, you define a tolerance value, and PC-DMIS looks at the nominals for any point features in your measurement routine and compare them against the CAD model.

PC-DMIS lists all of the points. It lists any points that deviate beyond the defined tolerance value in a red/orange color; you can then adjust them as needed.

The **Point Nominal Deviation** dialog box contains the following options:

Deviation cutoff - This box filters the point features that display. The distance between a point feature's theoretical position and the CAD model nominal position is its deviation. Only points whose deviation is greater than or equal to the deviation cutoff are listed.

Angle deviation cutoff - This box controls the nominal CAD position. The CAD surface normal and the point vector must be within this angle. The range for this angle is 0–90 degrees.

Find nominals tolerance - This box defines the tolerance value that are allowed for each point feature. Points with a deviation that exceeds this tolerance value turn red.

Search priority surfaces - This check box specifies whether or not PC-DMIS only uses priority surfaces when searching for a solution. You can define priority surfaces using the **Set as priority** check box in the **Edit CAD Elements** dialog box. See the "Editing CAD" topic for more information on how to do this. If you clear this check box, and PC-DMIS

cannot find an acceptable value within the defined priority surfaces, it checks all of the other surfaces on the CAD model.

Update - This button lists all of the points on the CAD model.

Move Checked Features - This button updates the XYZ and IJK values of those point features whose check boxes you have selected to match the CAD model's XYZ and IJK values.

Copy Checked to Clipboard - This button copies the information about all of the selected points to the Clipboard. For an example, see "Clipboard Example".

Override - Sometimes the best point that the algorithm finds is incorrect. When you select a single feature, the **Override** button becomes available. Click this button to display the **Override** dialog box, which you can use to override the found point with a different point. For more information, see "Using the Override Dialog Box".

Feature - This column displays the name of the ID of the feature. The column's header contains a check box. Selecting or clearing this check box selects or clears the check boxes for all point features in the list.

Current XYZ - This column displays the feature's current theoretical position.

Current IJK - This column displays the feature's current theoretical vectors.

CAD XYZ - This column displays the feature's nominal position if it were moved onto the CAD.

CAD IJK - This column displays the feature's nominal vector if it were moved onto the CAD.

Deviation - This column displays the distance between the feature's current theoretical position and its corresponding nominal CAD position.

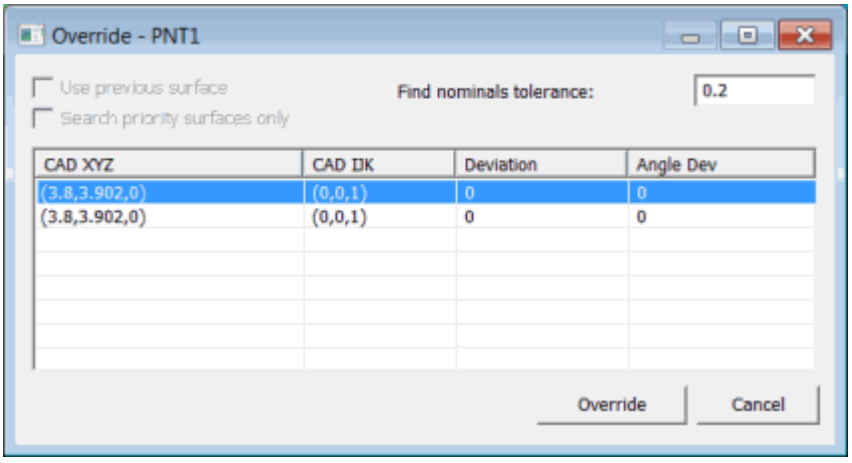
Angle Dev - This column displays the angle deviation between the feature's current theoretical vector and its corresponding nominal CAD vector.

Clipboard Example

```
Feature     Current XYZ          Current
IJK              CAD XYZ                CAD
IJK                  Deviation   Angle Dev
PNT1       (3.8,3.902,0)         (0,0,1)                  (3
.8,3.902,0)           (0,0,1)                  0           0

PNT2       (4,0.23,0)            (0,0,1)                  (4
,0.23,0)              (0,0,1)                  0           0
PNT3       (8.3,1.91,0)          (0,0,1)                  (8
.3,1.91,0)            (0,0,1)                  0           0
PNT4       (2,0,-0.67)           (0,-1,0)                 (2
,0,-0.67)             (0,-1,0)                 0           0
PNT5       (6.36,0,-0.45)        (0,-1,0)                 (6
.36,0,-0.45)          (0,-1,0)                 0           0

PNT6       (3.33,0.978,1.5784)   (-0.6591,0.4865,0.5735)
 (6.665,3.3021,-0.497)   (-0.7205,0.2411,0.6502)
 4.5641      15.1919
PNT7       (0.2056,1.75,0.555)   (0.2579,-0.1052,0.9604)
 (0.2058,1.7499,0.5558)  (0.2622,-0.1008,0.9597)
 0.0009      0.3552
PNT8       (9.2,1.9685,-0.5027)  (0,-1,0)                 (9
.2,1.9685,-0.5027)    (0,-1,0)                 0           0
PNT9       (9.4094,1.28,-1.234)
 (1,0,0)                  (9.4094,1.28,-1.234)    (1,0,0)
              0           0
```

Example showing the points copied to the Clipboard (best viewed in full screen).

Using the Override Dialog Box



Override Dialog Box

The **Override** dialog box appears when you select the **Override** button in the **Point Nominal Deviation** dialog box. It shows a list of all of the CAD nominal points that correspond to the selected feature from all surfaces within the Find Nominals Tolerance search zone. PC-DMIS initially sorts these points from the smallest to the largest deviation.

Usually, the first point that is listed is the point used in the **Point Nominal Deviation** dialog box. Each point is graphically represented with a cross-hair in the Graphic Display window.

Selecting a point in the list highlights that point as well as any curve or surface on which the point lies in the Graphic Display window. Once you find the desired point, click the **Override** button. The **Override** dialog box closes and the **Point Nominal Deviation** dialog box reappears, showing the updated point value.

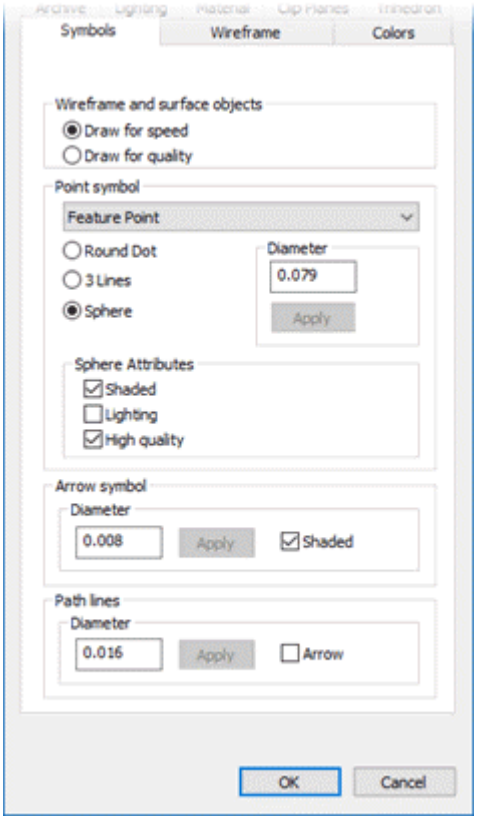
Use previous surface - You should use this check box when you override several points. For example, if you know that points for several features should all lie on the same surface but the algorithm incorrectly put the points on different surfaces, you would override the first feature as usual. Then on the second and subsequent overrides, you could select the **Use previous surface** check box. This would cause PC-DMIS to only show the points found on the surface of the previous override.

Find nominals tolerance - This box functions the same as the box with the same name in the **Point Nominal Deviation** dialog box, but with different results. PC-DMIS searches the CAD for nominal points that correspond to the selected feature. It searches in a spherical zone around the current theoretical position of the selected feature. This box defines the size of that spherical search zone. The value regulates the amount of CAD that PC-DMIS evaluates while generating this point override list.

The **Search priority surfaces only** check box is discussed in the table above.

Modifying Display Symbols

You can select **Edit | Graphic Display Window | Display Symbols** (or the **Display Symbols** icon from the **Graphic Modes** toolbar) to display the **Symbols** tab on the **CAD and Graphic Setup** dialog box.



CAD and Graphic Setup dialog box - Symbols tab

This tab lets you change how PC-DMIS displays different symbols in the Graphic Display window. The initial values for the settings on the **Symbols** tab come from the measurement routine file (.prg). If no settings information exists in the measurement routine, then the initial values for the settings come from the JSON file or from hard-coded defaults.

Available symbols you can modify include **Wireframe and Surface objects**, the **Point Symbol**, the **Arrow Symbol**, and **Path Lines**. Corresponding areas exist on the **Symbols** tab. Whenever you make a change to one of the check boxes or option buttons, PC-DMIS applies that change automatically so you can see what effect it has. PC-DMIS only saves your changes once you click the **OK** button. To see changes to the symbols' sizes, click the appropriate **Apply** button.

Draw for speed - This option turns off anti-aliasing for wireframe and surface objects. This option optimizes the drawing speed of CAD objects in the Graphic Display window.

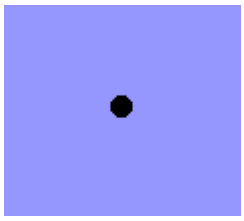
Draw for quality - This option turns on anti-aliasing for wireframe and surface objects. This option optimizes the drawing quality of CAD objects in the Graphic Display window.

Point Symbol - This list defines the type of Point to modify. You can choose **Scan Point**, **CAD Point**, or **Feature Point**. The default item is **Feature Point**.

Diameter / Width in Pixels - This box defines the size of the point symbol. The **Round Dot** symbol uses the **Width in Pixels** value. **3 Lines** and **Sphere** use the **Diameter** value. This diameter value is in measurement routine units. Note that the maximum size for **Round Dot** is based on the computer's physical video card. If the size exceeds the limits

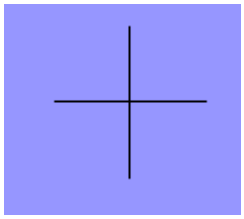
of the current computer's video card, PC-DMIS displays the symbol at the largest size available to the current system's video card.

Round Dot - This option displays the point symbol as a round dot.



Example Round Dot symbol

Three Lines - This option displays the point symbol as a three-lined cross hair.



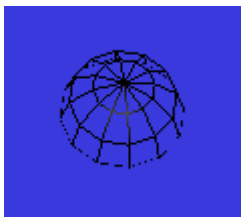
Example Three Lines symbol

Sphere - This option displays the point symbols as a sphere. This is the slowest symbol for PC-DMIS to draw, especially if you have all the attributes selected. These check boxes become available if you select the Sphere symbol. They provide you with additional attributes to further control the display of a sphere symbol in the Graphic Display window:

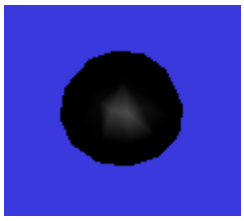
- **Shaded** - This check box produces a shaded (opaque) sphere symbol.
- **Lighting** - This check box adds OpenGL lighting to the sphere symbol.
- **High Quality** - This check box produces a smoother-looking sphere symbol.

While these check boxes improve the image quality of the sphere symbol, they also cause a slight increase in the time it takes to draw the sphere symbols whenever the screen gets refreshed.

Shaded, High Quality



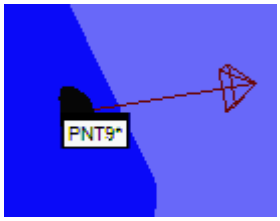
Lighting, High Quality



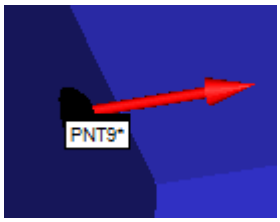
Shaded, Lighting, High quality

Arrow Symbol - This box controls the diameter size (in measurement routine units) of the arrows displayed in the Graphic Display window. The size of the actual arrow in the display only changes if you select the **Shaded** check box.

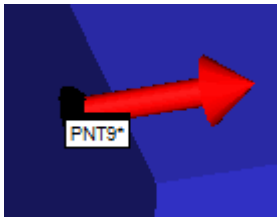
Shaded - This check box shades arrow symbols so that it draws arrows as cylinder tubes with a set diameter. If not selected, the software draws arrows as simple lines without size.



Not Shaded

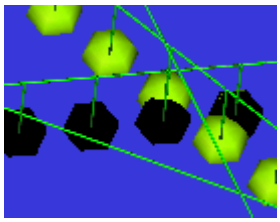


Shaded

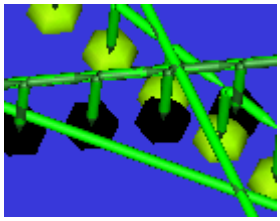


Increased Diameter

Path Lines - The **Diameter** value in this area controls the fixed diameter size (in measurement routine units) of the path lines. Fixed size means the path lines do not change size on the screen when you zoom in or out on part model.

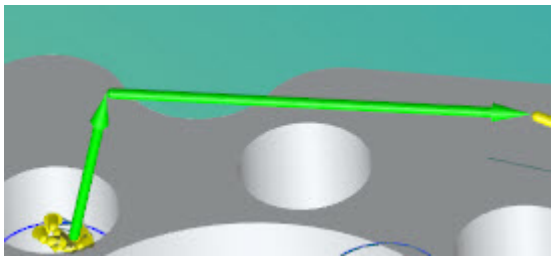


Default Size

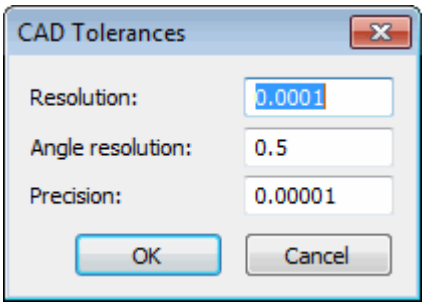


Increased Diameter

The **Arrows** check mark shows an arrow on the path lines:



Changing CAD Tolerances



CAD Tolerances Dialog Box

The **Edit | Graphic Display Window | CAD Tolerances** menu item displays the **CAD Tolerances** dialog box. This dialog box allows you to view and modify the following CAD tolerances that are stored in the .cad file:

- **Resolution** tolerance - Determines the distances at which two XYZ points are considered coincident.

- **Angle Resolution** tolerance - Determines the angle (in degrees) at which two angles are considered equal. It also determines if two vectors are considered equal by checking the angle between the two vectors.
- **Precision** tolerance - Specifies the accuracy of iterated solutions on the geometry.

The **Resolution**, **Angle Resolution** and **Precision** boxes allow you to view and modify these tolerances.

CAD Resolution Tolerance Information

The resolution tolerance is the distance at which two XYZ points are considered to be coincident.

This tolerance determines whether or not a freeform geometry defines an analytic shape. For example, a freeform curve, depending on the tolerance, may or may not be defined as a circle. If the curve defines a circle within the resolution tolerance, PC-DMIS identifies it as a circle, and any functions that depend on this information work properly. For example, the Single-Click Auto feature functionality depends on this information.

In addition, this tolerance determines whether or not two adjacent geometries are connected. For example, if you have PC-DMIS perform a Section Cut scan operation on the CAD model, PC-DMIS compares the gap between adjacent surfaces against the resolution tolerance. If the gap is within this tolerance, PC-DMIS considers the surfaces to be connected and there is not a break in the section cut.

The default resolution tolerance is 0.01 mm. The minimum resolution tolerance is 0.0001 mm and the maximum is 2.0 mm. When you import a new CAD model, PC-DMIS sets the resolution tolerance to the default value, except in cases where the imported CAD file itself specifies a resolution tolerance. For example, if an imported IGES file specifies a resolution tolerance in the header, PC-DMIS uses that value instead of the default value.

CAD Angle Resolution Tolerance Information

The CAD angle resolution tolerance determines the angle (in degrees) at which two angles are considered equal. It also determines if two vectors are considered equal by checking the angle between the two vectors.

This tolerance is mainly used to check if geometry forms a feature when angles are involved. For example, a round slot has two circular arcs at its ends. The angle between the two circle normals must be within the angle resolution for the feature to be recognized as a round slot.

Another example is a square slot where adjacent sides must form 90 degree angles. If this angle is off more than the angle resolution, then the feature is not recognized as a square slot.

The default angle resolution tolerance is 0.5 degrees. The minimum angle resolution tolerance is 0.01 degrees and the maximum is 5.0 degrees. When you import a new CAD model, PC-DMIS sets the angle resolution tolerance to the default value.

CAD Precision Tolerance Information

The precision tolerance specifies the accuracy of iterated solutions on the geometry. For example, during some find nominals operations, an iteration process is used to compute the solution on the CAD geometry. Once the solution is within the precision tolerance of the actual solution, the iteration process stops.

The smaller the precision tolerance, the slower the algorithms that use this tolerance for computing a solution.

The default precision tolerance is 0.00001 mm and this is the recommended tolerance to use. The minimum precision tolerance is 0.0000000001 mm and the maximum is 0.001 mm. When you import a new CAD model, the precision tolerance is set to the default value.

Centering the Probe on the Screen

By default, the part remains stationary on the Graphic Display window and the animated probe moves around the part as its physical counterpart moves in 3D space; to see a different section of a larger part on the screen, you have to translate (or move) the part image inside the Graphic Display window.

The **Operation | Graphic Display window | Center Probe** menu item changes this default panning functionality so that as the probe physically moves, the part on the screen moves instead thereby keeping the animated probe image always centered on the screen.

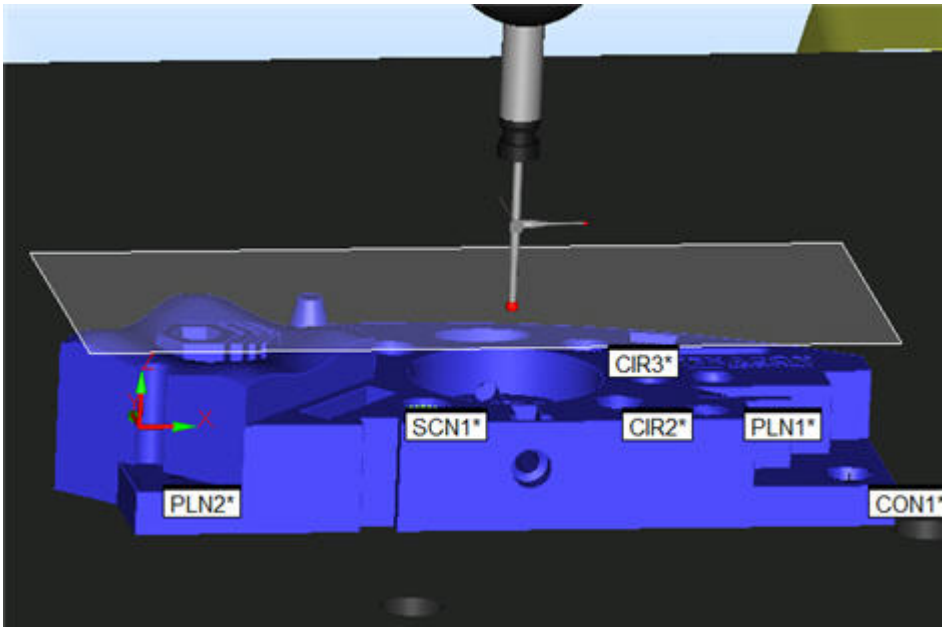
You can also access this from the **Graphic Modes** toolbar, and clicking the **Center Probe** icon .

You may find this centering functionality helpful if you are using portable devices to measure a large part. It allows you to walk around a large part, and even though you aren't near the computer, the Graphic Display window automatically updates so that the probe remains visible in the window along with the section of the CAD model you are inspecting.

Viewing Clearance Planes

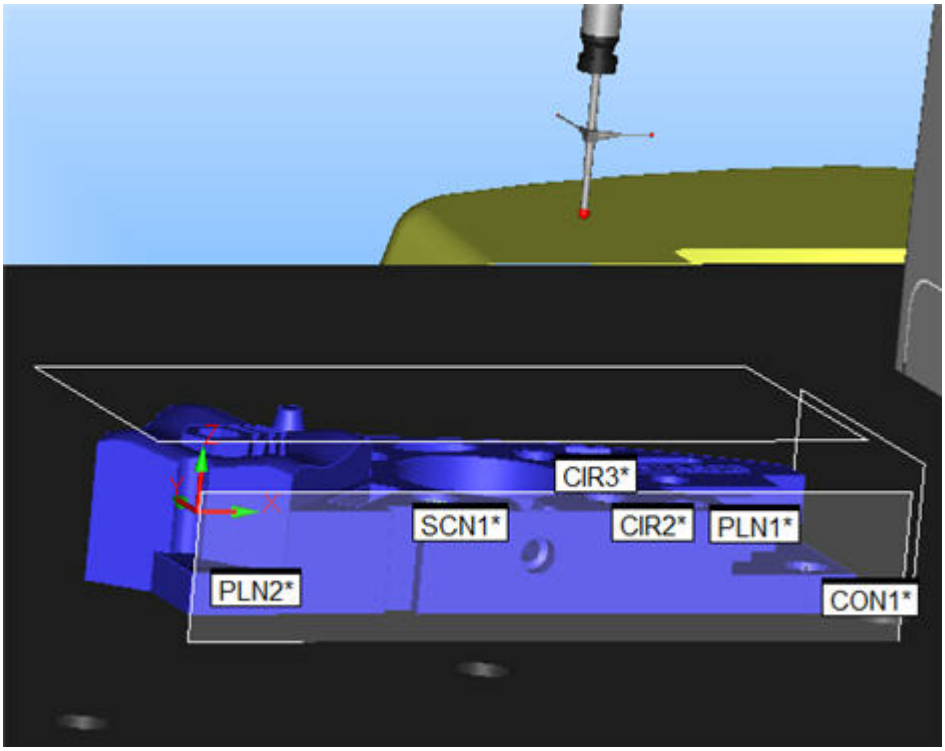
You can display the active clearance plane as a translucent image in the Graphic Display window. To do this, from the Graphic Items toolbar, select the **Toggle Clearance Plane Items** icon .

By default, the active clearance planes are displayed as white translucent planes; while all non-active clearance planes are displayed as white polylines.



A sample clearance plane displayed as a translucent image.

PC-DMIS displays up to one clearance plane along each axis, for a maximum of three clearance planes at a time (one along Z, one along Y, and one along X). If you have more than one clearance plane defined along the same axis and it isn't the currently displayed clearance plane, it displays the most recently displayed clearance plane used on that axis.



Example showing active and non-active clearance planes.

You can further control the display of both the active and non-active clearance planes by modifying these entries in the **OpenGLSettingsClearancePlanes** section in the PC-DMIS Settings Editor:

- `ActiveSymbol` - Displays the active clearance plane as either a polyline or transparent plane (0=Polyline; 1=Transparent Plane; the default is 1).
- `ActiveColor` - Determines the color of the active clearance plane.
- `ActiveLineWidth` - Determines the line width of the active clearance plane when drawn as a polyline.
- `ActivePercentTransparency` - Determines the amount of transparency for the active clearance plane.
- `Symbol` - Displays the non-active clearance planes as either polylines or transparent planes (0=Polyline; 1=Transparent Plane; the default is 1).
- `Color` - Determines the color of the non-active clearance planes.
- `LineWidth` - Determines the line width of the non-active clearance planes when drawn as a polyline.
- `PercentTransparency` - Determines the amount of transparency for the non-active clearance planes.

You can find details on these settings in your PC-DMIS Settings Editor documentation. For information on how to use the Settings Editor, see "Modifying Setting Entries".

Editing a Measurement Routine

Editing a Measurement Routine: Introduction

The main purposes of PC-DMIS are to allow you to create, edit, and execute measurement routines with ease. This chapter discusses using the **Edit** menu (with other menu options) to edit your measurement routines. While the **Edit** menu works with the commands that appear in the Edit window, a discussion of the Edit window itself is beyond the scope of this chapter. For an in-depth look at the Edit window, see the "Using the Edit Window" chapter.

This chapter covers these topics:

- Using Standard Edit Commands
- Editing Patterns of Features
- Finding and Replacing Text
- Editing Data Fields
- Jumping to Specified Line Numbers
- Jumping to and from Referenced Commands
- Overriding Guessed Features
- Overriding Found Nominals
- Marking Commands for Execution
- Creating and Executing Marked Sets
- Using Bookmarks
- Using Breakpoints
- Changing Fonts and Colors
- Editing External Objects
- Protecting Measurement Routines from Edits
- Updating Nominals from CAD

- Resetting Measured Values to Nominals

Using Standard Edit Commands

PC-DMIS, like most Windows applications, allows you to use standard edit commands in your measurement routine. The following standard menu options are available:

- Undo
- Redo
- Cut
- Copy
- Paste
- Delete
- Select All

Undo

The **Edit | Undo** menu option removes the last change you made to the Edit window. You can continue to select the **Undo** option to remove multiple changes to the Edit window.

Be aware that switching to a different Edit window mode, completely removes all changes from memory.

Redo

The **Edit | Redo** menu option reverses the last changes that took place when you used the **Undo** menu option. Like the **Undo** option, you can use the **Redo** option multiple times to replace multiple changes.

Be aware that switching to a different Edit window mode, completely removes all changes from memory.

Cut

The **Edit | Cut** menu option allows you to "cut" or move commands (or text) from one area to another using the clipboard.

To move commands using **Cut**:

1. Highlight the commands you want to move.
2. Select the **Cut** option. The commands are removed from the Edit window and stored on the Clipboard.
3. Select the **Paste** command to restore the cut commands. (For information, see "Paste".)

When you select the **Cut** command, the Clipboard contents are replaced with the cut selection.

Copy

The **Edit | Copy** menu option also allows you to copy and move commands (or text) from one area to another using the clipboard. The only difference between this option and the **Cut** option is that the commands are not removed from the Edit window when it is copied.

To copy and paste commands:

1. Highlight the desired commands.
2. Select the **Copy** command. The text remains in its current location but is stored in the Clipboard as well.
3. Select the **Paste** command to restore the cut text. (See "Paste".)

When you select the **Copy** command, the Clipboard contents are replaced with the copied selection.

Paste

The **Edit | Paste** menu option pastes the copied commands (or text) from the Clipboard into the Edit window at the current insertion point (or cursor location).

When you paste a command, PC-DMIS keeps the original command's ID, so two commands then have the same ID. Give the pasted command a new ID that the measurement routine does not already use. If you type an ID that another command already uses, PC-DMIS does not accept it.

Delete

The **Edit | Delete | Selection** menu option deletes the currently highlighted selection from the Edit window. This does not place the contents of the deleted selection in the Clipboard.

Select All

The **Edit | Select All** menu option allows you to select all the contents of the Edit window for editing purposes.

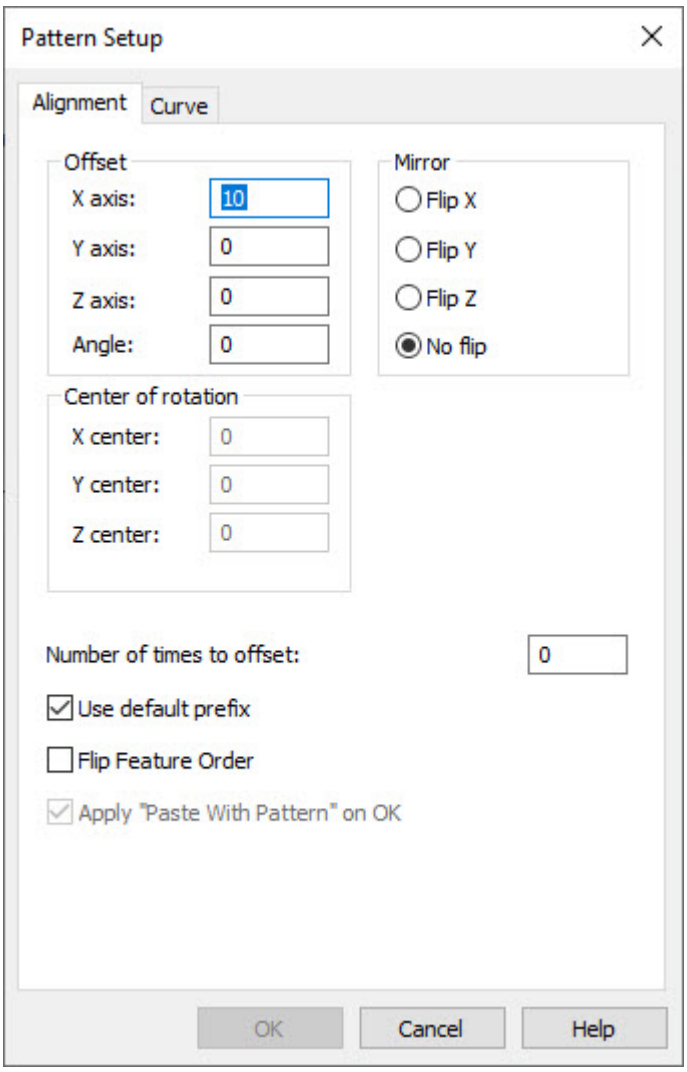
Defining and Pasting Patterns of Features

Patterns of features often occur in parts. With PC-DMIS, you can create patterned features.

You can use **Edit | Pattern** to open the **Pattern Setup** dialog box. This dialog box contains two tabs, the **Alignment** tab and the **Curve** tab. These tabs define the offsets and other options to use when you select the **Edit | Paste With Pattern** menu item.

Pattern Setup - Alignment Tab

The **Alignment** tab on the **Pattern Setup** dialog box provides the basic options of pattern setup for non-curve patterns. It can also establish a temporary alignment for an angle offset pattern.



Offset

The **X axis**, **Y axis**, and **Z axis** boxes allow you to set the X, Y, and Z Offsets between pattern occurrences. PC-DMIS offsets the nominal X, Y, and Z data of the pasted feature (or set of features) by these offset values.

The **Angle** box allows you to set the angular offset between pattern occurrences.

An angular offset may be useful when you measure the position location of several holes in a bolt hole pattern where the datum is a hole in the center of the pattern. PC-DMIS rotates the offset around the origin defined with the **Center of Rotation** area.

You can use angular offsets with X, Y, and Z offsets and the flips in the **Mirror** area. PC-DMIS applies these offsets in this order: flips, rotations, translations.

Mirror

This allows you to mirror an axis. The **Flip X**, **Flip Y**, or **Flip Z** option flips the selected X, Y, or Z axis for the pasted pattern. **No Flip** means the axis is not flipped.

Center of Rotation

The **Center of Rotation** area contains boxes for you to define a rotation center for an angle offset pattern (for example, a bolt hole pattern around a central hole). Prior to version 2021.1, if you wanted to do an angle pattern around a central hole, you had to create an alignment at that feature. Now, you don't have to create that alignment. Instead, you can click on the central circle feature, and PC-DMIS extracts the XYZ center location from the CAD and puts those values into **X center**, **Y center**, and **Z center** boxes. PC-DMIS applies the rotation around the center of rotation and within the currently active workplane.

This box becomes available to edit, if you type a value in the **Angle** box under **Offset**.

Number of times to offset

This box lets you set the number of times that PC-DMIS offsets a pattern. The default value is 1.

Use default prefix

The **Use default prefix** check box specifies whether PC-DMIS uses a default ID prefix when you use the **Paste With Pattern** menu item to paste features.

- If you select this check box, the **Paste With Pattern** menu item uses the default ID prefix when you paste new features.
- If you clear this check box, the **Paste With Pattern** menu item uses each feature ID as the base feature name and adds an incremental and numerical suffix.

If you don't use a default prefix when you paste many lines, the processing time is longer because each feature ID represents a new pattern.

Flip Feature Order

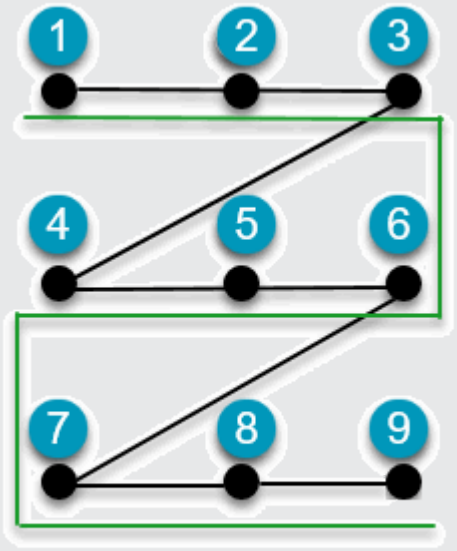
The **Flip Feature Order** check box reverses the order of the pasted features.

This check box works only with Auto features and measured features. Also, this works with only these move commands:

`MOVE/POINT,MOVE/INCREMENT,MOVE/CLEARPLANE,MOVE/CIRCULAR,MOVE/ALL,and MOVE/CLEARPOINT`

Suppose you have three Auto features: 1, 2, and 3. If you don’t select this check box, PC-DMIS pastes the new features as 4, 5, 6.

If you select this check box and change the **Number of times to offset** value from 1 to 2, you have nine features in this order: 1, 2, 3, 6, 5, 4, 7, 8, 9. Each new set of features (6, 5, 4 and 7, 8, and 9) reverses its order from the previous set. PC-DMIS follows that order during execution. In the diagram below, you can see this with the green lines. If you don't select this check box, the black lines show the original execution order:



Apply "Paste With Pattern" on OK

You can manually select the **Paste With Pattern** menu option if you like, or you can use this check box. This check box applies the **Paste With Pattern** operation behind the scenes automatically when you click **OK**.

You can use Paste with Pattern to pattern a TIP command for star probe configurations. PC-DMIS searches for a TIP on the star probe with a vector that matches the vector of the patterned feature. It handles the TIP as follows:

- If a matching TIP is available within the tolerance defined by the **Wrist warning delta** value on the **Part/Machine** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**), PC-DMIS assigns it to the patterned feature.
- If no matching TIP is found, PC-DMIS inserts an operator comment in the Edit window instead of a TIP command. The comment indicates that the required TIP was not found. You must create or select the required TIP and modify the measurement routine as needed.

Paste With Pattern - Non Curve Procedure

To create a non-curve pattern, follow this procedure:

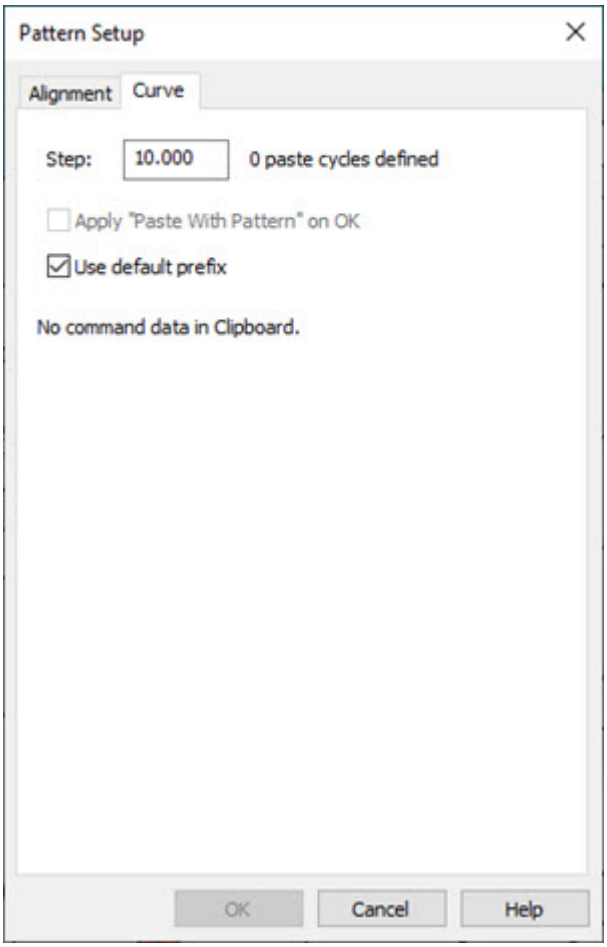
1. Define the Auto feature, features, and other commands that you want to paste, and then copy them to the Clipboard. If you want to copy and paste one or more Auto features and want to use a predefined clearance plane, make sure that you have set `CLEARPLANE=ON` inside the Auto features you are copying.
2. Choose **Edit | Pattern** to open the **Pattern Setup** dialog box.
3. Access the **Alignment** tab and type the desired offset values in the **Offset** area. If you're doing an angle pattern, type an angle offset, and then click on the center hole to get the values for the **Center of Rotation** area.
4. If you want to mirror the axis, select the desired option from the **Mirror** area.
5. Type the number of times to offset in the **Number of times to offset** box.
6. Mark **Apply "Paste With Pattern" on OK**.
7. Click **OK** to store the definition and to paste the defined pattern.

Pattern Setup - Curve Tab

The **Curve** tab on the **Pattern Setup** dialog box allows you to set up a pattern for a part with one or more edge segments around a part surface. These segments can be straight edges or curved edges. With this tab open, you can click on your CAD model to select an edge. To select multiple edges, press Ctrl and click on the part to add a new edge. You may find this tab helpful to measure a profile comprised of several points pasted around a part's surface.

To remove a selected edge, hover over it and click on it again. To remove the last selected edge, click the blue ball.

When you paste the pattern, the copied commands are pasted at the defined offset value along the selected edges.



Step

This defines the offset distance between patterned features. The default is 10. The units are based on the measurement routine units which get converted accordingly. The **Step** value must be greater than 0 (zero).

When you type a value in the **Step** box, you need to tab out of the box or click outside the box to set the value. PC-DMIS updates the text to the right of the **Step** box based on the value you typed.

Apply "Paste With Pattern" on OK

You can manually select the **Paste With Pattern** menu option if you like, or you can use this check box. This check box applies the **Paste With Pattern** operation behind the scenes automatically when you click **OK**.

Use default prefix

The **Use default prefix** check box specifies whether PC-DMIS uses a default ID prefix when you use the **Paste With Pattern** menu item to paste features.

- If you select this check box, the **Paste With Pattern** menu item uses the default ID prefix when you paste new features.
- If you clear this check box, the **Paste With Pattern** menu item uses each feature ID as the base feature name and adds an incremental and numerical suffix.

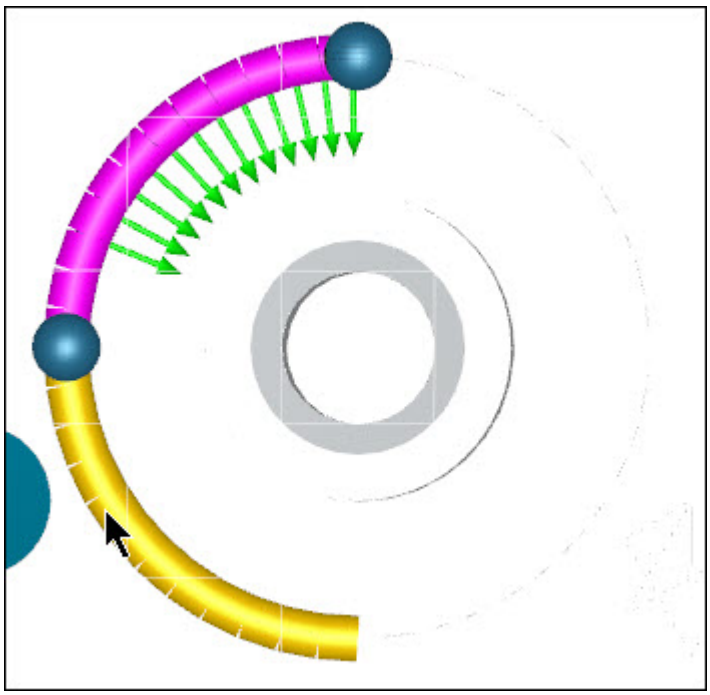
If you don't use a default prefix when you paste many lines, the processing time is longer because each feature ID represents a new pattern.

Once you click the **OK** button, you can press the Esc key to stop the Paste With Pattern process. Features that have already been pasted remain, but PC-DMIS does not paste any new features.

Paste With Pattern - Curve Procedure

To create a curve pattern with one or more edge segments, follow this procedure:

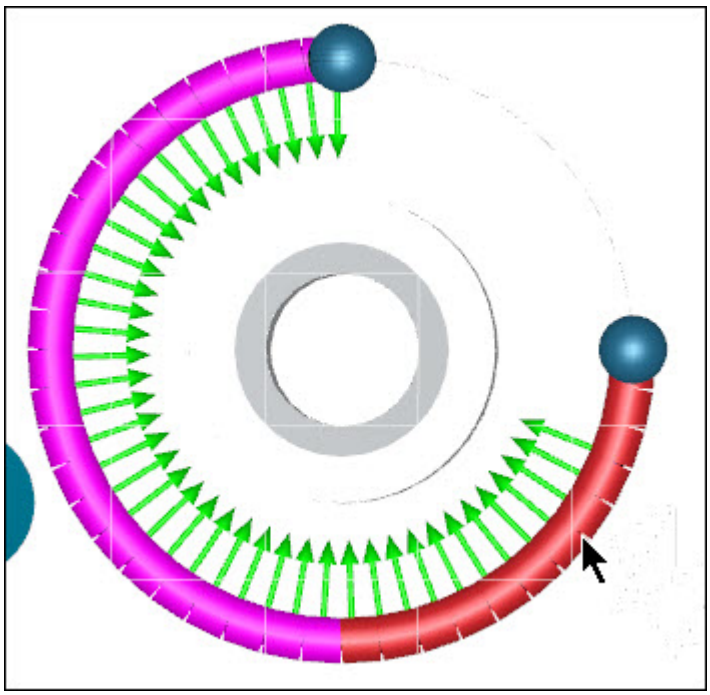
1. Define the Auto feature, features, and other commands that you want to paste, and then copy them to the Clipboard. If you want to copy and paste one or more Auto features and want to use a predefined clearance plane, make sure that you have set `CLEARPLANE=ON` inside the Auto features you are copying.
2. Choose **Edit | Pattern** to open the **Pattern Setup** dialog box.
3. Access the **Curve** tab and type the desired **Step** value in the current measurement routine units (mm or inch).
4. Keep the dialog box open and then hover your pointer over the first segment. It turns yellow.
5. Click on that segment to select it. The selected segment turns pink.



Example of an added segment (pink) and a new segment to add (yellow).

6. If you want to select multiple segments, hold down Ctrl before each additional click on segments that you want to add.

7. If you make a mistake and want to remove the last segment (or if PC-DMIS interprets the vectors incorrectly), press Ctrl and hover your pointer over the last-selected segment. When it turns red, click on that segment again to remove it. You can then re-select the segment and select a different segment.



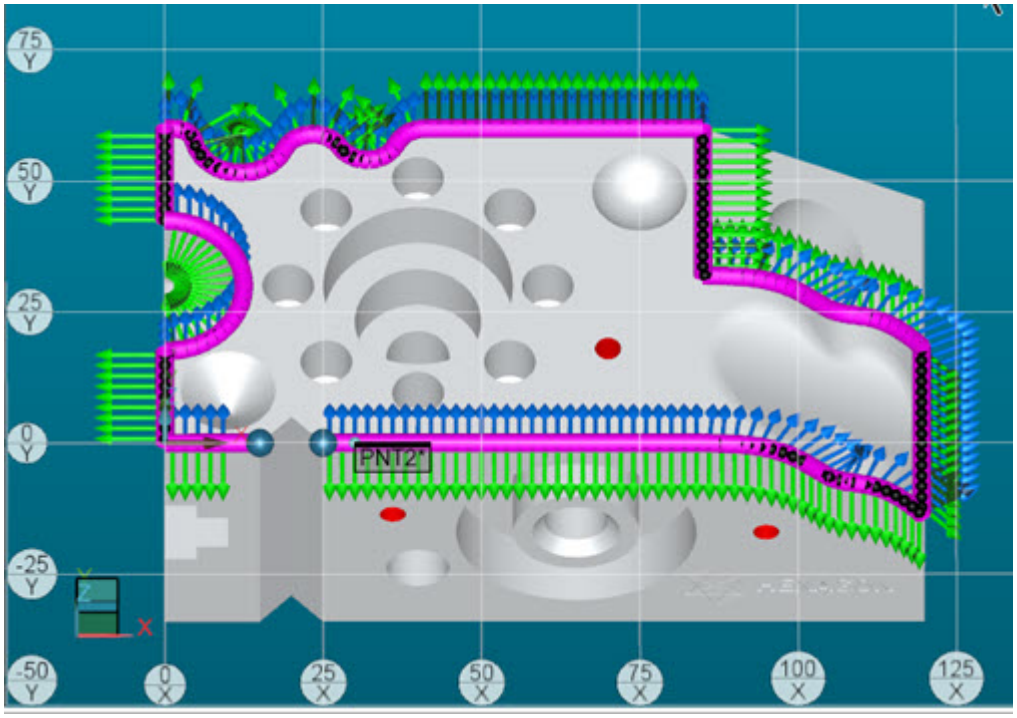
Example of a segment to delete (red).

8. Mark **Apply "Paste With Pattern" on OK**.

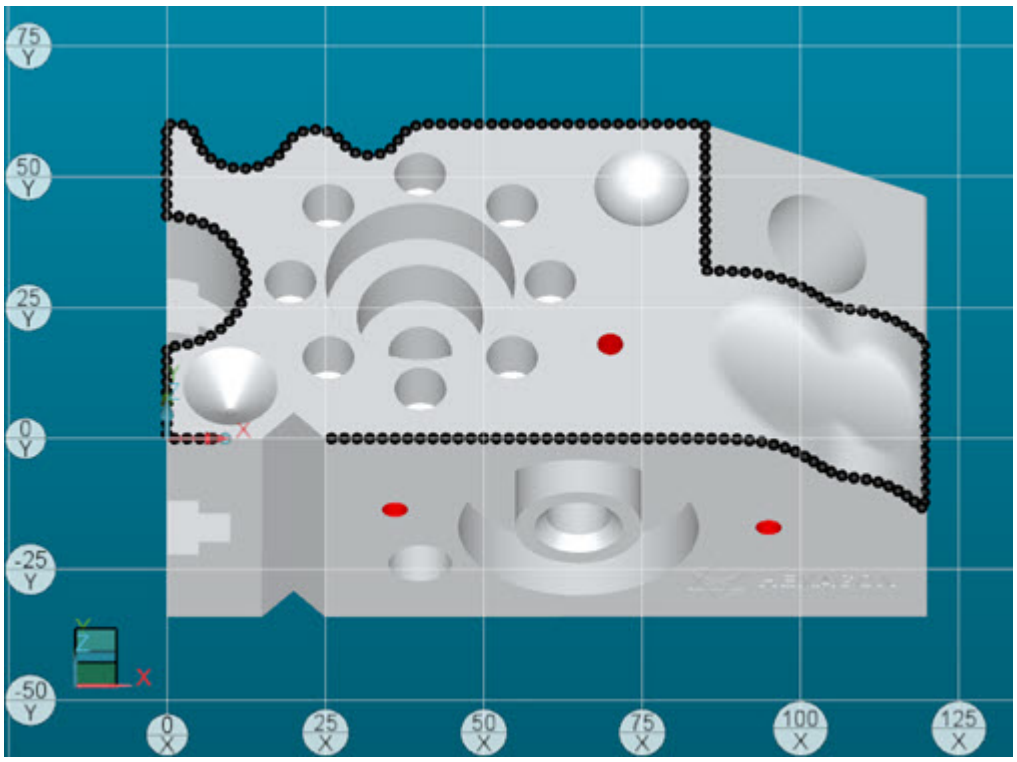
9. Click **OK** to store the definition and to paste the defined pattern.

Example

This first image shows a paste with pattern of a single auto edge point feature pasted around the top surface of the Hexagon demo block at a step value of 2 mm. With the dialog box open, PC-DMIS shows blue and green arrows for the surface and edge vectors of each point. The pink lines and curves indicate the selected edges:



When you click **OK** to accept the pasted pattern and hide the feature labels, the above example results in hundreds of generated point commands equally spaced along the selected edges:



Paste With Pattern

The **Edit | Paste With Pattern** menu item pastes a feature or group of features from the Clipboard into the Edit window using the offsets and other parameters defined in the **Pattern Setup** dialog box.

You don't need to select this menu item if you marked the mark the **Apply "Paste With Pattern" on OK** check box on the **Alignment** or **Curve** tabs of the **Pattern Setup** dialog box.

PC-DMIS creates these new features based on those that were copied to the Clipboard.

PC-DMIS draws the new features in the Graphic Display window.

PC-DMIS assigns a new ID to each feature that it creates, so the new features do not use the same IDs as the copied features.

For Non-Curve Patterns

PC-DMIS applies features with the pattern offsets defined in the **Alignment** tab of the **Pattern Setup** dialog box.

PC-DMIS pastes the contents of the Clipboard into the Edit window by the number of times shown in the **Number of times to offset** box in the **Alignment** tab of the **Part Setup** dialog box.

Each time PC-DMIS pastes the Clipboard's contents, the new features are offset with respect to the previously inserted features by the defined pattern offsets.

PC-DMIS generates feature IDs based on how the **Use default prefix** check box in the **Pattern Setup** dialog box is set.

For Curve Patterns

PC-DMIS pastes the contents of the Clipboard into the Edit window equally spaced by the **Step** value among all the selected edges.

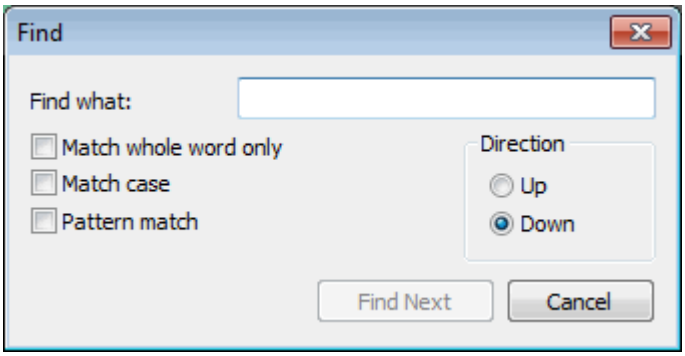
Finding and Replacing Text

You can find and replace text in Edit window fields by using standard regular expressions described in these menu options:

- Find
- Replace

Find

The **Edit | Find and Replace | Find** menu option opens the **Find** dialog box which allows you to find a specified keyword within the Edit window.



Find Dialog Box

Using Wildcard Characters

Select the **Pattern match** check box in the **Replace** dialog (**Edit | Find and Replace | Replace**) box to use wildcard characters to perform pattern matching.

This option also allows you to find text matching a variety of different search criteria. Consider the following table:

Text To Find	Wildcard to Use	Some Examples
Any one character	.	D.M matches "DIM" or "D4M".
Character set	[ ]	D[MI]S finds "DIS" and "DMS" but not "DMIS" or "DUS".
Any one character in a range	[-]	LINE[2-6] finds "LINE2" and "LINE3" but not "LINE1" or "LINE7". Ranges must be in ascending order.
Any one character except character(s) inside brackets	[^]	POINT[^32] finds "POINT1", "POINT5", and "POINT12", but not "POINT3", "POINT2", "POINT21", or "POINT30".
Any one character except range of characters inside brackets	[^x-z]	LINE[^2-5] finds "LINE6" and "LINEH", but not "LINE3"
Character sets that include the "-" character should have the "-" as the first or last character of the range. If the range excludes the "-" then the "-" should be the second character.	[-]	LINE[0-9-] matches "LINE4" and "LINE-". LINE[^-0-9] matches "LINEH" but not "LINE-".
0 or more instances of the preceding character, unless the following: "*" starts a string "*" follows a "^" that starts a string "*" starts a parenthesized sub-expression	*	lo*p matches "lp" and "looooop". ^* matches a "*" if it starts a line.

Line beginning with a specified phrase or character. The "^" is special only at the beginning of a string.	^	^CIRCLE finds all lines that begin with the word "CIRCLE". Note that indented lines in the software's Edit window begin with spaces.
Line ending with a specified phrase or character. The "$" is special only at the end of a string.	$	ENDMEAS/$ finds lines ending in "ENDMEAS/" but not "MEAS/"
Parentheses in the **Find What** box store what is within them to be recalled later in the **Replace With** box. In the **Replace With** box a backslash "\" character, followed by a number would bring that defined pattern into the replace text.	() and \	Find What: BO(BB)Y(RAY) Replace With: DO1YP2 Would Give: DOBBYPRAY \1 uses the first set of characters offset by parentheses, \2 the second set and so on.

For pattern matches, you no longer need to precede parentheses with a backslash (\) character as you had to do in some earlier versions of PC-DMIS.

Using Predefined Character Sets

To match a predefined character set, select the **Pattern match** check box. You can then search the text by using these special predefined character sets:

Character set	Characters in set
[[:alnum:]]	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz 0123456789
[[:alpha:]]	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz
[[:blank:]]	(The space character and the tab character)
[[:digit:]]	0123456789
[[:graph:]]	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz 0123456789 !"#$%&'()*+,-./:;<=>?@[]^_`{ }~
[[:lower:]]	Abcdefghijklmnopqrstuvwxyz
[[:print:]]	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz 0123456789 !"#$%&'()*+,-./:;<=>?@[]^_`{ }~
[[:punct:]]	!"#$%&'()*+,-./:;<=>?@[]^_`{ }~
[[:upper:]]	ABCDEFGHIJKLMNOPQRSTUVWXYZ
[[:xdigit:]]	0123456789ABCDEF abcdef

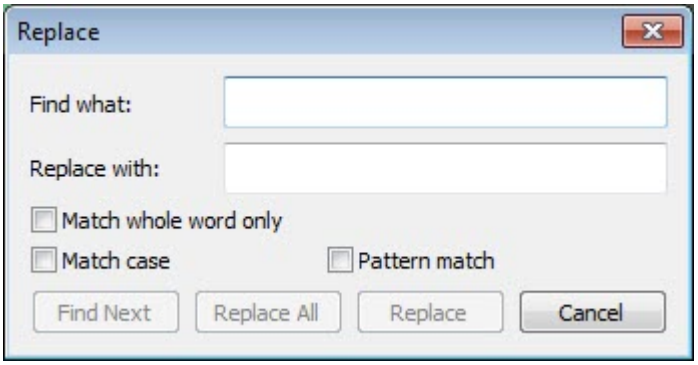
To search for a string which consists of a letter, followed by some letters or numbers, followed by that same first letter, you would use a back reference, as shown here: ([[:alpha:]])[[:alnum:]]\{1,\}\1.

To Search for a Specific Word or Phrase:

1. Select **Edit | Find and Replace | Find** from the menu bar.
2. Type in a keyword or phrase to search.
3. Indicate the direction to search (**Up** or **Down**).
4. Select the **Match whole word only** check box if you want PC-DMIS to match the entire word. For example, if you type "CIR", PC-DMIS will find "CIR" but not "CIRCLE".
5. Select the **Match Case** check box if you want PC-DMIS to search based on upper or lower case.
6. Press the **Find** button (or press the Enter key).

Replace

The **Edit | Find and Replace | Replace** option works much the same way as the **Find** option (see "Find"). Once the text is found, however, you can replace that text by using the **Replace** dialog box.



Replace Dialog Box

You can search and replace with any matching string as long as the last part of the string corresponds to an editable field, and as long as it is the only editable field in the string that will be changing.

A Valid Find and Replace Example:

Find string:	`=AUTO/CIRCLE,SHOWALLPARAMS= NO`
Replace string:	**`=AUTO/CIRCLE,SHOWALLPARAMS= YES`**

Notice that the end of the string corresponds to an editable field. You can only change the last editable field of the search string.

Find and Replace Examples That Are Not Valid:

You *cannot* replace two editable fields in one replace as in this example:

Find string:	`=AUTO/CIRCLE,SHOWALLPARAMS= NO`
Replace string:	**`=AUTO/LINE,SHOWALLPARAMS = YES`**

This is not valid because CIRCLE to LINE is one editable field, and NO to YES is another editable field.

You *cannot* replace an editable field with a non-editable field as in this example:

Find string:	`=AUTO/CIRCLE,SHOWALLPARAMS`
Replace string:	**`=AUTO/CIRCLE,MYSHOWALL`**

This is not valid because `MYSHOWALL` is a constant and not an editable field.

Find What

You can use this box to type the text for PC-DMIS to locate. You can use any of the various wild cards and search patterns discussed in the "Find" section. See "Find".

Replace With

You can use this box to type the text that you want to replace the found text.

Match Whole Word Only

If you select this check box, PC-DMIS only looks for an entire word containing that text. For example, if you type in "CIR" in the **Find what** box, and select this check box, PC-DMIS finds only "CIR". It does not find "CIRCLE" or "CIRCULAR".

Match Case

This check box tells PC-DMIS to search for text matching the case (upper or lower) used in the Find what box. For example, if you type "Point" in the **Find what** box and selected the **Match case** check box, PC-DMIS finds "Point" and "Points", but not "POINT".

Pattern Matching

This check box tells PC-DMIS to look for specified patterns. You should select this check box when searching for pattern matches using wildcard characters or other special character matching.

Find Next Button

The **Find Next** button tells PC-DMIS to find the next instance of the text for which you're searching.

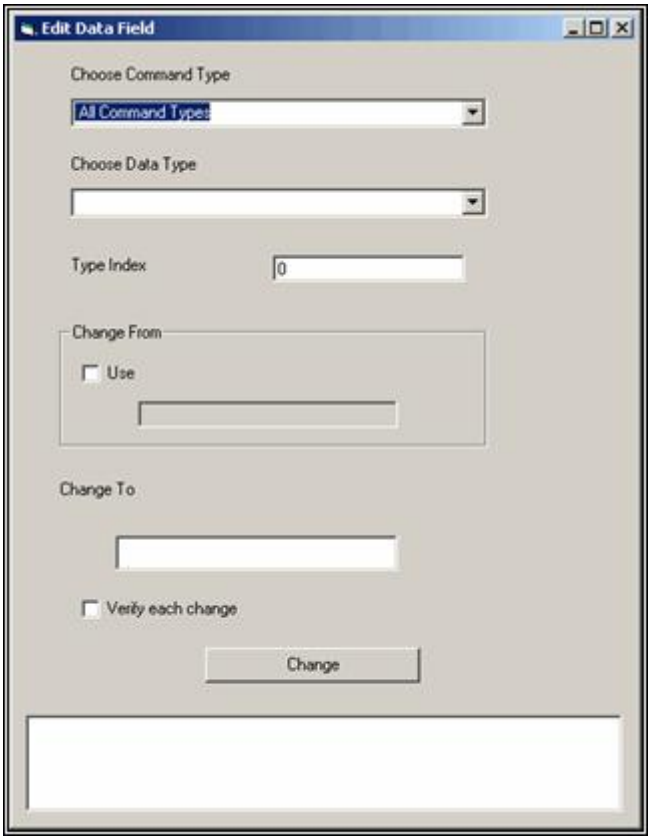
Replace Button

The **Replace** button replaces the found text with the text string in the **Replace with** box. This allows you to replace instances of found text one at a time.

Replace All Button

The **Replace all** button tells PC-DMIS to instantly find all occurrences of text in the **Find what** box and replaces them with the text in the **Replace with** box.

Editing Data Fields



Edit Data Field Wizard

The Edit Data Field wizard is an external program that works with PC-DMIS to let you change data inside specified data fields of certain command types inside the Edit window. To access this wizard, select the **DataFieldEdit** icon from the **Wizards** toolbar.

This wizard differs from the **Replace** dialog box. It allows you to change data items globally inside any mode of the Edit window, whereas the **Replace** dialog box only works inside of Command Mode. Additionally, this will allow you to replace certain values that you cannot change using the **Replace** dialog box.

The **Edit Data Field** dialog box contains these items:

Choose Command Type – This list displays the available command types.

Choose Data Type – This list displays the data types for the selected command type. If you want to see all data types in this list, select **All Command Types**, from the **Choose Command Type** list.

Type Index – This box lets you specify an exact field you want to change by its numerical index. You will find this useful when you have more than one field of the same data type and you want to change only one of the fields. Typing an index value lets you determine which one to change. For example, the `CLEARP` command has two distance fields, one for the clearance distance and one for the pass distance. Typing an index of 1 would target the clearance distance while a value of 2 would target the pass distance.

Change From – This area contains the **Use** check box and a box that holds a value. This check box lets you specify a value you want to change from. This means that PC-DMIS will only target those fields that have the given value.

Change To – This box lets you determine the new value for the data field you want to update.

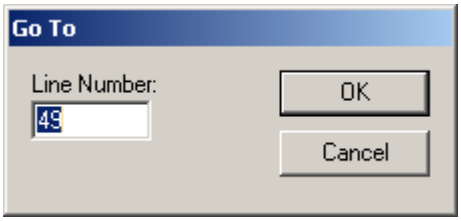
Verify Each Change – If you select this check box, PC-DMIS will display a message box showing the ID of the item to change, its current value, and whether or not you want to change that data field to the new value.

Change – This button begins the process of finding the data field and changing it to the new value.

Status – The box at the bottom of the dialog box displays any status or diagnostic messages during the change process.

Jumping to Specified Line Numbers

PC-DMIS can easily jump to specified line numbers. Use the **Edit | Find and Replace | Go To** menu option to jump to a specified line within the Edit window.



Go To Dialog Box

To move the cursor to a specific line number in *Command* or *DMIS Mode*, follow these steps:

1. To open the **Go To** dialog box, choose the **Edit | Go To** menu option.
2. The **Go To** dialog box initially shows the line that the cursor is currently on in the **Line Number** box.
3. Type the desired line number to jump to.
4. Click the **OK** button (or press Enter). PC-DMIS locates and places your cursor on the specified line.

Jumping to and from Referenced Commands

In the Edit window's Command Mode, if one command in the Edit window references another command, you can jump to the referenced command, by highlighting the label to the referenced command and selecting the **Edit | Find and Replace | Jump To** menu option.

When you arrive at the referenced command, you can quickly jump back to the previous feature by selecting the **Edit | Find and Replace | Jump Back** menu option.

Suppose in an alignment block you have the following commands and you want to jump to the LIN2 feature referenced in the block:

```
A1= ALIGNMENT/START,RECALL:STARTUP, LIST= YES
    ALIGNMENT/LEVEL,ZPLUS,PLN1
    ALIGNMENT/ROTATE,XPLUS,TO,LIN1,ABOUT,ZPLUS
    ALIGNMENT/TRANS,XAXIS,LIN2
    ALIGNMENT/TRANS,YAXIS,LIN1
    ALIGNMENT/TRANS,ZAXIS,PLN1
    ALIGNMENT/END
```

You would highlight LIN2 and select the **Jump To** menu option from the menu bar or right-click and select it from the shortcut menu. You can then jump back to the command you were just at by selecting **Jump Back**.

You can also select these menu options from a shortcut menu that appears when you right-click on the features in the Edit window's Command Mode.

Editing Nominal and Target Values

Most of the time you can measure your features just fine, but sometimes, due to manufacturing errors, a feature won't be close to its theoretical (nominal) location, but rather shifted somehow. For Measured features, you need to adjust your theoretical values to handle this situation. However, using the more advanced Auto features, you can modify the target instead. This will adjust the path accordingly, yet will allow dimensioning to take place relative to the original theoretical values.

Modifying Nominal Values (for both Measured and Auto Features)

PC-DMIS provides you with simple ways to modify the nominal value. To modify the nominal value field in Command Mode, place the cursor over the feature then press the F9 key. A dialog box will appear displaying the nominal information which can be updated as necessary.

Modify the nominal information and click **OK** or **Create**. The dialog box closes.

- If you are modifying a measured feature, PC-DMIS asks if you want to update measured values as well. If you click **Yes**, the measured values are updated to match the newly entered nominal values. If you click **No**, the measured values remain what they were.
- If you are modifying an Auto feature, PC-DMIS asks if you want to update the target values to match the new theoretical values. If you click **Yes**, the target values are updated to match the newly entered nominal values. If you click **No**, the target values remain what they were.

When you have an associated dimension, if you then change the associated nominal values in the feature, a message box appears. If you choose **Yes** to update the dimension nominals, PC-DMIS updates the nominal information in the associated legacy dimension command. Be aware that Geometric Tolerance commands always use the feature nominals. For information on FCF dimensions, see the "Using Geometric Tolerances" chapter.

Modifying Target Values (for Auto features only)

If you have an Auto feature, you can choose to modify target values instead of the theoretical values. The target values determine the location where PC-DMIS will try to measure the Auto feature. This lets you keep the theoretical values for dimensioning purposes, but still changes the location at which the software attempts measurement. To modify the target values, use the Edit window:

Modifying Target Values in Command Mode: Press the Tab key to move your cursor to the field to change, enter the new value, and press the Tab key again to store the change.

You cannot modify the target values in Summary Mode.

Overriding Guessed Features

The **Edit | Override Guess** submenu allows you to change the type of feature last measured.

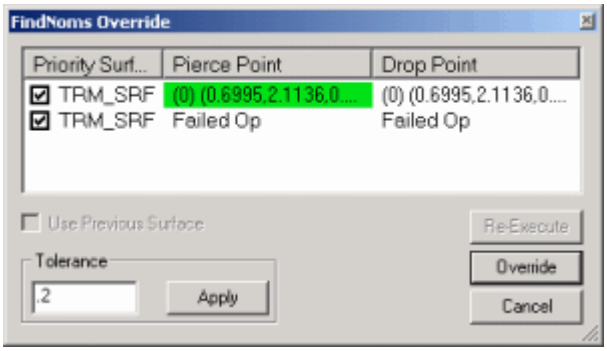
For example, if a circle was the last feature measured and PC-DMIS guessed that it was a plane, you can use this option to convert the feature to the proper type. You must use this option before any other features are measured or constructed.

To use the Override feature:

1. Access the Edit window.
2. Place your cursor on the incorrect feature type in the Edit window.
3. From the **Override Guess** submenu, select the correct feature type.

PC-DMIS updates the feature type in the Edit window.

Overriding Found Nominals



FindNoms Override Dialog Box

The **FindNoms Override** dialog box (**Edit | Override Find Noms**) allows you to choose the appropriate nominal point from CAD surfaces for Auto Vector Points, Auto Surface Points, and individual hits inside other features (such as scans). This dialog box overrides the default FindNoms behavior of PC-DMIS during Learn mode and Execute mode.

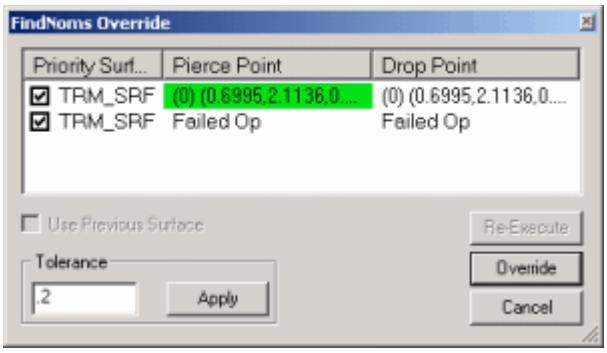
When attempting to find a valid nominal point (within the FindNoms tolerance) by taking a manual hit, PC-DMIS will, most of the time, choose the correct point. However, there are exceptional circumstances when PC-DMIS cannot find an appropriate point and instead chooses an inaccurate point on the CAD model. In these situations, you can use the **FindNoms Override** dialog box to help PC-DMIS find the appropriate nominal point from the CAD surfaces.

To enable the **Override Find Noms** menu option, do the following:

1. Ensure that you have imported a valid CAD model that supports surfaces.
2. Select the Surface Mode icon.
3. Select the Program Mode icon.
4. From the **Setup Options** dialog box, from the **General** tab (**Edit | Preferences | Setup**), select both the Find Nominals and Point Only Mode check boxes.

The following topics describe the **FindNoms Override** dialog box itself and how to use the dialog box in the different modes:

Understanding the FindNoms Override Dialog Box



FindNoms Override Dialog Box

The following topics describe the **FindNoms Override** dialog box (**Edit | Override FindNoms**).

The Priority Surface, Pierce Point, and Drop Point Columns

This area of the dialog box contains these three columns that display the relevant nominal point values and their T-values (the value displayed to the left of the X, Y, and Z values in parentheses):

A "T-value" is the distance of the measured hit from the nominal point along the nominal CAD surface vector.

- **Priority Surface** - This column lists the surfaces that you have stored as priority surfaces in the **Edit CAD Elements** dialog box (**Edit | Graphic Display Window | CAD Elements**). It displays the type of surface and a check box. The check box allows you to add or remove the surface from the list of priority surfaces maintained by PC-DMIS. If you select the check box, PC-DMIS adds the surface to the priority list. Deselecting the check box removes the surface from the list.
- **Pierce Point** - This column lists the nominal point values found using PC-DMIS's Pierce Point operation. If you see the text "Failed OP" in this column, it means that the pierce point operation failed to find the nominal point on that surface. If PC-DMIS highlights a point in green then that point is the best nominal point found by the selection algorithm. You can, of course, override this selected point.
- **Drop Point** - This column lists the nominal point values found using PC-DMIS's Drop Point operation.

The Tolerance Box

The **Tolerance** box allows you to specify a new tolerance value to use when finding the nominal points using this dialog box.

The Use Previous Surface Check Box

The **Use Previous Surface** check box tells PC-DMIS to use the most recent surface to look for the nominal points. This check box becomes available for selection only after you take a hit on a surface with the dialog box open. After you take a hit, PC-DMIS then has a surface to work with for future hits.

The Override Button

The **Override** button overrides an incorrect nominal point found by PC-DMIS with the nominal point data selected from the list containing the **Priority Surface**, **Pierce Point**, and **Drop Point** columns.

Using the FindNoms Override Dialog Box in Learn mode:

To access this dialog box in Learn mode:

1. Select the **Point Only Mode** and **Find Nominals** check boxes from the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**).
2. Ensure that you have imported a CAD's solid model.
3. Select the **Surface Mode** icon from the **Graphics Modes** toolbar.
4. Select the **Override FindNoms** menu option. The **FindNoms Override** dialog box appears.

With the **FindNoms Override** dialog box open, when you take a manual hit, PC-DMIS follows its normal logic to choose as accurate a nominal point as possible by using the most recently measured hit coordinates and approach vector.

You can then use the **FindNoms Override** dialog box to perform additional procedures allowing you to override the found nominal point.

Procedure 1: Finding Nominal Points on the Previous Surface

If you select the **Use Previous Surface** check box, the FindNoms Override dialog box attempts to find a nominal point from the most recent surface used. If the dialog box can find a point within FindNoms tolerance on the most recent surface it will select that nominal point and create an Auto Vector Point based on the nominal values.

Procedure 2: Finding Nominal Points on Priority Surfaces

If the **FindNoms Override** dialog box cannot find a nominal point on the recent surface (see "Procedure 1: Finding Nominal Points on the Previous Surface"), it tries to find a nominal point from the *priority surfaces*. Priority surfaces are surfaces that the user has selected as a subset of CAD surfaces to find nominal points. Priority surfaces can be selected or deselected using the **Set as Priority** check box in the **Edit CAD Elements** dialog box (see the "Editing CAD" topic in the "Editing the CAD Display" section). If PC-DMIS finds an appropriate point among the priority surfaces, it selects the point and creates an Auto Vector Point based on the nominal values. It also displays the valid nominal points within the FindNoms tolerance of all the priority surfaces in the **FindNoms Override** dialog box.

Procedure 3: Finding Nominal Points on All CAD Surfaces

If PC-DMIS still cannot find a valid nominal point by using the procedures discussed in procedures 1 and 2, (see the "Procedure 1: Finding Nominal Points on the Previous Surface" and "Procedure 2: Finding Nominal Points on Priority Surfaces" topics), it does the following:

- Looks for the best nominal point on all the CAD surfaces within the FindNoms tolerance
- Creates an Auto Vector Point based on the best available nominal point.
- Displays all the CAD surfaces that had nominal points in the **FindNoms Override** dialog box.

Procedure 4: Choosing a Nominal Point Later

If PC-DMIS still cannot find an appropriate point on all the CAD surfaces, it asks if it can create the Auto Vector Point and you can choose a nominal point later. If you accept, then PC-DMIS creates an Auto Vector Point in the Edit window and you can select from these options:

Option 1 - You can take a new hit and PC-DMIS will try to find a new nominal point. Essentially you're starting over if you do this.

Option 2 - You can choose a different nominal point from the dialog box and click the **Override** button. PC-DMIS will then override its earlier chosen nominal point with the new selected point for the recently created Auto Vector Point.

You will usually want to do this if you don't like the nominal point that PC-DMIS attempted to find in the procedures discussed in these topics:

- "Procedure 1: Finding Nominal Points on the Previous Surface"
- "Procedure 2: Finding Nominal Points on Priority Surfaces"

- "Procedure 3: Finding Nominal Points on All CAD Surfaces"

Options 3 - You can select a CAD surface on the screen. PC-DMIS would then attempt to find nominal points on the selected surface and display those points in the **FindNoms Override** dialog box.

You can then override the previous found nominal, by selecting a point from this new surface and clicking **Override**.

PC-DMIS ignores the FindNoms tolerance with this option. So, even if the selected CAD surface is out of FindNoms tolerance, the dialog box still displays the points.

Option 4 - You can increase the FindNoms tolerance by changing the value in the **Tolerance** box and clicking the **Apply** button. PC-DMIS will then apply these procedures again, using the higher tolerance value and display any appropriate points:

- "Procedure 1: Finding Nominal Points on the Previous Surface"
- "Procedure 2: Finding Nominal Points on Priority Surfaces"
- "Procedure 3: Finding Nominal Points on All CAD Surfaces"

You can then choose a valid point and click **Override** to accept a nominal point for the recently created Auto Vector Point.

Increasing the FindNoms tolerance in the **Tolerance** box has the same effect as changing the FindNoms tolerance from the **General** tab of the **Setup Options** dialog box.

After Choosing the Best Nominal Point

After you choose the best nominal point, PC-DMIS remembers the selected surface for that feature.

During execution in Manual Mode, PC-DMIS:

1. Uses the tolerance value in the **FindNoms During Execution** box on the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**). For this to work, you must select the **Find Nominals During Execution** check box on the **General** tab.
2. Attempts to use the same surface and CAD operation (Pierce Point or Drop Point operation) to find the new nominals for the recently executed point.

If PC-DMIS cannot find a nominal point on that CAD surface, it opens the **FindNoms Override** dialog box again. For information on the execution time behavior, see the "Using FindNoms Override in Execute Mode:" topic in the PC-DMIS Core documentation.

Using FindNoms Override in Execute Mode:

As mentioned in the "After Choosing the Best Nominal Point" topic, PC-DMIS displays the **FindNoms Override** dialog box in execute mode upon meeting the following conditions:

- You must import a solid model with surfaces and select the **Surface Mode** icon from the **Graphic Modes** toolbar.
- You must select the **Find Nominals During Execution** check box from the **General** tab of the **Setup Options** dialog box.
- The Auto Vector Point or Auto Surface Point must have a valid nominal point found in Learn mode (see "Using the FindNoms Override Dialog Box in Learn mode:").
- Execution must take place in Manual Mode.
- PC-DMIS must execute the Auto Vector Point or Auto Surface Point feature.
- PC-DMIS must fail to find a nominal point for executed Auto Vector Point or Auto Surface Point.

Once the dialog box appears, execution stops and you can interact with the dialog box in ways already discussed in the "Using the FindNoms Override Dialog Box in Learn mode:" topic.

You can also choose to re-execute the recently measured point feature. In that case, PC-DMIS re-executes and attempts to find nominal values again. If it still can't find any nominal values for the point, PC-DMIS again displays the **FindNoms Override** dialog box.

You can then do one of the following:

- Re-execute again.
- Use the **Override** button as discussed in the "Procedure 4: Choosing a Nominal Point Later" topic
- Close the dialog box by clicking the **Cancel** button. If you close without choosing an appropriate nominal, PC-DMIS replaces the measured data with the previous nominal values.

Using the FindNoms Override Dialog Box from the Edit Window

To access the **FindNoms Override** dialog box, right-click on one of these features and select **Override FindNoms** from the shortcut menu:

- Vector point

- Surface point
- Individual Hit

The dialog box then uses the feature's measured data (point and vector) to find an appropriate nominal point as discussed in the "Using the FindNoms Override Dialog Box in Learn mode" topic.

Marking Commands for Execution

In order to execute PC-DMIS commands, you must first mark them for execution. The following menu options and commands control marking:

Mark

From the **Edit | Markings | Mark** menu option, you can mark a specified feature or command for execution. By default, whenever you create a new measurement routine, newly added features and commands are already marked for execution. If you select **Mark** on an already marked item, the marking is cleared.

Subroutines are an exception to unmarked commands. PC-DMIS executes unmarked commands in subroutines whenever subroutines are called. For information on calling subroutines, see "Calling a Subroutine" in the "Branching by Using Flow Control" chapter.

PC-DMIS indicates a marked item using the current Marked Text Color in the Edit window. PC-DMIS also displays an asterisk (*) next to the feature ID in the Graphic Display window. For information on the Marked Text Color, see "Defining Edit Window Colors" in the "Setting your Preferences" chapter.

You can mark features or commands in any of these ways:

- *Mark the desired feature or command in the Edit window.* Position the cursor over the feature to be marked, and select **Mark**. PC-DMIS highlights the marked command.
- *Mark multiple commands in the Edit window.* Select multiple commands to be marked in the Edit window, and select **Mark**. PC-DMIS highlights the marked command.
- *Mark the desired feature in the Graphic Display window.* Make sure PC-DMIS is in Translate Mode. Hold down the Alt key and click the desired feature.
- *Mark Multiple features in the Graphic Display window.* Make sure PC-DMIS is in Translate or Text Box Mode. Hold down the Alt key and drag a box around the desired features.

If you select **Edit | Markings | Parent Mode**, and you mark a constructed feature or a dimension, PC-DMIS also marks any related features used in the construction or dimension process. For details on Parent Mode, see the "Parent Mode" topic in the PC-DMIS Core documentation.

If you select **Edit | Markings | Child Mode**, and mark a feature that is used to construct a feature or create a dimension, PC-DMIS also marks any related child features. For details on Child Mode, see the "Child Mode" topic in the PC-DMIS Core documentation.

Features and commands can be unmarked by repeating one of the above procedures on marked features. To unmark all features, you can select the **Clear Marked** menu item.

Once an item is marked, specific lines within some items can be unmarked. For example, you can unmark a `HIT/BASIC` line in a feature. When you unmark such a line, PC-DMIS does not execute that line.

If dimensions are marked after they are executed, PC-DMIS sends the results to the Report window or statistics file based on the dimension output and the report output configuration. For more information, see "Output To" in the "Using Legacy Dimensions" chapter.

To Unmark Specific Lines Within a Feature:

1. Move the cursor to the desired line within the marked feature (such as the `HIT / BASIC` line).
2. Re-select the **Edit | Markings | Mark** option to unmark the indicated line.

Mark All

The **Edit | Markings | Mark All** menu option marks all of the features or commands in the Edit window. PC-DMIS marks the selected items in the Edit window, displaying them in green colored text. PC-DMIS also displays an asterisk (*) next to each feature ID in the Graphic Display window that is marked for execution.

When you select **Mark All**, PC-DMIS asks if you also want to mark manual alignment features.

- If you select the **Yes** button, then PC-DMIS marks the entire measurement routine for execution, including the alignment features.
- If you select the **No** button, PC-DMIS marks the entire measurement routine for execution, but does not mark the alignment feature. Additionally, since move commands don't function in Manual Mode, they remain unmarked as well.

Clear All

The **Edit | Markings | Clear All** menu option clears (or unmarks) all the items in the Edit window that were earlier marked for execution.

Parent Mode

The **Edit | Markings | Parent Mode** menu option places PC-DMIS in a mode where it marks associated parent commands whenever you mark a child command. Whenever PC-DMIS is in this mode, the check mark next to the **Parent Mode** menu option changes from gray to black.

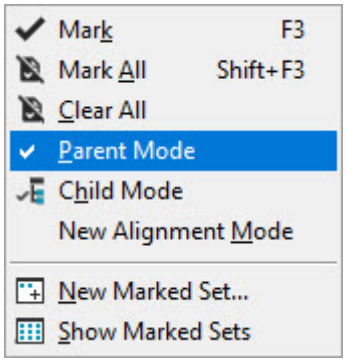
- A "parent" is a command (or information from a command) that is used in another command.
- A "child" is a command that uses information from a parent command.

For example, if you enable **Parent Mode** and then you mark a constructed feature, PC-DMIS marks the constructed feature (a child command), PC-DMIS marks the constructed feature as well as any features used to create the constructed feature (parent command).

The **Parent Mode** and **Child Mode** functions only work when you are in **Command** Mode in the Edit window.

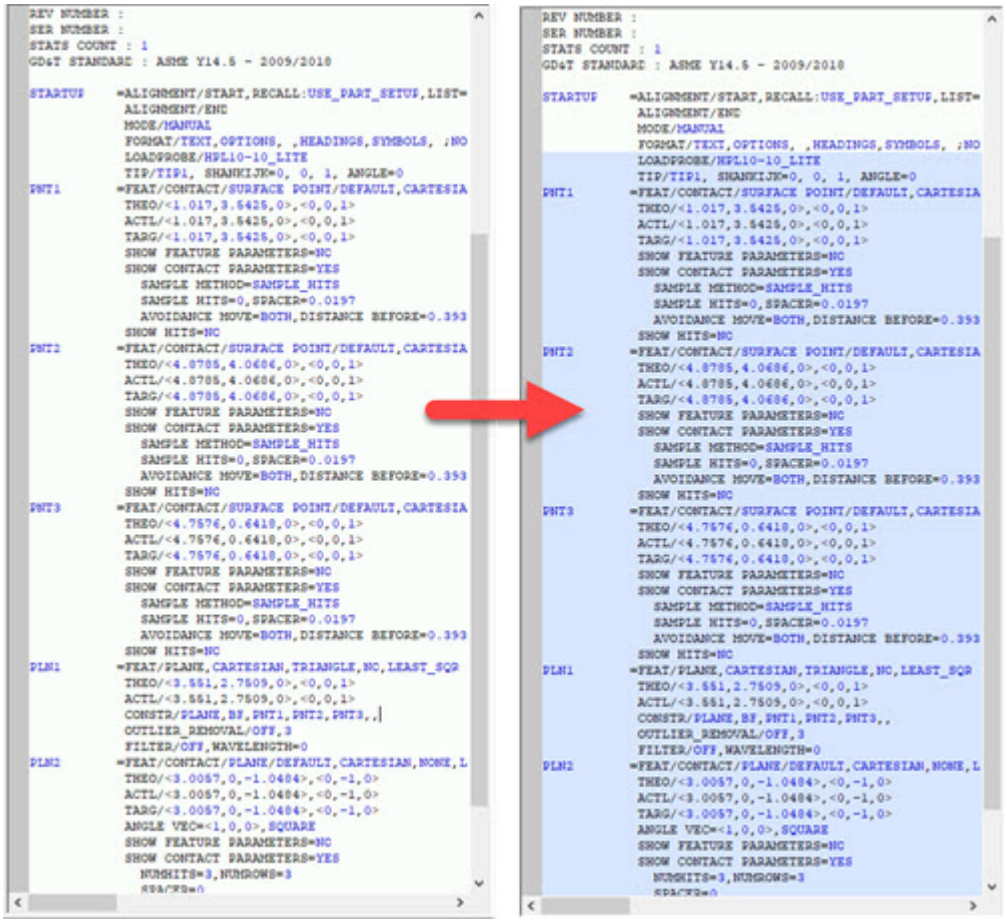
To better understand this:

1. Select and enable the **Parent Mode** option from the menu (**Edit | Markings | Parent Mode**).



2. From the Edit window toolbar (**View | Toolbars | Edit Window**), click the **Mark All** button and then click the **Clear Marked** button .

   You can see in the image below that on the left is what the Edit window looks like after you click the **Mark All** button and on the right is what the same measurement routine looks like after you click the **Clear Marked** button.



In the above image on the right, `PLN1` is constructed from `PNT1`, `PNT2`, and `PNT3`. Those three points are all shown as unmarked.

3. In the Edit window, click anywhere in the `PLN1` command and then from the **Edit Window** toolbar, click the **Mark** button .

   The constructed plane and all of the features that are used to define it are also marked as shown on the right in the image below.

If you unmark a marked child command, the parent command stays marked.

Child Mode

The **Edit | Markings | Child Mode** menu option places PC-DMIS in a mode where it marks associated children commands whenever you mark a parent command. Whenever PC-DMIS is in this mode, the check mark next to the **Child Mode** menu option changes from gray to black.

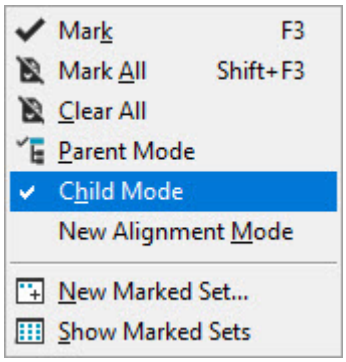
- A "parent" is a command (or information from a command) that is used in another command.
- A "child" is a command that uses information from a parent command.

For example, if you select this option and you unmark a constructed feature and any features used to create it, when you then mark one of the features used to create it (parent feature), PC-DMIS also marks the associated constructed feature (child feature).

The **Parent Mode** and **Child Mode** functions only work when you are in **Command** Mode in the Edit window.

To better understand this:

1. Select and enable the **Child Mode** option from the menu (**Edit | Markings | Child Mode**).



2. From the **Edit Window** toolbar (**View | Toolbars | Edit Window**), click the **Mark All** button and then click the **Clear Marked** button .

   You can see in the image below that on the left is what the Edit window looks like after you click the **Mark All** button and on the right is what the same measurement routine looks like after you click the **Clear Marked** button.

In the image above on the right, `PLN1` is constructed from the unmarked `PNT1`, `PNT2`, and `PNT3` Point features. `PLN2` is constructed from the unmarked `PNT3`, `PNT4`, and `PNT5` Point features. Constructed Planes `PLN1` and `PLN2` share the same `PNT3` child feature.

3. In the Edit window, click anywhere in the `PNT3` command and then from the **Edit Window** toolbar, click the **Mark** button .

   Since the constructed planes, `PLN1` and `PLN2,` share the same parent feature `PNT3`, they are also marked as shown on the right in the image below.

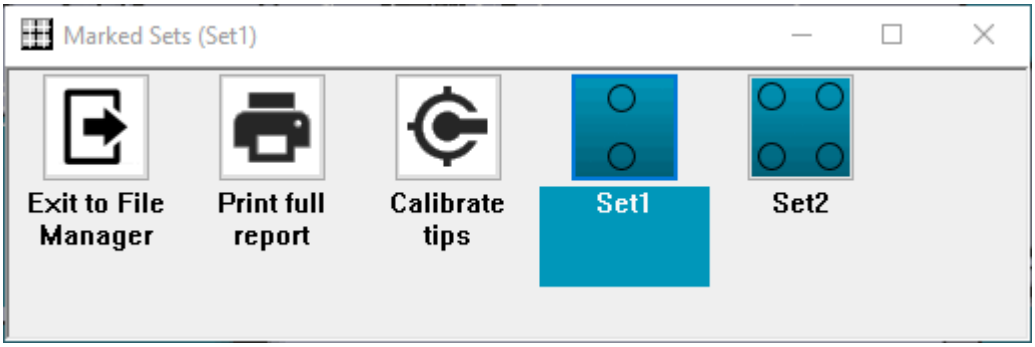
If you unmark a marked parent command, the child command stays marked.

New Alignment Mode

The **Edit | Markings | New Alignment Mode** (indicated with a check mark next to the menu option) marks corresponding alignment data every time a feature/dimension is marked for execution.

Creating and Executing Marked Sets

You can organize marked features into groups called "sets". You can then execute your stored sets of features with the Marked Sets window (**Edit| Markings | Show Marked Sets**).



A Sample Marked Sets window with two created marked sets (Set1 and Set2)

- You can define up to thirty (30) marked sets for a measurement routine.
- When you execute a marked set, PC-DMIS executes only those features within the active marked set.
- PC-DMIS displays every marked set as an icon in the Marked Set window. Each icon graphically indicates all of the features that are associated with that marked set.

When the Marked Sets window is hidden, all marked sets are disabled and the standard mode for executing a measurement routine is followed.

Available Buttons

Exit to File Manager - This button closes any open measurement routines and returns you to the **Open** dialog box. For information on the **Open** dialog box, see "Opening Existing Measurement Routines" in the "Using Basic File Options" chapter.

Print full report - This button sends the current report to the selected output. (For information on printing options, see "Printing the Report Window" in the "Using Basic File Options" chapter.)

Calibrate tips - This button accesses the **Probe Utilities** dialog box so you can calibrate your probe tips. PC-DMIS disables this button for the Romer and Garda interfaces. (For information on probe definition and tip calibration, see "Defining Probes" in the "Defining Hardware" chapter.)

If you click a button for a marked set, PC-DMIS executes the features associated with that marked set.

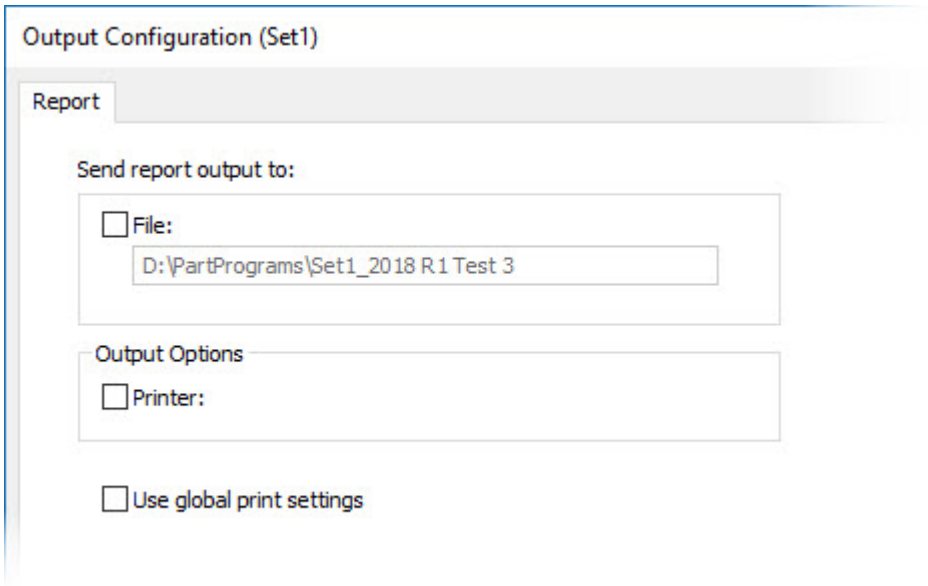
To Create New Marked Sets

1. Select the **Edit | Markings | New Marked Set** option, or double-click anywhere in the Marked Sets window. The **New Mark Set** dialog box appears requesting a name for the new marked set.

New Mark Set Dialog Box

2. In **Enter mark set label**, type a name to identify the marked set. While there's no limit to the length of the name, short descriptive keywords improve readability.

3. Click **OK**. Your new marked set is stored in the Marked Sets window. At this point, PC-DMIS also displays an **Output Configuration** dialog box. You can use this dialog box to define printing options specific to this marked set. (For information on the options in this dialog box, see "Setting Output and Printer Options for the Report Window" in the "Using Basic File Options" chapter.)



Output Configuration Dialog Box

4. Define specific printing options for this set. If you want to use existing print options that are defined for your entire measurement routine, select the **Use Global Print Settings** check box.

5. Click **OK** to close the dialog box.

6. Now, you need to select this Marked Set and add features to it. To do this, follow the procedure in the "To Add or Remove Features on Existing Marked Sets" topic.

To Add or Remove Features on Marked Sets

1. Select the **Edit | Markings | Show Marked Sets** option to open the Marked Sets window.
2. With the Marked Sets window active, select the marked set to modify. You can press Tab to cycle through the sets, or you can click on the set once to select it.
3. To add features, from the Edit window, mark the features you want to add.

You can choose **Edit | Markings | Mark** (or press F3) to toggle the marking state on selected features.

The icon for the set in the Marked Sets window changes dynamically to reflect your changes.

To Customize the Marked Set Icons

Marked sets can have user defined bitmap images, if desired, in place of the default graphics. To apply a user bitmap,

1. Create a color bitmap using the Paint application that comes with Windows or some other image editor program of your choice. The bitmap should not be larger than 48 x 48 pixels in size.
2. Save the bitmap file in the same directory as the measurement routine. The bitmap filenames must use the following naming convention in order to be recognized by PC-DMIS:

   MARKST00.BMP for the first marked set

   MARKST01.BMP for the second marked set

   MARKST02.BMP for the third marked set

   …

   MARKST30.BMP for the thirty-first marked set

If you place the bitmap file MARKST00.BMP in the measurement routine directory when you create the first marked set, PC-DMIS shows the bitmap image contained in MARKST00.BMP. If a marked set does not have a bitmap file, PC-DMIS shows the default image. Also, it is not necessary to use consecutive bitmaps. In other words, you can use a bitmap for marked set 1 and marked set 5, but use the default graphics for marked sets 2, 3, and 4.

To Reposition Marked Sets

To reposition a marked set:

1. Select the **Edit | Markings | Show Marked Sets** option to open the Marked Sets window.
2. Hold down the Shift key and click the desired marked set.
3. Drag the marked set to the new location.
4. PC-DMIS updates the Marked Set window with the change.

To Execute Marked Sets

1. Select the **Edit | Markings | Show Marked Sets** option to open the Marked Sets window.
2. Double-click on the set you want to execute. PC-DMIS executes the marked set.
3. Close the Marked Sets window when finished.

To Lock Marked Sets

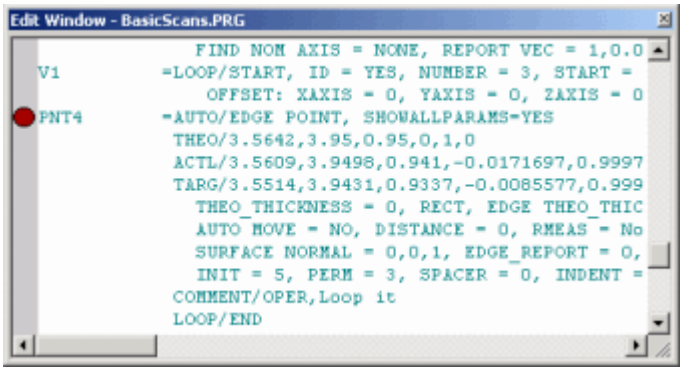
Once a marked set has been defined for a measurement routine, a lock can be established to prevent anyone from accidentally deleting or otherwise modifying the current configuration. For more information on this option see the "Lock Marked Sets" topic in the "Setting your Preferences" chapter.

To Delete Marked Sets

You can easily delete any previously created marked sets. To do this:

1. Select the **Edit | Markings | Show Marked Sets** option. The Marked Sets window appears.
2. Ensure the marked set you want to delete is active (press the Tab key to cycle through the sets, or simply click on the set once to select it).
3. Press the Delete key to open a confirmation box that you can use to confirm the deletion of the marked set.
4. Click the **Yes** button to delete the marked set. This also deletes the icon from the Marked Sets window.
5. When done, close the Marked Sets window.

Using Breakpoints



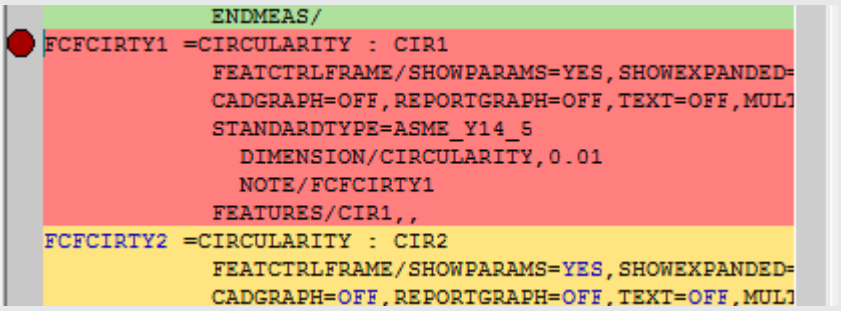
Example of the Edit window with a breakpoint (red icon).

The **Edit | Breakpoints** menu option provides a useful debugging tool to create, test, and run a measurement routine. Generally, measurement routines are executed sequentially, line by line. By placing a breakpoint at a particular line in the measurement routine, the execution pauses when it reaches the breakpoint. If the measurement routine makes use of variable expressions and flow control, you can examine these variables to help ensure that the measurement routine functions as desired.

Once PC-DMIS pauses for the breakpoint, you can use the **Step Next** button on the **Execution** dialog box to continue the measurement process one step at a time. This pauses the CMM after executing each step of any command that tells the CMM to move. While in Step Mode, you can insert additional hits into features, or you can insert new features between existing features or commands. Step Mode can also be simulated offline.

When you are ready to continue with the measurement routine's execution, click **Continue** on the **Execution** dialog box. You can use menu and keyboard commands to speed up the placement and removal of breakpoints. Breakpoints are saved with the measurement routine; they remain available the next time you open the measurement routine.

If your measurement routine uses breakpoints and the Edit window is in Command Mode, PC-DMIS continues to display the Edit window during execution but highlights the current command to execute in red.



Example showing the colors used in Command mode during execution with a breakpoint.

Breakpoint

The **Edit | Breakpoints | Breakpoint** menu option allows you to set or remove a breakpoint. The software sets or removes the breakpoint on the line in the Edit window at the current cursor position.

When you set a breakpoint, PC-DMIS displays a small circular red icon in the left margin on the line in the Edit window to indicate that a breakpoint is set. The icon disappears when you remove a breakpoint.

Insert Defaults

The **Edit | Breakpoints | Insert Defaults** menu option allows you to set breakpoints throughout the measurement routine at default locations. Default locations are defined as lines in the Edit window that contain commands which generate motion of the CMM or cause branching to occur as a result of a flow control command (such as IF THEN statements). See the "Branching by Using Flow Control" chapter for more information.

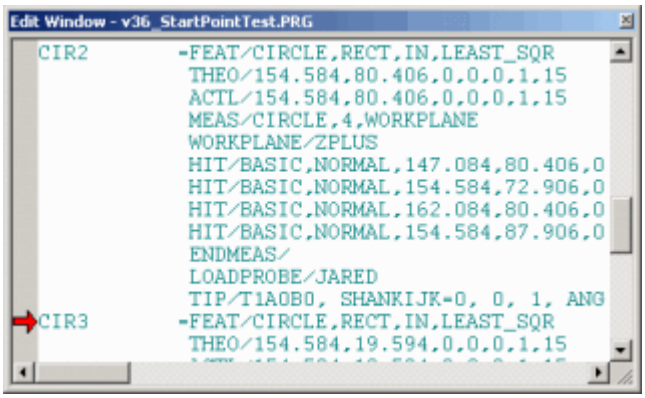
Remove Defaults

The **Edit | Breakpoints | Remove Defaults** menu option allows you to remove breakpoints throughout the measurement routine from default locations. PC-DMIS only removes those breakpoints that are set on default locations. Any breakpoints that are set in non-default locations remain in place. For details, see "Insert Defaults".

Remove All

The **Edit | Breakpoints | Remove All** menu option allows you to remove *all* breakpoints from a measurement routine.

Setting Start Points



Edit window with start point (red arrow)

You can only set start points when the Edit window is in Command Mode (see "Working in Command Mode" in the "Using the Edit Window" chapter).

When you insert a start point into the measurement routine and then select **File | Partial Execution | Execute From Start Points**, PC-DMIS begins the measurement routine execution from the first start point instead of starting the execution flow at the beginning of the measurement routine.

> Be aware that if the current tip for that location in the measurement routine does not match the current orientation of the probe head, PC-DMIS does not try to and go back to execute the tip command above it in order to change the tip orientation.

Start points are especially useful if you are working in Multiple Arm Mode and you need to set a different start point for each arm (see the "Using Multiple Arm Mode" chapter).

To insert a start point into your measurement routine:

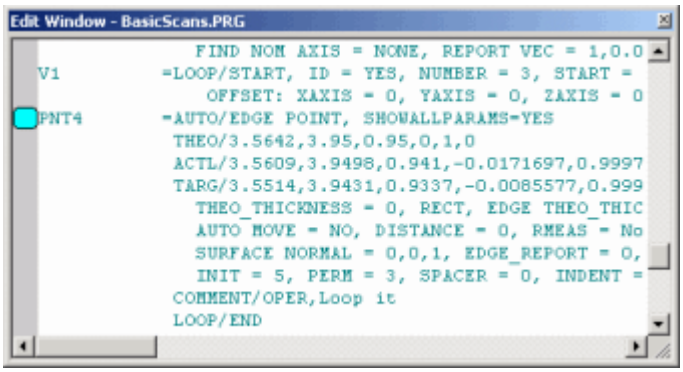
1. Click the location in the Edit window where you want the start point to appear
2. Then select the **Set Start Point** button from the **Edit window** toolbar. You can also right-click in the Command Mode and select the **Set Start Point** option from the shortcut menu.

For more information on the **Set Start Point** icon, see "Edit Window Toolbar" in the "Using Toolbars" chapter.

For information, on how to enable start points to always track the execution of a measurement routine, see "Start Points Always Track Execution" in the "Check Boxes for the General tab" chapter.

For information on start points for multiple-arm machines, see "Setting Start Points for Multiple Arms" in the "Using Multiple Arm Mode" chapter.

Using Bookmarks



Example of the Edit window with a bookmark (blue icon).

You can set bookmarks when the Edit window is in Command Mode (see "Working in Command Mode" in the "Using the Edit Window" chapter) or, if enabled, in DMIS Mode. Bookmarks mark frequently-accessed lines in a measurement routine. Once a bookmark is set, you can use menu or keyboard commands to move to it. You can remove a bookmark when you no longer need it. Bookmarks are saved between editing sessions and are therefore available the next time you open the measurement routine.

Be aware that bookmarks are assigned to line numbers, not commands. Thus, setting bookmarks in one mode and then switching to another mode may cause the bookmarks to show up on different commands even though they're on the same line numbers.

Bookmark

The **Edit | Bookmarks | Bookmark** menu option allows you to set or remove a bookmark. The software sets or removes a bookmark on the line in the Edit window at the current cursor position.

When you set a bookmark, PC-DMIS displays a small blue icon in the left margin on the line in the Edit window to indicate that a bookmark is set. The icon disappears when you remove a bookmark.

Next Bookmark

The **Edit | Bookmarks | Next Bookmark** menu option allows you to move to the next bookmark in the Edit window. If the search finds that no bookmarks exist below the current cursor position, the search resumes at the top of the Edit window.

Clear All Bookmarks

The **Edit | Bookmarks | Clear All Bookmarks** menu option allows you to remove all bookmarks from the Edit window.

Changing Fonts and Colors

You can edit the fonts and colors used in the Edit window by following instructions described in:

- "Customizing User Interface Fonts" topic of the "Navigating the User Interface" chapter.
- "Setting up the Edit Window" topic of the "Setting your Preferences" chapter.

Editing External Objects

PC-DMIS allows you to edit embedded external objects in Command Mode only. To do this, select the object in Command Mode, and double-click on the selected object.

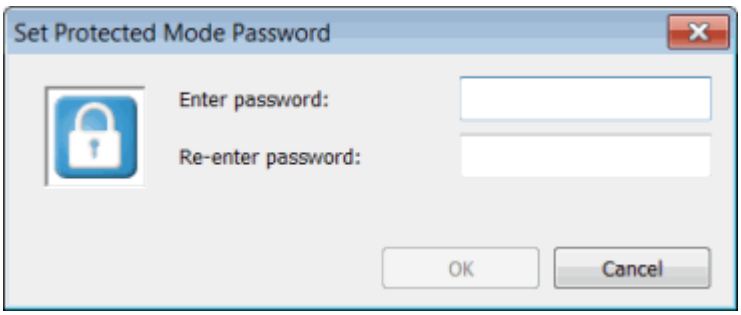
For information on creating and inserting external objects, see "Inserting External Objects" in the "Adding External Elements" chapter.

Using Protected Mode to Protect Measurement Routines from Edits

Enabling Protected Mode

To protect a measurement routine from unwanted edits, before you open a measurement routine, from the PC-DMIS Home screen, select the **Edit | Protected Mode** menu option to enable Protected Mode. This menu option is only available if no measurement routine is currently open.

When you select this option, PC-DMIS displays the **Set Protected Mode Password** dialog box where you can define a password to enable Protected Mode:



To define the password, type the exact string of characters into the two boxes and click **OK**. When you enable Protected Mode, a check mark appears next to the **Protected Mode** option in the **Edit** menu.

The Protected Mode password supersedes and overwrites any other password you have defined. For instance, if you have enabled password protection for the **Setup Options** dialog box (**Edit | Preferences |Setup**) or a .ipd (Inspection Plan Default) file, you must use the Protected Mode password instead.

As a PC-DMIS administrator, you can create marked sets to allow the execution of a partial measurement routine. Operators can execute marked sets while in Protected Mode.

Effects of Protected Mode

To protect your measurement routine, when you are in Protected Mode, you cannot do the following:

- Modify commands
- Add commands
- Modify some settings
- Calibrate probes

After you enable Protected Mode on a computer, you cannot modify any measurement routines for that version of PC-DMIS until Protected Mode is disabled.

With Protected Mode enabled, you can still do the following:

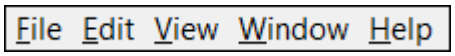
- View the Edit window and Report window
- Load and view different report templates
- Change the marking state of commands
- Execute the measurement routine

You cannot modify the measurement routine or reporting templates in any way.

Visual Indicators

When PC-DMIS is in Protected Mode, you will notice these visual indicators:

- Only a few menus are available, and most options within those menus are not editable. For example:

- Only the **Edit Window** toolbar, **Probe Mode** toolbar, and **Graphic View** toolbar are available.
- PC-DMIS displays the Edit window only in Summary Mode and disables both Command Mode and DMIS Mode.

How to Disable Protected Mode

To disable Protected Mode, follow these steps:

1. Close all open measurement routines.
2. Select the **Edit | Protected Mode** menu option to open the **Protected Mode Password** dialog box.
3. Type your password.
4. Click **OK**.

PC-DMIS then removes all editing restrictions.

If you are a member of the **PC-DMIS-User** or **PC-DMIS-Programmer** group, see the "Certify" section and "Edit Certified Measurement Routines" topic in the PC-DMIS Core documentation.

Updating Nominals from CAD

You can update the nominals of features in the Edit window based on an imported CAD file. This makes it possible to easily update your features' nominal (or theoretical) values to match the nominals of the CAD model. This is useful in these situations:

- You have already measured features without any CAD, but you now have CAD, and want to update your features' nominals.
- You want to update your features' nominals to match an updated CAD model.

To update your features' nominals from the CAD:

1. Select **File | Import** to import the CAD model you want to use. For more information on how to import a model, see "Importing a CAD file" in the "Using Advanced File Options" chapter.

2. Select **Operation | Features | Update Nominals from CAD** and choose one of these submenu items:

   All - Updates all the features' nominals in the measurement routine.

   Current - Updates only the selected feature's nominals.

The Status bar shows the operation results as it runs.

How it Works

This operation uses the **Find Noms** tolerance located on the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**). For information on the **Find Noms** tolerance, see "Other Edit Boxes for the General tab" topic.

This list provides information about how this operation updates the nominals for specific features:

- **Point** - PC-DMIS uses the XYZ and IJK of the point feature to draw an imaginary line that pierces the CAD model. It updates the nominal values to match the closest intersection to the original XYZ with the CAD model.

- **Edge Point, Angle Point, Corner Point** - PC-DMIS looks on the CAD model for the nearest feature of that type. The XYZ value must lie within the **Find Noms** tolerance of the original feature. It then updates the nominal values to match the CAD model.

- **Plane** - PC-DMIS projects the plane's XYZ onto the CAD model's closest planar surface. The XYZ must lie within the **Find Noms** tolerance of the original feature. It then updates the nominal values to match the CAD model.

- **Circle, Ellipse, Cylinder, Cone, Sphere** - PC-DMIS looks on the CAD model for the nearest feature of that type. The XYZ and diameter must each lie within the **Find Noms** tolerance of the original feature. Cylinder and Cone also have a length value that must also lie within the **Find Noms** tolerance. It then updates the nominal values to match the CAD model.

- **Square Slot, Notch Slot, Round Slot**, **Flush & Gap** - PC-DMIS looks on the CAD model for the nearest feature of that type. The XYZ as well as the length and width of the sides must each lie within the **Find Noms** tolerance of the original feature. It then updates the nominal values to match the CAD model.

- **Polygon** - PC-DMIS looks on the CAD model for the nearest feature of that type. The XYZ and the diameter must each lie within the **Find Noms** tolerance of the original feature. The feature on the CAD must have the same number of sides as the original feature. It then updates the nominal values to match the CAD model.

- **Line** - PC-DMIS projects the line's start and end points of the original feature onto the CAD model. Both points must lie within the **Find Noms** tolerance. It then updates the nominal values to match the CAD model.
- **All other feature types** - PC-DMIS locates all the features' hits, and projects each hit onto the CAD model to find the nearest XYZ point on the model. The XYZ of each projected point must lie within the **Find Noms** tolerance of the original hit.

Resetting Measured Values to Nominals

You can reset the measured values of features to match their respective nominal values. This is probably most useful when using portable devices. Measured features before the alignment are shown in relation to the device at the measured locations in the Graphic Display window. If those locations are from a previous run, and the part has moved, then those features will be drawn in an unexpected location on the screen. Resetting those measured values to the nominals will redraw the features so they rest appropriately atop the CAD model.

To reset features' measured values to match the nominal values:

1. Click **Operations | Features | Reset Measured Values to Nominals**.
2. You can update all nominals by clicking the **All** menu item. To update just the currently selected nominal feature, click the **Current** menu item.

Estimate Execution Time

The topics in this chapter include:

About Estimate Execution Time

Workflow

Calibrating Execution Time Parameters

Calibrating Execution Time Parameters on Your Machine

    Performing Execution Time Calibration

Estimating Execution Time

About Estimate Execution Time

This function estimates the execution time of a measurement routine in Offline Mode. You can use a computer connected to a CMM or an offline computer used to create measurement routines to estimate the execution time.

Execution time is an estimation; actual execution time may vary. Several factors affect the actual execution time. Among these are:

- Execution time depends on the jog box speed setting. It should be set to full speed and should not be reduced.
- Execution time depends on the type of commands used in the measurement routine. Some commands, like unknown contour scans, do not have a stored path. In such cases, the machine navigates the shape of the part, making the time for such scans incalculable.
- Execution time depends on the communication time with the controller, which varies depending on the computer's available RAM.

To estimate execution time accurately on an offline computer, ensure that global parameters like move speed, touch speed, and scan speed match those set on the machine for which the execution time is being calculated or define these parameters in the measurement routine.

The Estimate Execution Time function does not support horizontal arm machines or machines with rotary tables.

Workflow

To estimate execution time:

1. Calibrate the Execution Time parameters for your machine or use a parameter file from another machine to estimate the execution time on that machine.
2. Create a measurement routine.
3. Use the Estimate Execution Time command to estimate the execution time.

Calibrating Execution Time Parameters

Every machine is different. The machine’s kinematics, controller parameters, and hardware determine the execution time of a measurement routine. These parameters contribute the most to the execution time:

- **TIP command** - This command rotates the wrist mounted on a machine.
- **Time for measuring HIT points** - The time required to measure HIT points depends on the type of probe used. Scanning probes take longer to record a hit point than touch-type probes.

- **Tool Change time** - This depends on the complexity of the tool change. It can be as simple as changing the probe-holding disk or as complex as changing both the module and the probe-holding disk.
- **Machine Acceleration** - Modern machines, such as Hexagon's, move with very high acceleration. This allows them to achieve the defined move speed over a short duration.

During calibration, these parameters are measured and recorded. They are used to estimate execution time. You can get the parameters for calculating execution time by:

1. Calibrating your machine
2. Using a packaged generic parameters file that best matches your measurement machine.

Calibrating Execution Time Parameters on Your Machine

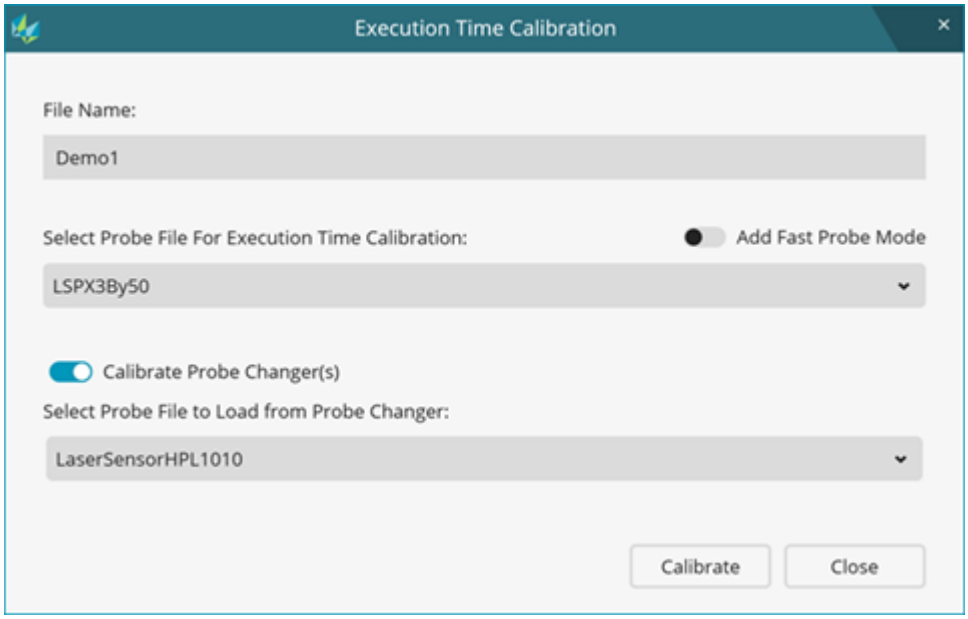
To calibrate execution time parameters on your machine, follow these steps:

1. Create and calibrate the probe files. Ensure to add these angles to the probe file:
    - TIP/T1A0B0
    - TIP/T1A90B0
    - TIP/T1A90B90
    - TIP/T1A90B-90
    - TIP/T1A90B-180
2. The probe file used to exchange probes from the tool changer must have at least one calibrated probe tip.
3. Calibrate probe tips.
4. For a fixed-head machine, create a star probe and calibrate it.
5. Mount the calibration sphere on your measuring machine. Ensure that there is enough space around it to measure points in the Z, X, and -Y directions. If you select tool changer calibration, ensure the path to the probe changer is collision free.
6. If your measuring machine is equipped with a tool changer, ensure that the probe file for calibration and at least one additional probe file is assigned to the appropriate slot on the tool changer. The calibration process exchanges tips from the tool changer multiple times.

Performing Execution Time Calibration

To perform execution time calibration, follow these steps:

1. From the Home page, select the menu option **Operation | Execution Time Calibration**. PC-DMIS displays the **Execution Time Calibration** dialog box.



2. Complete the options:

    - **File Name**: Type a name for the file. The name should help you identify the machine, wrist, and probe.
    - **Select Probe File For Execution Time Calibration**: Select the probe file for calibration. We recommend using a 3 to 5 mm diameter probe tip. Ensure that you create and calibrate the tip angles required for calibration.
    - **Add Fast Probe Mode**: Turn this on if your machine's controller supports fast probes with scanning sensors and you use them in your measurement routines.
    - **Calibrate Probe Changer(s)**: If you turn this on, you must define the second probe.
    - **Select Probe File to Load from Probe Changer**: Select a probe file other than the one selected above for calibration. This option is enabled only if you turn on the **Calibrate Probe Changer(s)** option.

3. Do one of the following:

    - To start the calibration, click **Calibrate**.
    - To cancel the calibration and return to the Home page, click **Close**.

Once you click the **Calibrate** button, PC-DMIS generates a new measurement routine. Its execution starts immediately after generation. It is important to generate and start execution this way; otherwise, the required parameters are not created, and time estimation is not possible.

Follow the prompts provided by the calibration routine. Complete the menu to enter the nominal diameter of the calibration sphere used. The calibration routine asks you to take one point on top of the calibration sphere. After that, it determines the sphere position and measures points multiple times. At the end of the time calibration process, PC-DMIS generates a CSV file and saves it in this folder:

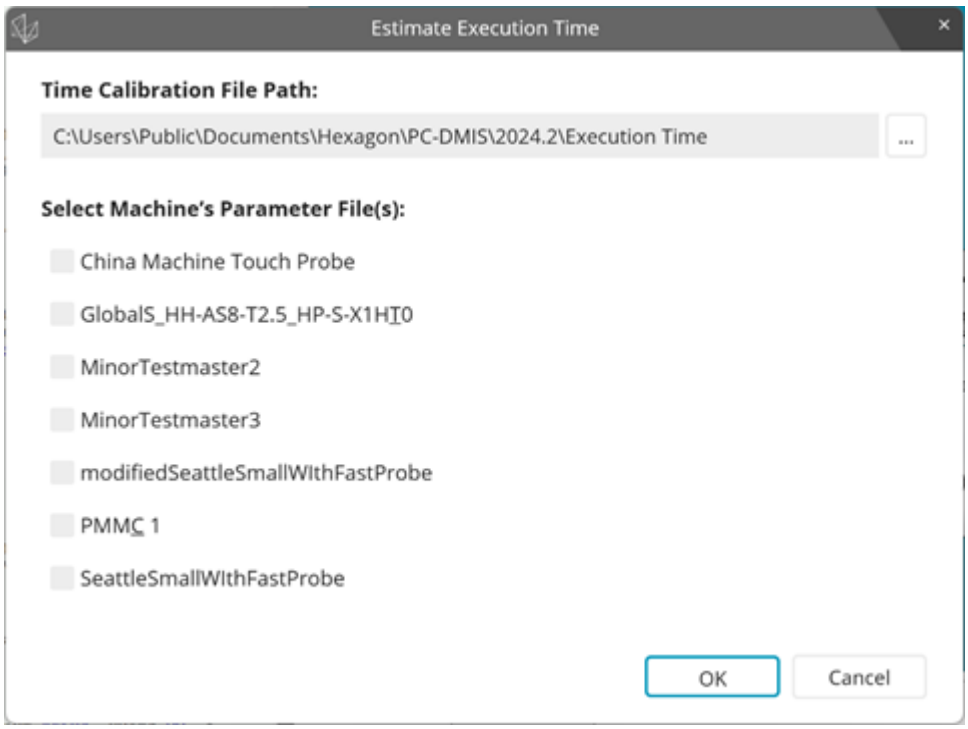
C:\Users\Public\Documents\Hexagon\PC-DMIS\Variable "currentversion" is not defined\Execution Time

You can copy and store the calibration file in any desired location. If your organization owns multiple machines, you can copy it into a folder on a network drive, making it accessible to multiple users.

Estimating Execution Time

You can use the Estimate Execution Time functionality to approximate the time PC-DMIS takes to execute your measurement routine. To do this, follow these steps:

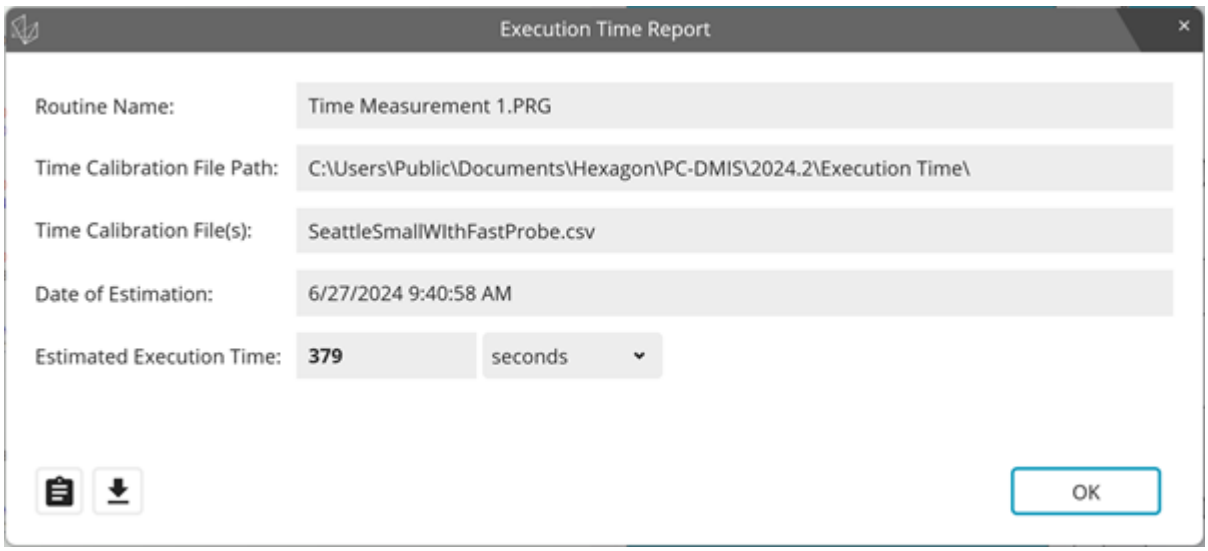
1. Open the measurement routine.
2. Select the menu option **Operation | Estimate Execution Time** to open the **Estimate Execution Time** dialog box.



Complete the options:

- **Time Calibration File Path** - Use this box to type the path for the calibration file or click the **Browser** (...) button and navigate to the location of the calibration file.
- **Select Machine's Parameter File(s)** - Select the calibration file(s) for your machine.

3. To start the calculation, click **OK**.
4. At the end of the execution time estimation, PC-DMIS displays the **Execution Time Report** dialog box.

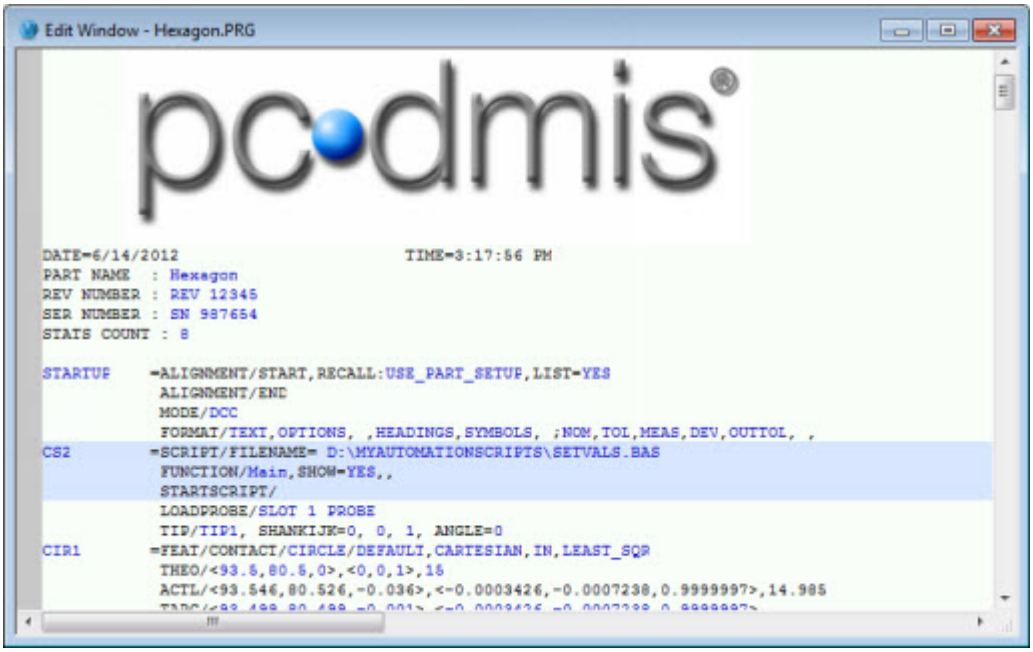


The dialog box displays the routine name, time calibration file path, time calibration file(s) used, date of calculation, and estimated execution time. You can select the estimated execution time in seconds, mm:ss, or hh:mm:ss.

- **Copy** - Copies the report to the clipboard.
- **Download** - Downloads the report in PDF format.
- **OK** - Closes the **Execution Time Report** dialog box.

Using the Edit Window

Using the Edit Window: Introduction



Example of the Edit window in Command mode.

One of the main tools that you can use to edit a measurement routine is the Edit window. The Edit window houses all of the commands for the measurement routine. You can perform editing operations such as cutting, copying, pasting, and modifying existing text and commands. You can also use the Edit window to add new commands, execute existing commands, display a command's information, and debug code.

To access the Edit window, select the **View | Edit Window** menu option.

This chapter covers the Edit window and how to edit measurement routines in general in these topics:

- Understanding Core Concepts
- Working in Summary Mode
- Working in Command Mode
- Working in DMIS Mode
- Working with User-Defined Groups

Understanding Core Concepts

The Edit window allows you to easily access the current measurement routine. Changes can be made to the measurement routine using only the Edit window, the dialog boxes, or a combination of the Edit window with the applicable dialog boxes. The sections below provide core concepts that can aid you in learning to use the Edit window.

Navigating the Edit Window

The Edit window houses all the commands that make up your measurement routine.

To change the display of the Edit window, you can switch between different modes. PC-DMIS enables the two most common modes by default. These are Command Mode and Summary Mode. You can enable a third mode, called DMIS Mode, from the **Setup Options** dialog box (**Edit | Preferences | Setup**). To do this, select the **Use DMIS button in Edit Window** check box.

- Summary Mode - This mode provides a visual summary of your measurement routine.
- Command Mode - This mode shows a detailed code view of each command.
- DMIS Mode - This mode shows a view similar to Command Mode, but the code language is written in the DMIS programming language. Also, you don't have all the editing functionality that you have in Command Mode.

You can click the appropriate icon on the **Edit Window** toolbar to switch between these different modes. The **Edit Window** toolbar also lets you perform other operations on your measurement routine. For information on this toolbar, see the "Edit Window Toolbar" topic in the "Using Toolbars" chapter.

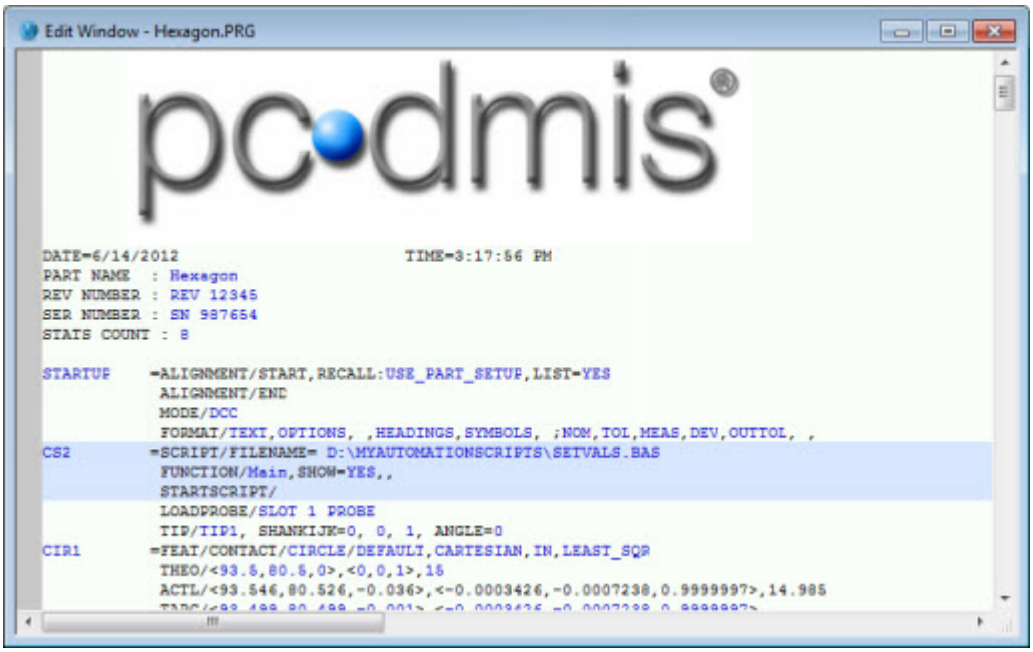
In each mode, you can work with existing commands and add new commands, although the capabilities of each mode and the methods of working with commands differ. Commands can have both editable and non-editable fields. Editable fields are fields that take a value. Some editable fields are called *toggle fields* and allow you to switch between a predefined set of acceptable values.

- *If you're in Command Mode or DMIS Mode*, press Tab to move the cursor to the next editable field. You know that you're on an editable field when the value is highlighted in the highlight color (the default is blue).
- *If you're in Summary Mode*, PC-DMIS provides you with the **Select**, **Add**, **Remove**, **Copy**, **Cut**, **Paste**, **Mark**, **Unmark**, and **Edit** commands. For more information, see "Editing a Measurement Routine from Summary Mode".

Moving to a Feature Using the Graphic Display Window

To move the cursor to a specific feature's location in the Edit window, click on that feature in the Graphic Display window while you hold down the Ctrl key. PC-DMIS displays a message for your confirmation to move the insertion point.

Understanding the Default Edit Window Colors and Formatting



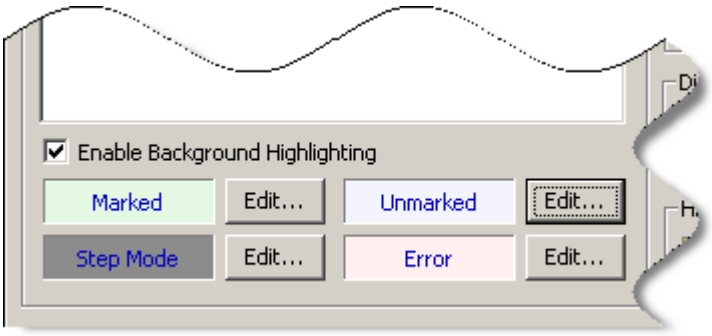
Edit window in Command Mode

PC-DMIS uses different background colors, command text colors, and text characters to offset information in Command Mode and DMIS Modes. This coloring and text formatting may seem somewhat confusing at first, especially if you are used to very old versions of PC-DMIS that only used a single color. If you don't like the default color scheme, you can configure your coloring scheme similar to that of old versions or come up with your own. For information on how to do this, see "Defining Edit Window Colors" in the "Setting Your Preferences" chapter.

The colors and formatting are valuable tools that can help increase your productivity. The following describes these tools.

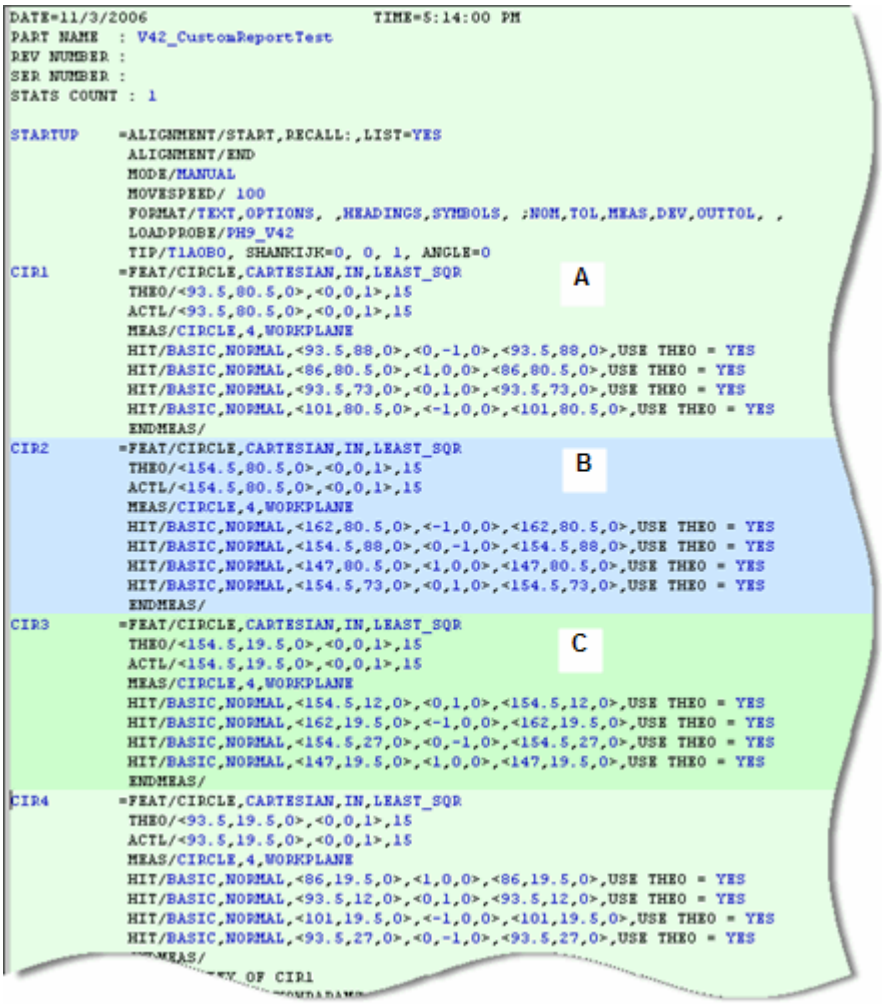
Background Colors

Different background colors help you know what commands are marked, unmarked, have errors, are active, or are being stepped through. You can find the definitions for these background colors in the **Color Editor** dialog box (**Edit | Preferences | Edit Window Colors**) for the Edit window.



Edit window background color information in the Edit window's Color Editor

Consider this example screenshot:



Edit window example showing the coloring scheme
***A** - A light-green background means the command has been marked for execution.*
***B** - The light-blue background means the command has not yet been marked for execution.*
***C** - A slightly darker background color (either darker green for marked commands or darker blue for unmarked commands) shows the current active command.*

Command Text Colors

Black text - Any non-editable command text. You cannot change this text.

Blue text - Any editable command text.

Red text - For non-dimension commands, this indicates an error in the command text. The command is skipped during measurement routine execution. Errors come from unsupported commands, unsupported probe types, and syntax errors (usually while working with expression or scripting commands). For dimension commands, red text indicates that the dimension is out of tolerance.

For details on how PC-DMIS reports dimensions that use skipped features during execution, see the "Reporting a Dimension Using a Feature Skipped During Execution" topic in this documentation.

If your measurement routine contains unsupported commands (commands in red text), check it thoroughly before you execute it in Online Mode. Since unsupported commands are skipped during measurement routine execution, in some cases this could cause a probe collision if you aren't careful.

Highlighted Text Shows the Active Command or Group

When you move your mouse over a command block, or command blocks that are grouped together, the entire command block, or the entire group, is highlighted with a slightly darker background color (see item C in the example above). This helps you immediately see the extents of an entire command block. Doing this also makes the command "active". This means you don't need to click on the command to operate on it. For example, you can hover your mouse over a command and press F9. The associated dialog box appears if one exists. To delete the command, you don't need to select the entire command, just hover your mouse over it until it becomes active, and press Delete. When editing large measurement routines, shaving off a mouse click here and there can increase your productivity.

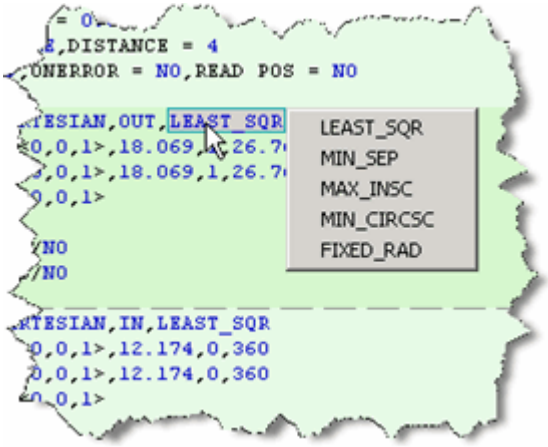
"<" and ">" Characters Provide Enhanced Readability

The "<" and ">" characters group like values together, thereby more easily delineating the different fields for various X, Y, Z and I, J, K values. These characters show where these pairs begin and end. This enhances readability and makes it much easier to locate needed values.

```
PLN2        =FEAT/CONTACT/PLANE,CARTESIAN,
              THEO/<186.865,30.085,-1.705>,
              ACTL/<186.865,30.085,-1.705>,
              TARG/<186.865,30.085,-1.705>,
```

Popup Menu Provides an Additional Toggle Field Selection

While you can cycle through a list of available commands on a toggle field, you can also hover your mouse over the darker blue text for a moment, and the background color takes on a button-like quality. Click on the button to show a drop-down list from which you can select the desired value for that toggle field.



Easier Filename Fields

Certain fields take filename values. If the file moves to a new location, there's an easier way to update it than arduously retyping a long directory pathway. Instead, hover your mouse over the field; the filename turns into a button. Click the button to display a **File Open** dialog box, which allows you to locate and update the file's location.

Inserting Commands

In many cases, commands are automatically inserted when you take measurements or when you use dialog boxes to enter information. However, you can also add commands directly from within the Edit window. Depending on what mode you're in, PC-DMIS gives you different options:

- *If you're in Command Mode*, you can type the first few letters of the command and press Tab or Enter to insert new commands.
- *If you're in Summary Mode*, you can right-click and select **Add Command** from the shortcut menu to insert new commands. PC-DMIS displays a list of the commands that you can add.
- *If you're in DMIS Mode*, you can press Enter on a line to insert new commands. A list appears from which you can select the appropriate DMIS command to add.

Indenting Commands

To improve the Edit window's readability, PC-DMIS automatically performs an indent inside of supported code blocks, loops, and flow control pairs. This occurs whenever you insert these types of commands into the Edit window.

Summary Mode uses a fixed indent value. Command Mode and DMIS Mode use a two-character default for the indent. You can modify the number of spaces with the `EWGroupIndent` entry in the PC-DMIS Settings Editor.

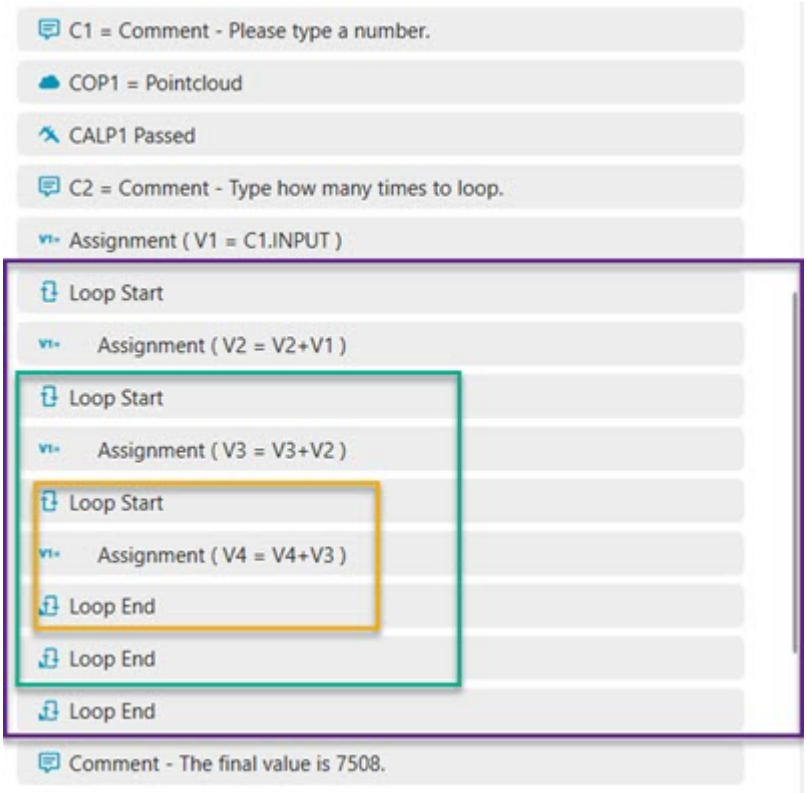
This shows a list of commands that support indenting:

Supported Commands

- Alignment (ALIGNMENT/START and ALIGNMENT/END)
- Measure (MEAS/<FEATURE> and ENDMEAS for all measured features)
- GROUP and ENDGROUP
- LOOP/START and LOOP/END
- IF and END_IF
- ELSE_IF and END_ELSEIF
- ELSE and END_ELSE
- While and END_WHILE
- DO and UNTIL
- SELECT and END_Select
- CASE and END_CASE
- DEFAULT_CASE and END_DEFAULT_CASE

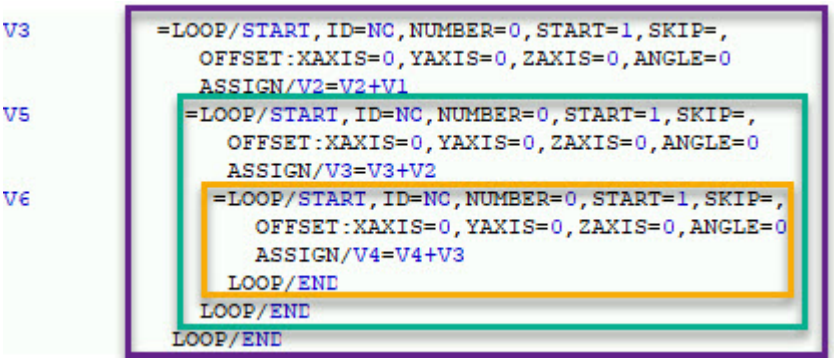
The following examples show how nested indents appear in each Edit window mode (Summary Mode, Command Mode, and DMIS Mode):

Summary Mode Example



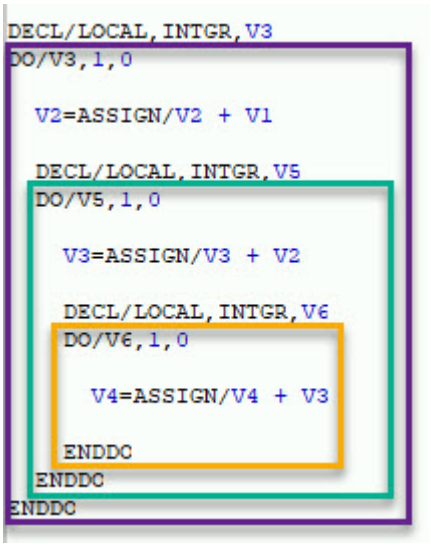
Example of nested loops in Summary mode.

Command Mode Example



Example of nested loops in Command mode.

Dmis Mode Example



Example of nested loops in DMIS mode.

> The Edit window can display a maximum of 256 characters on a single line, including white space. This means many levels of nested commands can push the innermost commands so far to the right that the Edit window cannot display their text.

Editing Values and IDs

You can use the Edit window to change a command's settings.

If you're in Command Mode or DMIS Mode, press the Tab key to go to the editable field you want to change. Either type in a new value or press F7 or F8 to cycle forward or backward through a list of available values. You can also move your pointer over a toggle field and after a moment it changes to a gray background. You can then click on that to select from a menu of available values.

If you edit a feature or dimension's ID using the Edit window, PC-DMIS asks if you want to make this ID the default for that same feature or dimension type. If you click **Yes**, features

or dimensions of that same type, when created later, will use the new ID as the default. If you click **No**, then future IDs continue to use the current default value.

Selecting Commands

You can select most commands in any mode in PC-DMIS.

- In Summary Mode, click on a collapsed group command to select the entire command block. PC-DMIS also selects all of the data items associated with the commands present in the group. For details see "Selecting Commands in Summary Mode".
- In Command Mode or DMIS Mode, click the start of the command, hold down the mouse, and then drag it until you highlight the entire command.

The **Edit | Select All** menu option selects the entire window's contents for editing purposes.

As you select feature commands, PC-DMIS highlights those features in the Graphic Display window in a light blue color. Also, in Command Mode, this happens if you click anywhere in a feature command. To change the highlight color, see "Highlight" under "Changing Screen Colors" in the "Editing the CAD Display" chapter. Auto Plane features also highlight the entire surface.

Repositioning Commands

You can reposition many commands in PC-DMIS*. If you're in Summary Mode, Command Mode, or DMIS Mode*, do the following to reposition a command:

1. Select the entire command block.
2. Select the **Edit | Cut** menu option to remove the command from its current location.
3. Move the cursor in the Edit window to the new location and select the **Edit | Paste** menu option to place the command in its new location.

You can also use the **Edit | Copy** menu option to make a copy of a command and store it in the Clipboard until you decide to paste the copy into another location.

For information on these and other standard edit commands, see "Using Standard Edit Commands" in the "Editing a Measurement Routine" chapter.

Deleting Commands

To delete a command from the Edit window, select the command and press **Delete**.

You can restore any deleted command if you immediately click the **Undo** icon on the **Edit Window** toolbar or if you press Alt + Backspace.

Accessing Dialog Boxes

When you edit features or commands, you may want to access dialog boxes associated with a particular feature or command. To do this, when you are in one of the editing modes, place the cursor on the feature or command block and then press F9. The software opens the dialog box associated with the feature. You can then make any changes in the dialog box itself. After you click **OK** or **Apply**, the software automatically updates the Edit window with your changes.

On some commands, you can press F9 to access the dialog box only if PC-DMIS is in Command Mode.

Modifying the Edit Window's Headers and Footers

There are three data files that are used to format the header / footer text in the Edit window. These files are LOGO.DAT, HEADER.DAT, and ELOGO.DAT. These files are located in your PC-DMIS installation directory.

You can edit the Edit window's header or footer by making and saving changes to these text files using a standard text editor (such as Notepad).

To modify the header or footer of a generated report, not the Edit window's header, you have these options that the "Reporting Measurement Results" chapter covers:

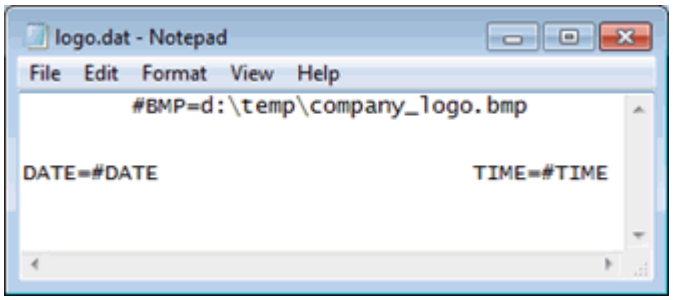
1. Modify templates to control what gets displayed in the Report window. (See "Modifying the Report's Header".)
2. Use LOGO.DAT, HEADER.DAT, and ELOGO.DAT files to a limited extent. (See "Using .DAT Files in Generated Reports".)

The Edit Window's Header

To define the header layout in your Edit window, you can edit the LOGO.DAT and HEADER.DAT files.

LOGO.DAT – This file defines the header for the very first page of the Edit window. You can define a bitmap with your company's logo, specific date, and time formats.

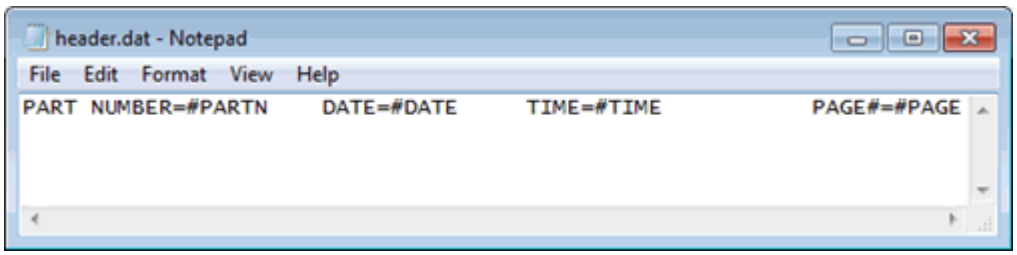
Example of the Edit window's header formatting.



Example of a Logo.dat file in Notepad.

That PART NAME, REV NUMBER, SER NUMBER, and STATS COUNT are static fields and cannot be changed in the LOGO.DAT file.

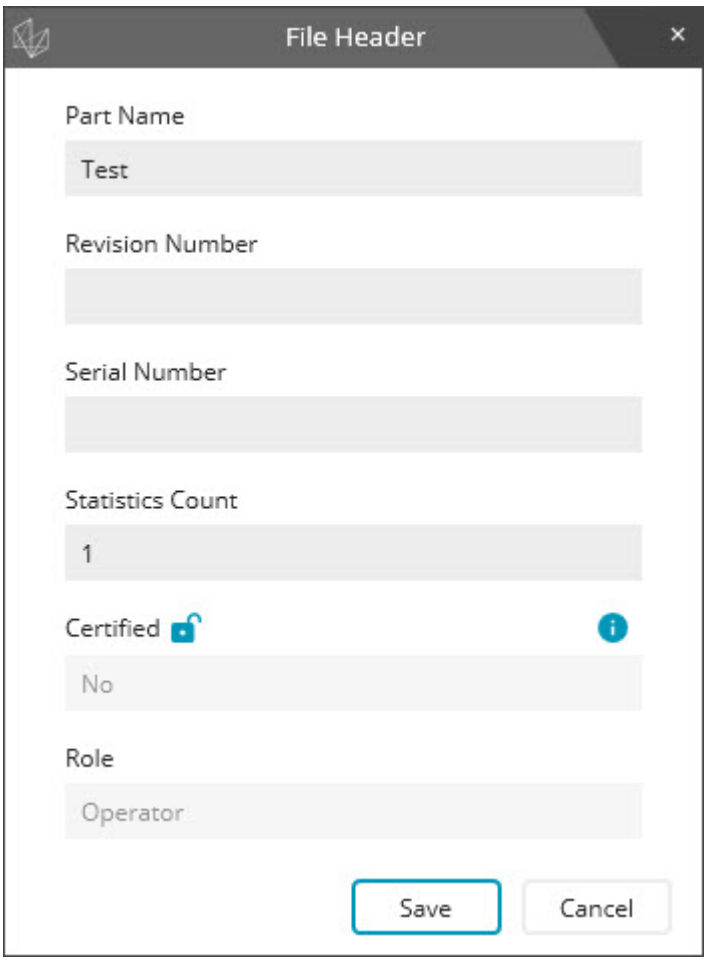
HEADER.DAT – This file is used to format page headers for all other pages.



Example of a Header.dat file in Notepad.

File Header dialog box

To edit header file values, you can change them in the Edit window or press F9 to open the **File Header** dialog box.



File Header Dialog Box

Part Name - This box displays the part name.

Revision Number - This box displays the revision number.

Serial Number - This box displays the revision number.

Statistics Count - This box displays the number of transactions or runs.

Certified - This box informs you if the measurement routine is certified or not. You cannot edit this value. For information on how to certify a measurement routine, see "Certify".

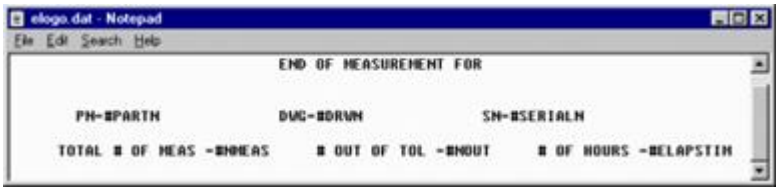
Role - This box displays the role of the author who opened the measurement routine. You cannot edit this value.

The Edit Window's Footer

ELOGO.DAT is used to format a footer for the last page only of your Edit window.

Example of the Edit window's footer formatting.

The corresponding ELOGO.DAT file is displayed below:



Example of an Elogo.dat file in Notepad.

Edit Window Header and Footer Formatting Keywords

Following is a list of the available formatting keywords and their functions. You can insert these keywords into the .dat files to have them displayed in a header or footer in the Edit window.

The keywords *are* case sensitive.

#DATE
Inserts the current date.

#TIME
Inserts the current time.

#PAGE
Inserts the current page number. This is ideal for use in the HEADER.DAT file.

#TRn
Inserts both the name and the value of tracefield n, where n is the tracefield number.

#PARTN
Inserts the measurement routine name.

#DRWN
Inserts the revision number.

#SERIALN
Inserts the serial number.

#SEQUENCE
Inserts the sequence number.

#SHRINK
Inserts the scale factor.

#NMEAS
Inserts the total number of dimensions.

#NOUT
Inserts the total number of dimensions that are out of tolerance.

#ELAPSTIM
Inserts the time elapsed between start and end of execution. This is ideal for use in the FOOTER.DAT file.

#BMP=<path>
Inserts a bitmap where <path> represents the full path pointing to the bitmap file (for example, D:\Files\Bmp\Pcdhead.bmp).

Using Expressions and Tracefields to Customize Edit Window Headers and Footers

You'll notice that the keywords themselves aren't customizable. For example the `#DATE` keyword in the logo.dat file only gives you one way to display the date, in the mm-dd-yyyy format. For example, in this format, May 5, 2018 is written as 5-5-2018 in your Edit window's header or footer.

For more information on the elogo.dat, logo.dat, and header.dat files, and where to find them, see the "Understanding Data Files" topic in this chapter.

The following example shows how you can use expressions and tracefields to change the format of your date on the Edit window.

1. Type the following commands somewhere into your measurement routine:

   ```
   ASSIGN/V1=SYSTEMDATE("MMM dd, yyyy")
   ```

   This gives V1 the string value of your current date (for example, "May 05, 2018").

   ```
   TRACEFIELD/DISPLAY,LIMIT=15;DATE:V1
   ```

   This assigns V1 to the TRACEFIELD.

2. Assuming this is the first tracefield in your measurement routine, open up your logo.dat file inside a text editor and modify the DATE= field so that it looks like this:

   ```
   DATE=#TR1
   ```

   Notice that the DATE field now references the first tracefield by using #TR1.

3. Save and close your logo.dat file.

4. Access the Edit window. Assuming your Edit window is enabled to show headers and footers, the header now uses the formatted system date that was stored in the V1 variable and subsequently in the first tracefield.

You can use tracefields and expressions to give you greater control over what you see in the Edit window.

For information on tracefields, see the "Using Tracefields" topic in the "Tracking Statistical Data" chapter.

For information on expressions, see the "Using Expressions and Variables" chapter.

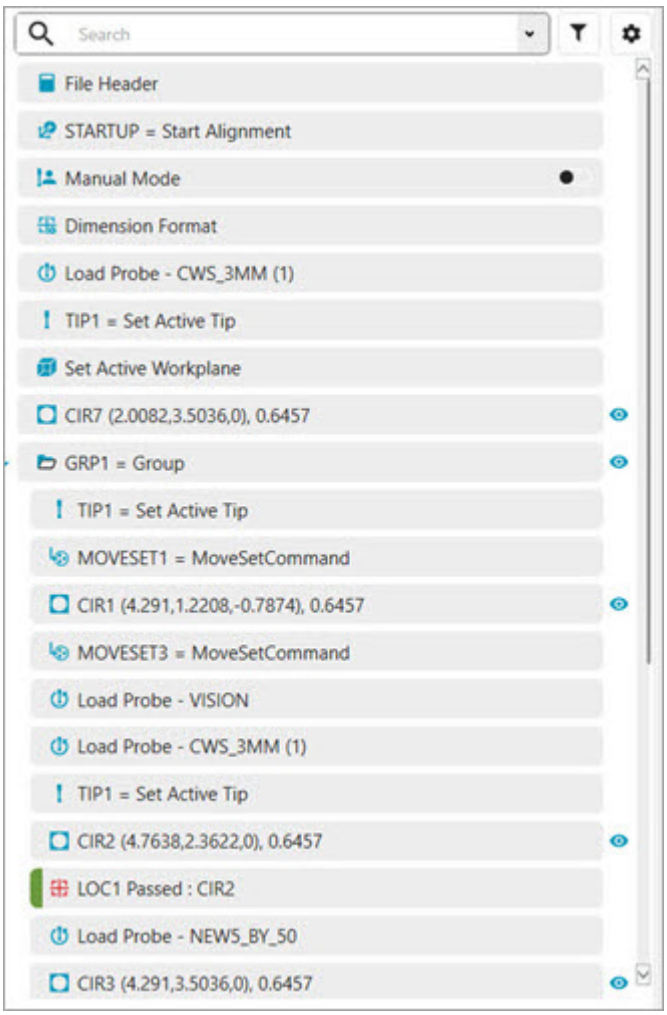
For information on how to modify your report's headers or footers with .DAT files, see "Using .DAT Files in Generated Reports".

For information on how to use templates and rules to modify your report, see "Modifying the Report's Header".

Working in Summary Mode

Summary Mode displays a summary of the measurement routine, as well as measurement routine commands in different levels of detail. Through the interface, you can also re-order and edit commands. You can use the **Search and Filter** toolbar to find specific features, dimensions, commands, or any other data.

To place the Edit window in Summary Mode, select **View | Summary Mode** from the menu, or from the **Edit Window** toolbar, click the **Summary Mode** icon .

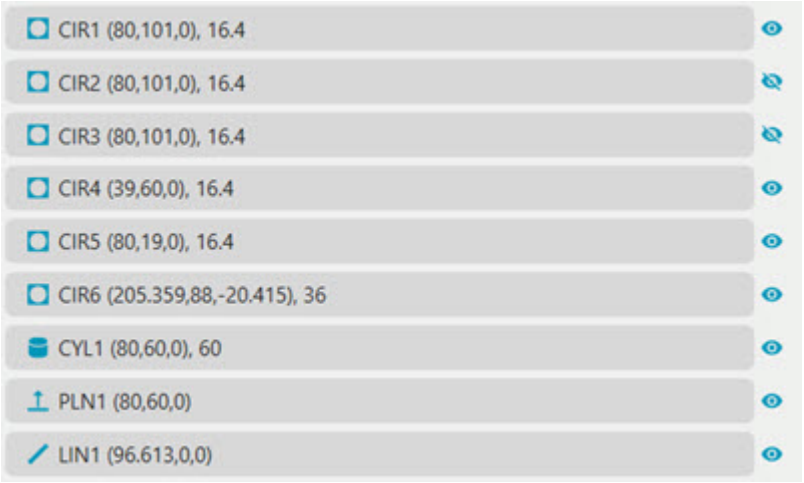


Example showing PC-DMIS Summary mode.

Summary Mode also appears when you create or edit a custom report. For more information on custom reports, see "Creating Custom Reports" in the "Reporting Measurement Results" chapter.

Customizing the Main Display to Show XYZ Information

You can customize the main display of the Summary Mode to show your features' XYZ information in the header line. To do this, access the PC-DMIS Settings Editor, expand the **USER_Option** section, and set the `ShowAdvancedSummaryText` entry to 1. PC-DMIS then displays the commands similar to the following:



Example showing PC-DMIS in Summary mode with advanced summary text.

For information on the Settings Editor, see the "Modifying Setting Entries" topic.

Layout

PC-DMIS's Summary Mode layout is in the form of pieces of data. At the top level, PC-DMIS displays command objects.

- To expand a group to view its data items, click the expand button ( ) to the left of the group command.
- To collapse a group, click the collapse button ( ).

You can also press the Right Arrow and Left Arrow keys to expand and collapse a group, respectively.

Summary Mode Commands

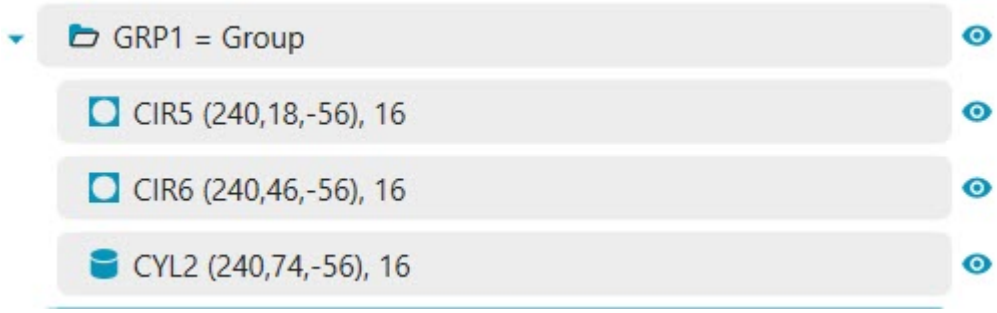
Summary Mode with commands

PC-DMIS displays the measurement routine commands at the top level. Also:

- The search and filter toolbar to find specific features, dimensions, commands, or any other data. For more information, see "Search and Filter Toolbar".
- Each command has a unique icon associated with it to help you quickly identify needed commands.

- If the command has an ID, PC-DMIS displays the ID before the command's description.
- The Peek window displays a command's complete information.
- The switch lets you change between DCC Mode and Manual Mode.
- The icon shows or hides a single feature. If you choose to hide a feature, the feature still exists in the measurement routine but is merely hidden from view in the Graphic Display window. For items that you can show or hide, see "Showing and Hiding Graphics".
- The **Settings** icon ( ) lets you show or hide the Peek window and access the **File Header** dialog box.

Summary Mode Groups



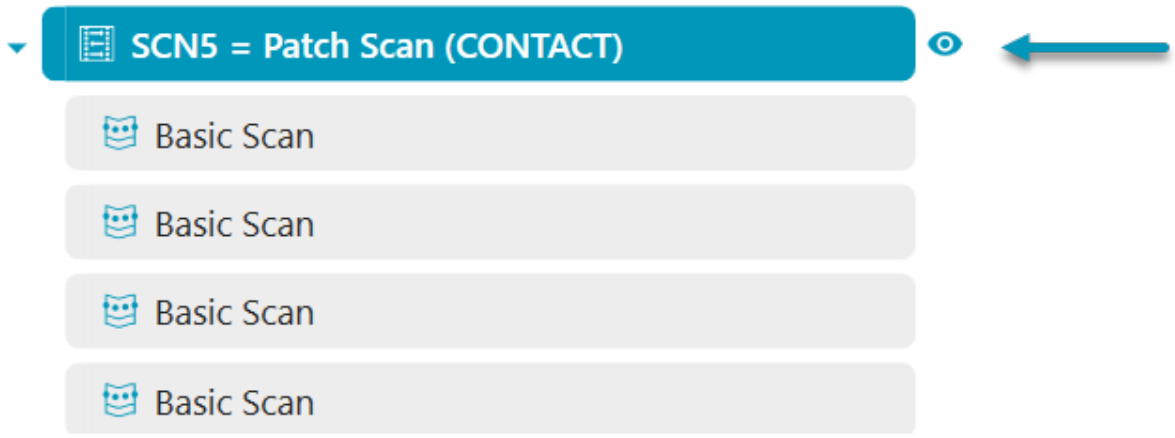
Summary Mode with groups (folder icons)

Groups appear with a folder icon in the Edit window. They are composed of a command's similar data items. If a group contains features that are shown and hidden, PC-DMIS displays this icon .

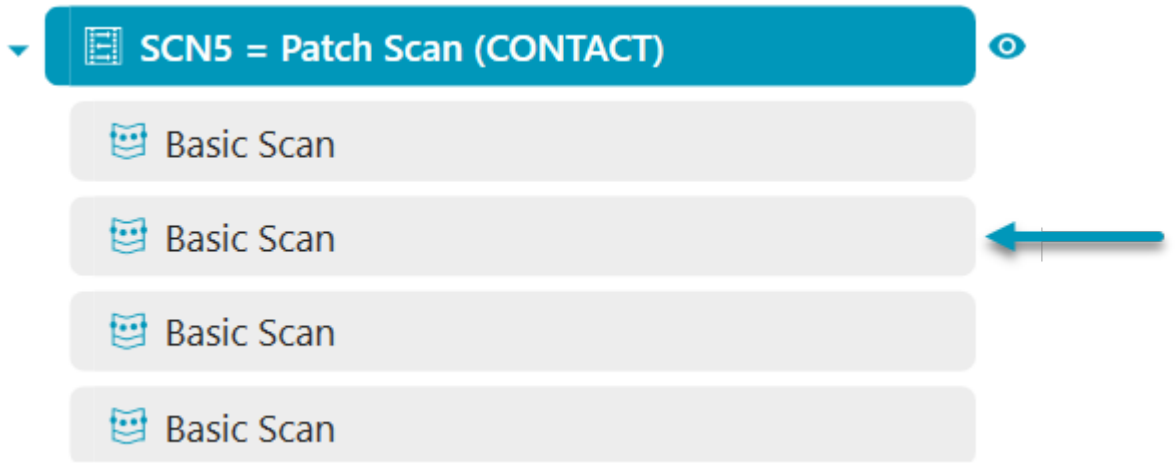
Scan Command

A scan command is composed of one or more Basic scans.

- To set the ClearanceCube motion for the start and end of the main scan, in the Edit window, place your cursor on the main scan's header line. For example, at this location:



- To set the ClearanceCube motion for a Basic scan, in the Edit window, place your cursor at the location of the desired Basic scan. For example, at this location:



From the Feature Shortcut Menu:

- If you select **Hide All Features**, **Hide Feature**, and **Show All Features**, the icon also turns on or off in the Summary Mode Edit window.
- If you select **Hide ID** and **Show ID**, the icon turns on or off only for the Point Infos, Dimension Infos, Datums, Pointclouds, Mesh, and Basic scan commands but not for the features.

Editing a Measurement Routine from Summary Mode

PC-DMIS gives you the ability to edit the measurement routine while in Summary Mode. With the provided interface, you can perform select, add, remove, copy, cut, paste, mark,

unmark, and edit commands. You can also show and hide features in the Graphic Display window.

Selecting Commands in Summary Mode

To select a command, either left-click with the mouse or use the Up and Down Arrow keys to navigate to it.

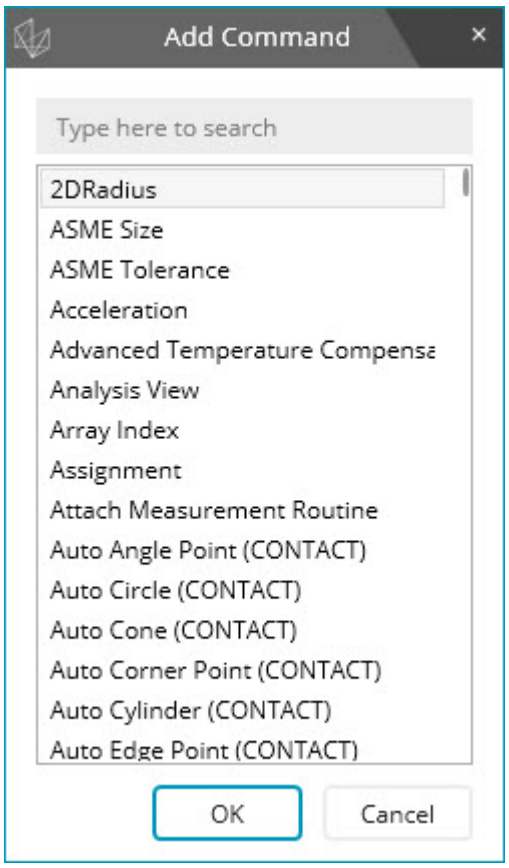
- To select multiple commands, hold down the Ctrl key while you left-click.
- To select a group of items at once, select the first item in the block of objects, hold down the Shift key, and select the last item in the block.

PC-DMIS highlights all of the selected items.

Adding Commands

In Summary Mode, you can use PC-DMIS's standard menu options or a special shortcut menu to add commands to the measurement routine.

1. To access the shortcut menu, right-click on a command item. For more information on the items that are available, see "Summary Mode Command Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" appendix.
2. From the shortcut menu, select **Add Command** to open the **Add Command** dialog box.



Add Command Dialog Box

3. To add a command, select the command and click **OK**. PC-DMIS inserts the command into the Edit window. If you want to close the **Add Command** dialog box, click **Cancel** or press Esc on your keyboard.

PC-DMIS generally inserts the new command after the currently-selected item in the expandable/collapsible list.

Renaming Feature IDs

The **Rename Feature IDs** option is only available when you select multiple features in Summary Mode.

In Summary Mode, you can use PC-DMIS's standard menu options or a special shortcut menu to rename feature IDs in measurement routines. This option lets you rename multiple feature IDs at the same time with a Prefix, Suffix, and Index number.

1. To access the shortcut menu, select multiple feature IDs and then right-click them.

   For more information on the available items, see "Summary Mode Command Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" chapter of the PC-DMIS Core documentation.

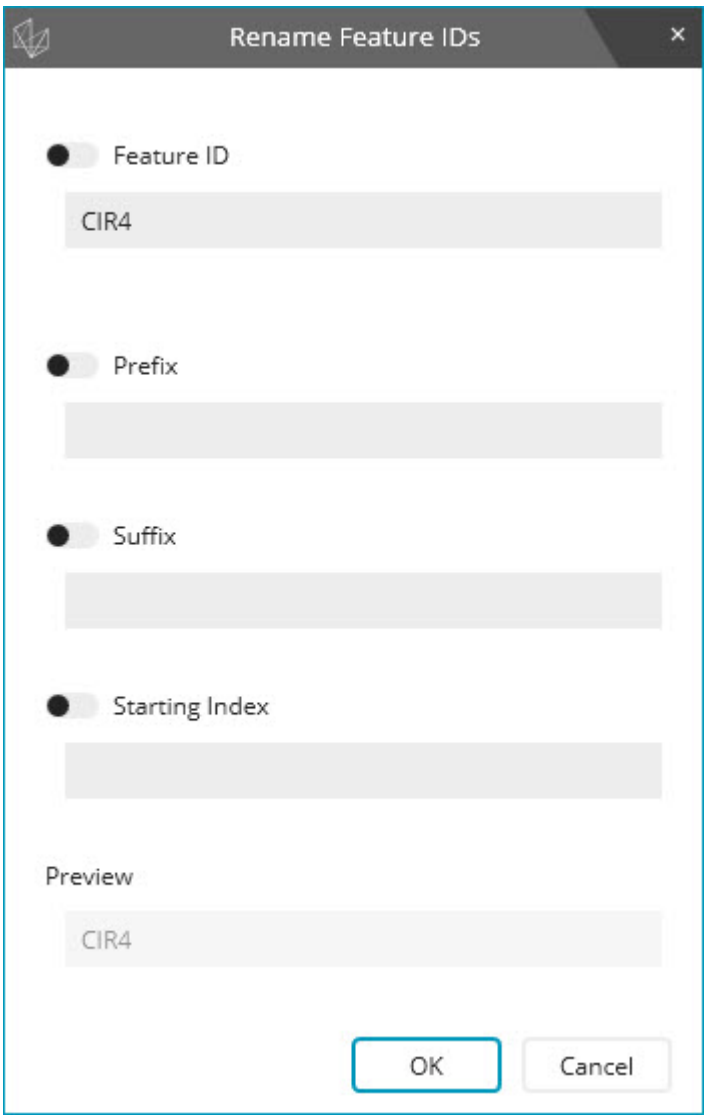
2. From the shortcut menu, select **Rename Feature IDs** to open the **Rename Feature IDs** dialog box. For information about the dialog box, see "Rename Feature IDs dialog box".
3. From the **Rename Feature IDs** dialog box, define a new unique ID.
4. Click **OK**.

PC-DMIS replaces the names of all the selected IDs with the new names in the Edit window.

Rename Feature IDs dialog box

This dialog box allows you to:

- Rename multiple feature IDs in a measurement routine
- Add a prefix to multiple feature IDs in a measurement routine
- Add a suffix to multiple feature IDs in a measurement routine
- Add an index number to multiple feature IDs in a measurement routine



Rename Feature IDs Dialog Box

Feature ID - This option enables you to define multiple feature IDs. PC-DMIS displays the first feature that you selected. To modify a feature ID, use the slider switch to enable the **Feature ID** option for editing. You can then type a new ID name.

Feature IDs must always start with a letter.

Prefix - This option enables you to define a prefix for multiple feature IDs.

Suffix - This option enables you to define a suffix for multiple feature IDs.

Starting Index - This option enables you to define a starting index number for multiple feature IDs. PC-DMIS increments the number by 1 after the first feature.

You must type an index number when you turn on the **Feature ID** option in the **Rename Feature IDs** dialog box.

Preview - This box shows you the preview of the first item of the selected multiple features.

Removing Commands

To remove a command from a measurement routine, select an object and press the Delete key.

If a group command contains any sub-commands, those commands are also deleted.

Copying, Cutting, and Pasting Commands

To copy or cut a command, follow these steps:

1. Select the desired command.
2. Choose the copy or cut command from any of these locations:
    - The Edit window's **Edit** menu
    - The keyboard (Ctrl + C for copy, and Ctrl + X for cut)
    - The shortcut menu's **Cut** or **Copy** option that appears when you right-click on the command

If the command contains any sub-commands, PC-DMIS also cuts or copies those commands.

To paste a command that you have cut or copied, follow these steps:

1. Select the command that precedes the command that you want to paste.
2. Choose the paste command from any of these locations:
    - The Edit window's **Edit** menu
    - The keyboard (Ctrl + V)
    - The shortcut menu's **Paste** option that appears when you right-click on the command.

When you paste a command, PC-DMIS keeps the original command's ID, so two commands then have the same ID. Give the pasted command a new ID that the measurement routine does not already use. If you type an ID that another command already uses, PC-DMIS does not accept it.

Dragging and Dropping Commands

To rearrange commands, you can cut and paste commands in Summary Mode. You can also drag and drop one or more contiguous commands to a new location.

To drag and drop a command, follow these steps:

1. Select one or more contiguous commands in the Edit window.
2. Use the left mouse button, and drag the selected commands to a new location in the Edit window.
3. Release the mouse button to move the selected command below the command that was under the mouse pointer when you released the mouse button.

Marking and Unmarking Commands for Execution in Summary Mode

To mark or unmark a command for execution in Summary Mode, follow these steps:

1. Select the command in the Edit window.
2. Press F3.

Editing from a Dialog Box

To edit a command via the dialog box specific to that command, follow these steps:

1. Select the command in the Edit window.
2. Double-click, press F9, or select **Edit | Command** (or right-click on the command and choose **Edit** from the pop-up menu) to open the appropriate dialog box for the command. Be aware that in some cases, you can only edit a command if PC-DMIS is in Command Mode.

For commands that do not use a dialog box, F9 does nothing, and **Edit | Command** is unavailable for selection.

Showing and Hiding Features

In the Summary Mode, you can show and hide features in the Graphic Display window. The **Show / Hide** button shows the current visibility state of the feature.

- This button means the feature is already visible. Click it to hide the feature.

- This button means the feature is already hidden. Click it to show the feature.

You can also use this with multiple features:

This option toggles the display of selected feature commands and their IDs in the Graphic Display window. You can also use this on a GROUP command to toggle the display of features within the group.

From the Feature Shortcut menu in the Graphic Display window:

- If you select **Hide Feature** for a feature, PC-DMIS hides the feature and its ID in the Graphic Display window and Summary Mode turns off the **Show / Hide** button.
- If you select **Hide ID** for a feature, PC-DMIS hides the ID of a feature in the Graphic Display window but Summary Mode does not turn off the **Show / Hide** button.

Search and Filter Toolbar

You can use the **Search and Filter** toolbar to find specific features, dimensions, commands, or any other data in your measurement routine.

The **Search and Filter** toolbar is available only in the Summary Mode.

Search and Filter Toolbar

Filter by Text (Search)

You can type characters in the **Search** box to dynamically filter items in the Edit window according to the typed text. To clear the filter, click the **Remove** button. PC-DMIS displays the number of search results in the Edit window below the Search bar.

Filter by Text (Search)

To clear the Search box, click the **Clear Search** button.

Filter by Advanced Search

For more complex searches, you can use the **Advanced Search** option. This option allows you to type a string or expression into the **Search** box and, along with various filter options, display the resulting matches in the Edit window.

The available filtering options are:

- **Pattern match** - This option allows you to use special notations and characters in the **Search** box to define patterns of text to match items defined in your measurement routine. For more information, see "Pattern Match".

- **Match case** - This option defines whether to perform a case-sensitive search or not.
- **Match whole word only** - This option defines whether the search returns whole-word matches.

When you select the check box for any of the above advanced search options, PC-DMIS changes the search icon to show this icon .

> You cannot use the **Pattern match** option with the **Match whole word only** option.

Filter by Dimension

You can use the **Filter** button ( ) to filter according to dimension type. You can select multiple dimensions for your filter.

Filter by Dimension

For information on the available dimension types, see "Dimension Toolbar".

When you filter by dimension type, a small **Clear Filter** icon ( ) appears on top of the **Filter** button.

You can click the **Clear Filter** icon ( ) to clear all the dimension filters.

You can use a combination of filters to filter by text and by dimension at the same time. Suppose you want to search for all circle features that have a Position dimension in your measurement routine. To do this, follow these steps:

1. In the **Search** box, type "CIR". PC-DMIS filters the commands to show all the circles that start with a feature ID "CIR".
2. Click the **Filter** button ( ) to see the list of dimension types.
3. Select **Position Dimension** .

PC-DMIS displays all the circles with the Position dimension.

Pattern Match

With the **Pattern Match** option, you can search and filter your Summary Mode text using regular expressions.

Regular expressions provide a powerful, flexible, and efficient method to find text that matches a pattern that you define.

Pattern matches are case insensitive unless you choose **Match case** from the advanced options.

Common Operators for Expressions

The table below has a few common operators that you can use to build your search expression.

- You can combine multiple operators together in your search expression.
- The highlighted text in the Examples column below indicates matched text.

Operators	Description	Examples
.	Use this operator to match any single character, except a newline character.	c.r matches "**CIR**1", "**CIR**2", "**Car**", and "**Cor**osive" but not "Cody".
*	Use this operator to match zero or more instances of the preceding token or character. This matches as many characters as possible (greedy). • Precede this character with . to match one or more instances of any character. • Follow this character with ? to match as few characters as possible (lazy).	C*r matches “CI**R**1” and “SPHE**R**E1” In the phrase “CIR1 CIR2”, c.*r matches “**CIR1 CIR**2” (greedy) But c.*?r matches “**CIR**1 CIR2” (lazy)
+	Use this operator to match one or more instances of the preceding token or character. As discussed in the row above, you can follow this with ? to make the search lazy and match as few characters as possible.	L+1 matches "E**LL1**" and "CY**L1**" but not "CYL2"
^	Use this operator to match the beginning of a line followed by the text that begins the line.	^cir matches “**CIR**1 = CIRCLE(CONTACT)” but not “FCFCIRTY1 Passed : CIR1”.

$	Use this operator to match text that is at the end of a line. You must place the text that you want to search before this operator.	And$ matches “MOVESET1 = MovesetComm**and**”.
[a-f]	Use square brackets to match any single character from a range of characters within the brackets.	S[i-t] matches “**St**art”, “**So**rt”, and “**Si**ze” but not “Sam”.
|	Use this operator to match either of the strings on either side of the operator. This works similarly to an “or” function.	Active (Workplane|Tip) matches "**Active Workplane**" and "**Active Tip**" but not "Active Role".
\	Use this operator to suppress special meaning in any character or token after the operator.	\+ matches the text “ASSIGNMENT(V1 = 10**+**5)” rather than treating the + character as a regular expression operator.
\w	Use this operator to match any single character in a whole word.	Mo\we matches “Mode” and “Move” but not “Motor”.
\s	Use this operator to match any whitespace character.	Manual\sMode matches “**Manual Mode**” but not “Manual-Mode”.
\d	Use this operator to search any decimal digit character.	t\d matches “**T1**A0B0” but not “tip” or “top”

?	Use this operator to match the preceding token zero or once. (In essence, this makes the preceding token optional.) You can also use this token to make a search lazy (non-greedy). See * and + above.	m\w+? matches “co**mm**and” and “Align**me**nt”. \+?2 matches “3 **+ 2**” and “3**2**”.
(lar)	Use parentheses to group the characters within them together as a single token.	c.rc.(lar)? matches “**circ**le” and “**circular**” and “**circu**mference”.

Regular expressions can be difficult to understand and the information here provides some basics. For additional information and examples on regular expressions, we invite you to consult trusted resources on the Internet.

- This site provides examples and additional information: Microsoft | Learn : Use regular expressions in Visual Studio
- This site may be useful to help you test and create your own regular expressions: Regular Expressions 101

Using the Peek Window

The Peek window displays a command's information when you hover over the command in the Edit window. This feature enables you to quickly identify a command's properties and become accustomed to the command's language.

The Peek window automatically updates with the properties of the command that you point to. You do not need to click on the command to update the information in the Peek window.

Hits do not appear for features in the Peek window. For a group, only the first line of the commands in that particular group appears in the Peek window.

To use the Peek window, follow these steps:

1. Access the Edit window. For help, see "Using the Edit Window: Introduction".
2. Place the Edit window in Summary Mode. For help, see "Working in Summary Mode".

3. To enable the Peek window, mark the **Show Peek Window** check box on the **General** tab in the **Setup Options** dialog box. (The Peek window is enabled by default.) For help, see "Show Peek Window".

   In Summary Mode, you can also select the **Peek Window** icon ( ) from the **Settings** icon ( ) in the Edit window to enable the Peek window.

4. To use the Peek window, position your mouse pointer over a command. The Peek window aligns to the center of the command.

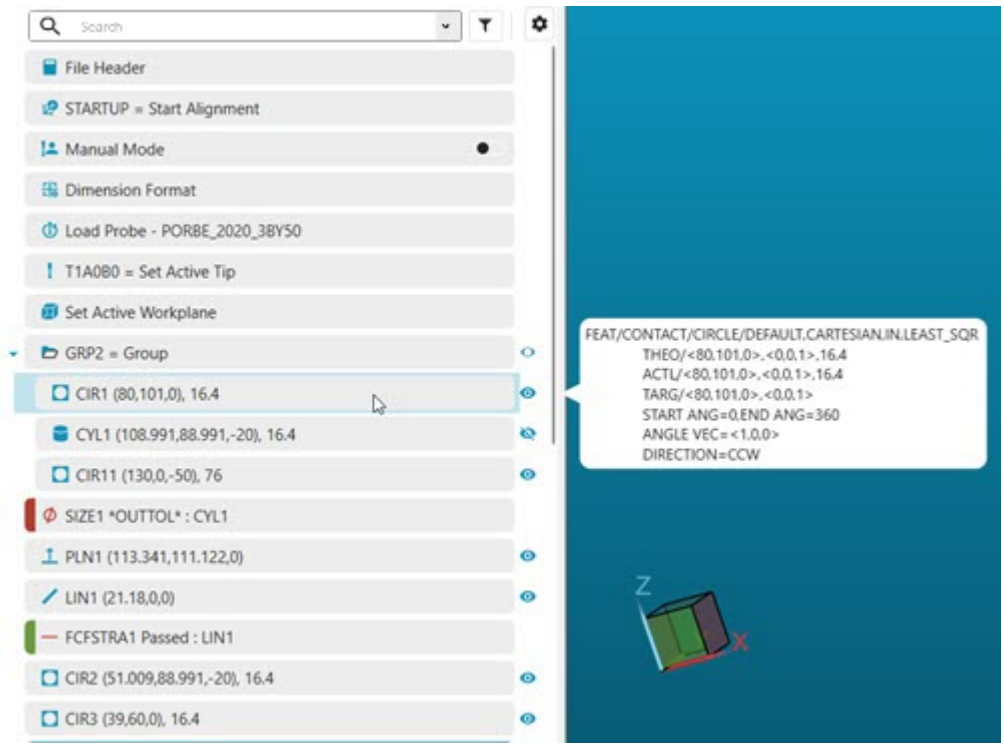
   If you move the mouse pointer, the Peek window becomes transparent with no text in it. The transparent Peek window enables you to see the application through it. The transparent window follows the mouse up and down the commands in the Edit window until you stop and hover over a command. At this point, the Peek window becomes active again and displays the command's information.

5. Execute your measurement routine.

   In Summary Mode, PC-DMIS displays an icon that indicates the execution status. For more information, see "Marking and Unmarking Commands in Summary Mode".

6. To close the Peek window, do one of the following:

   - Select a command.
   - Close the Edit window.
   - Move your mouse pointer to another area in PC-DMIS.
   - Press any sequence of keys.
   - In Summary Mode, select the **Peek Window** icon ( ) from the **Settings** icon ( ).



Example of an active Peek window.

Summary Mode Shortcut Keys

The following list shows the available keyboard functions:

- Shift + Tab - Moves the insertion point up one line.
- Ctrl + Click on feature in Graphic Display window - Moves the insertion point to the corresponding command item in the Edit window.
- Ctrl + A - Selects all of the commands in the Edit window.
- Ctrl + C - Copies text or objects to the Clipboard.
- Ctrl + End - Moves the cursor to the end of the measurement routine.
- Ctrl + Home - Moves the cursor to the beginning of the measurement routine.
- Ctrl + Q - Displays the **Execution** dialog box and automatically executes the measurement routine.
- Ctrl + V - Pastes the Clipboard contents. If you selected a field that can take a value, it pastes the text in that field. If a command has been cut or copied, it pastes the command after the current selected command.
- Ctrl + X - Cuts any selected text from a field that can take a value and stores it in the Clipboard. If you've selected a command, it cuts that command and stores it in the Clipboard.

- Ctrl + Y - Displays the **Execution** dialog box after you cancelled the execution and allows you to continue execution of the measurement routine from that specified location.
- Delete - Deletes any selected characters from a field that can take a value. If a command or command block is selected, and you can normally delete it, this function deletes it.
- Double-click - Opens a dialog box for the currently selected command if a corresponding dialog box exists for the command. If it's a group item, PC-DMIS expands and collapses a group item. You can also double-click on the `MODE` command in the Edit window to switch between the DCC Mode and Manual Mode.
- Down Arrow - Moves the insertion point down one line.
- F3 - Marks a single feature for execution. It also marks and unmarks multiple selected commands.
- F9 - Opens a dialog box for the currently selected command if a corresponding dialog box exists for the command. (For the `MODE` command in the Edit window, this key switches between the DCC Mode and Manual Mode. You can also double-click on a `MODE` command to switch between the two modes.)
- Page Down - Moves the insertion point down one page.
- Page Up - Moves the insertion point up one page.
- Shift - Facilitates selecting a block of objects. Select an object, hold Shift, and then click again to select a range of objects.
- Tab - Moves the insertion point down one line.
- Up Arrow - Moves the insertion point up one line.
- Left Arrow - Collapses an expanded command or group.
- Right Arrow - Expands a collapsed command or group.
- Alt + Backspace - Undo. Undoes the last edit made.
- Shift + Backspace - Redo. Redoes the last undone edit.

Related Topics

Using Shortcut Keys and Shortcut Menus

Working in Command Mode

Command Mode allows you to insert or edit a wide variety of PC-DMIS commands into the Edit window and to work with the Edit window in a pure textual format. To place the Edit window in Command Mode, select **View | Command Mode**, or from the **Edit Window** toolbar, click the **Command Mode** icon .

The main features or commands that you can add to the Edit window include:

- Constructed Features
- Dimensions
- Statistical Data
- Alignment
- Measured Features
- Hits
- Motion Commands
- Move Commands
- Sheet Metal Measurements
- Probe
- Comments
- Tracefields
- Expressions
- Screen Captures
- Global Commands
- Command Mode Shortcut Keys

You can also determine which commands get displayed in Command Mode. For additional information, see "Command Mode Display Options" in the "Setting Your Preferences" chapter.

Constructed Features

PC-DMIS allows you to create specified features from existing features. Additional information including specific rules for constructing features can be found in the "Constructing New Features from Existing Features" chapter.

PC-DMIS supports the construction of the following features:

- POINT
- CIRCLE
- ELLIPSE
- SPHERE
- LINE
- CONE
- CYLINDER
- PLANE
- SLOT
- CURVE
- SURFACE
- FEATURE SET
- GAUSSIAN FILTER SET

Dimensions

Dimension commands allow you to dimension features or the relationship between features. You can find additional information regarding the calculation of dimensions in the "Using Legacy Dimensions" chapter.

Dimension Format

Only if you mark a dimension for printing does the data appear on the inspection report. The `FORMAT` command allows you to create more than one dimensional format within a measurement routine. This allows you to show or hide the information stored in any dimension that follows the `FORMAT` command.

For example, for some dimensions, you may not want the nominals printed, such as with certain types of zone tolerances that always have a nominal of zero. In these cases, you can hide the `NOM` field. Later in the measurement routine you might want to display the nominals for a different type of dimension, such as Location or Position. The `FORMAT` command lets you control these manipulations.

The default `FORMAT` command prints out all of the columns and column headings. It also creates the statistics XSTATS11.TMP file.

For details on the locations of PC-DMIS files, see "Understanding File Locations".

- To hide an item, move your pointer over a field, once it turns into region you can click, select the empty item at the bottom of the list of options.
- To show an item, type it in the field and press Tab.

You can also press F8 or F7 to toggle the field values.

Command line in the Edit window:

```
FORMAT/
TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;NOM,TOL,MEAS,DEV,OUTTOL, ,
```

TEXT = This field controls whether or not additional text appears in the dimension's command block.

HEADINGS = This field controls whether the dimensions that follow the `FORMAT` command will have column headings above the numbers.

OPTIONS = This field controls whether or not option fields are displayed. These options include the following:

GRAPH = This shows or hides graphical analysis of the dimension.

TEXT = This shows or hides textual analysis of the dimension

MULT = This sets the multiplier for deviation arrows and tolerance zone used in the Graphic Display window. This is only used if GRAPH = YES.

OUTPUT = This determines where to send dimensional output. Options are statistics, report, both statistics and report, or none.

ID = This shows or hides the dimension's ID in the header.

SYMBOL = This field controls whether the deviation symbols will be printed. This field can be toggled on or off. A blank field indicates that this command is off.

SD = This shows or hides the dimension's standard deviation value.

NOM,TOL,MEAS,MAXMIN,DEV,OUTTOL,DEVANG = These fields appear after the semi-colon character (;). They control which columns get printed for dimensions following the `FORMAT` command. You can toggle any combination of these columns on or off. They can also be displayed in any order. Simply type the desired command in the order needed. (The order is displayed numerically in the **Parameters Dimension** dialog box.)

For more information consult the "Using Legacy Dimensions" chapter.

Available Dimensions

To edit these dimensions, view their individual sections in the "Using Legacy Dimensions" chapter.

Analysis

To edit the Analysis information, see the "Inserting Report Commands" chapter.

Statistical Data

To include and edit statistical data, see the "Tracking Statistical Data" chapter.

Alignment

The **Alignment** option allows you to recall or create a new coordinate system.

Alignment Commands

For more information on how to use these alignment commands in the Edit window, see the individual topics in the "Creating and Using Alignments" chapter.

CAD Equals Part

For more information on how to use the **CAD Equals Part** command in the Edit window, see "Equating CAD to Measured Part Data" in the "Creating and Using Alignments" chapter.

Equate Alignment

For more information on how to use Equate Alignment command in the Edit window, see "Equating an Alignment" in the "Creating and Using Alignments" chapter.

Measured Features

You can insert commands that create the following measured features:

- Measured Point
- Measured Line
- Measured Plane
- Measured Sphere
- Measured Cylinder
- Measured Cone
- Measured Circle

These features, along with information about measured features in general, are discussed in the "Creating Measured Features" chapter.

Hits

This command targets the location for actual touches by the probe.

Basic Hit

```
HIT/BASIC,x,y,z,i,j,k,x,y,z,USE_THEO=YES/NO
```

This is the simplest hit format. See "Understanding the Command Format" in the "Creating Measured Features" chapter for more information.

Sheet Metal Hits

```
HIT/type,x,y,z,i,j,k,x,y,z
```

type = VECTOR, SURFACE, EDGE, CORNER, ANGLE

There are five available types of sheet metal hits. The Sheet Metal package must be installed for this option to be available. (See "Sheet Metal Measurements".)

Adding Hit Lines

To add a hit line, place the cursor in the desired location and press the Enter key. Begin by typing the command **HIT**. Press the Tab key. PC-DMIS adds the new line depending on where the cursor is located. If the cursor is in the middle of a command, a new line is

created below the current line. If the cursor is placed anywhere within the first four lines PC-DMIS creates the new line immediately after the measured line.

Deleting Blank Lines

To delete a blank line press the Down Arrow or Enter key. The line may also be highlighted and deleted. (See "Command Mode Keyboard Functions".)

Motion Commands

The various Motion commands control the movement of the machine. All but the FASTPROBEMODE command are discussed in greater detail in the "Parameter Settings: Motion tab" and the "Parameter Settings: Optional Motion tab" topics of the "Setting Your Preferences" chapter.

Mode = MANUAL / DCC

```
MODE/MANUAL (or DCC )
```

This command allows you to select between Manual Mode and DCC Mode.

Prehit Distance

```
PREHIT/nnn.nnnn
```

This command determines the distance away from the theoretical hit location on the surface where PC-DMIS begins to search for the part. The machine travels at touch speed while traversing this distance to search for the part.

`nnn.nnnn` represents a number value for the machine prehit distance.

For additional information, see the "Prehit Distance" topic in the "Setting your Preferences" chapter.

```
MOVESPEED/nnn.nnnn
```

`nnn.nnnn` represents a number value for the speed. This command changes the point to point positioning speed of the CMM. Depending on the state of the **Display absolute speeds** check box in the **Part/Machine** tab of the **Setup Options** dialog box (**Edit |**

Preferences | Setup), this is either an absolute speed (mm/sec) or a percentage of the machine's total speed capability.

Touch Speed

```
TOUCHSPEED/nnn.nnnn
```

`nnn.nnnn` represents a number value for the speed. This command changes the speed at which the CMM takes hits. Depending on the state of the **Display absolute speeds** check box in the Part/Machine tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**), this is either an absolute speed (mm/sec) or a percentage of the machine's total speed capability.

Scan Speed

```
SCANSPEED/nnn.nnnn
```

`nnn.nnnn` represents a number value for the speed. This command changes the speed at which the CMM scans the part. Depending on the state of the **Display absolute speeds** check box in the Part/Machine tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**), this is either an absolute speed (mm/sec) or a percentage of the machine's total speed capability.

Retract Distance

```
RETRACT/nnn.nnnn
```

`nnn.nnnn` represents a number value for the distance. This command determines the distance the machine will move away from the actual hit location before going from "Touch Speed" to "Move Speed".

How the probe retracts depends on the CMM controller:

- **Some controllers handle the retract internally.** After a hit, the controller automatically retracts the probe, but it may only know the prehit distance, not the retract distance. In these cases, the controller retracts the probe to the prehit standoff, and PC-DMIS then issues an additional move to reposition the probe at the retract distance. If the retract distance is less than or equal to the prehit distance, the probe remains at the prehit standoff; PC-DMIS does not move the probe back toward the part after the controller's retract. *The retract value takes effect only when it is greater than the prehit.*
- **Some controllers do not retract on their own.** In these cases, PC-DMIS issues the move to perform the retract, and the distance is based on the probe tip's ball surface to the part's theoretical hit location.

When the controller does perform its own retract, the distance may be calculated from either the ball surface or the ball center to either the theoretical or measured hit location, depending on the particular controller.

Check Distance

```
CHECK/nnn.nnnn,p.pp
```

This command determines the distance in inches or millimeters (depending on the system of measurement that was initially set up for the particular measurement routine) past the theoretical hit location that the machine will continue to search for the surface of the part until it determines the surface is not there.

nnn.nnnn:

This represents the check distance.

p.pp:

This represents the percentage of the total check distance that PC-DMIS moves when performing a Find Hole operation. The default value is 1 which means 100% of the check distance. Thus .1=10%, .2=20%, .3=30% etc. This only functions with Find Hole operations.

- If PC-DMIS finds a surface within the specified check distance, it takes a hit.
- If no surface is found, PC-DMIS displays an error message that it has encountered an unexpected end of move.

If you enter .3 inches as the check distance, PC-DMIS moves 0.3 inches past the theoretical surface, searching for a surface on which to take the hit.

For additional information, see the example in the "Check Distance" topic in the "Setting your Preferences" chapter.

Using Fast Probe Mode

Use the **Insert | Parameter Change | Probe | Fast Probe Mode** menu item to insert a `FASTPROBEMODE/OFF` command into the Edit window. To enable Fast Probe Mode, after you insert this command, highlight OFF and press F8 to change it to ON.

This probing mode applies to analog scanning probes when taking individual hits rather than scanning. When it is ON, it provides a significant reduction in the time required to take individual hits. The biggest reduction in time occurs when the probe retracts from the part.

Some probe types (such as the SP25) do not support Fast Probe Mode. Some machines do not support Fast Probe Mode regardless of the probe type. If you have a measurement routine with the `FASTPROBEMODE/ON` command in it, but your probe type or machine does not support it, the command does nothing.

Move Commands

The commands listed below provide you with the ability to alter the movement of the probe between hits.

```
MOVE/POINT
MOVE/INCREMENT
MOVE/CLEARPLANE
MOVE/CIRCULAR
MOVE/SYNC
MOVE/SWEEP
MOVE/ROTAB
MOVE/EXCLUSIVE_ZONE
```

These are described in the "Inserting Move Commands" chapter.

Sheet Metal Measurements

Sheet metal measurements, also known as "Auto Features", are available only as an additional option to the basic PC-DMIS geometric software package. For more information, see the "Creating Auto Features" chapter in the PC-DMIS Core documentation. This option provides several alternative methods for taking hits. The available sheet metal features are listed below. When applicable, you can update the **Number of Hits**, **Number of Rows**, **Spacer**, and **Indent** values. The required minimum value appears as the default.

The following sheet metal measurements are available:

- Auto Vector Point
- Auto Line
- Auto Plane
- Auto Circle
- Auto Ellipse
- Auto Notch Slot
- Auto Round Slot
- Auto Angle Point
- Auto Corner Point
- Auto Edge Point
- Auto High Point
- Auto Surface Point
- Auto Square Slot
- Auto Cylinder
- Auto Cone
- Auto Sphere

Probe

The following commands allow you to access options that affect the probe. These commands allow you to change the active tip in a probe cluster or change the position of the rotating probe head. Probe compensation can also be turned on or off as needed.

Load Probe

```
LOADPROBE/<probe name>
```

In the Load Probe command, <probe name> represents a user editable field that allows you to load a file of qualified probe tips to be used within the measurement routine. For example, `LOADPROBE/TESASTAR` loads a probe named TESASTAR.

The **File | New** and **File | Open** menu options are unavailable while a probe change is in progress. This protects your hardware by preventing you from opening or creating a measurement routine that could power on a laser sensor and damage the probe mounted on the head.

You can only load probe files made in valid versions of PC-DMIS. For this version of PC-DMIS, if you attempt to load a probe file created before PC-DMIS version 2021.1, PC-DMIS shows an error message.

Example error message when you try to load an invalid probe file:

PC-DMIS Message

PC-DMIS cannot read this probe file <probe file>. This happens because of these reasons: It may be corrupt. It may come from a newer version. It may come from an unsupported earlier version. The earliest supported version is 2021.1.

Where <probe file> is the name and location of the file you are trying to open.

Probe Comp

```
PROBECOM/ON (or OFF)
```

The Probe Comp command allows you to turn probe compensation on or off. It is considered ON if the command is displayed in the Edit window.

Read Point

For information, see "Creating a ReadPoint from the Probe's Position" in the "Creating Generic Features" chapter.

Tip

The tip command tells PC-DMIS to use the specified probe tip.

```
TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
```

The `SHANKIJK` is just another vector form of specifying the A and B angles for the tip.

The `ANGLE` specifies the angle that the tip transformation matrix is rotated about the shank vector.

Comments

This option allows you to display comments during the execution of the measurement routine, or to send the comments to the inspection report. These comments are created using the **Insert | Report Command | Comment** menu option. The **Show Comments** option also allows you to add, edit, or delete operator notes and inspection report comments within the Edit window. The `COMMENT` command switches between the options `OPER`, `REPORT`, `YESNO`, `$$`, `INPUT`, and `READOUTS`.

After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

When you open a measurement routine that was saved from a later version, any commands that are not supported in the current version will show up as DOC comments.

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Operator

This option displays a message to the operator when the measurement routine is executed.

You can type directly into the Edit window:

1. Type the command `COMMENT/OPER` in the desired location of the Edit window.
2. Type in the desired text you want to display to the operator, and press Enter.

You can also access this option through a dialog box:

1. Select **Insert | Report Command | Comment** from the menu bar to show the **Comment** dialog box.
2. Then select **Operator**.
3. In **Comment Text**, type your text to display.
4. Click **OK**.

When the operator executes the measurement routine with this command, a message displays the comment to the operator.

You can press F9 while the cursor is in a `COMMENT/OPER` command line to display the **Comments** dialog box, allowing you to alter the displayed message.

Command line in the Edit window:

```
COMMENT/OPER,NO,Full Screen=YES,

comment text
```

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Report

This option sends text to the inspection report when the measurement routine is executed. Type the command `COMMENT/REPT` in the desired location of the Edit window. Type in the desired text, and press Enter. You can also access this option by selecting **Insert | Report Command | Comment** from the menu bar to show the **Comment** dialog box. Then select **Report**.

When the operator executes the measurement routine, PC-DMIS does not display these messages during execution. However, it does send them to the inspection report.

If you press the F9 key while the cursor is in a `COMMENT/REPT` command line, PC-DMIS displays the **Comments** dialog box, allowing you to alter the displayed message.

Command line in the Edit window:

```
COMMENT/REPT,
```

```
comment text
```

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Input

This option is similar to Operator in that it allows you to display text when you execute a measurement routine. In addition to displaying a message, a comment box appears so the operator can type information that is then written to the inspection report.

Type the command `COMMENT/INPUT` in the desired location of the Edit window. Type in the desired text. The input from the operator is assigned to the Comment ID and is accessible via an expression reference (such as `C1.INPUT` for example).

You can also access this option by selecting **Insert | Report Command | Comment** from the menu bar to open the **Comment** dialog box. Then select **Report**.

Pressing the F9 key while the cursor is in a `COMMENT/INPUT` command line displays display the **Comments** dialog box, allowing you to alter the displayed message.

Command line in the Edit window:

```
comment ID=COMMENT/INPUT,NO,Full Screen=NO,
            comment text
```

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

$$ (Document)

This option provides you with the ability to document (add programmer comments to) the internal measurement routine. It will not display any text when executing the measurement routine. Visually, this command does not have the usual COMMENT prefix to the command in the Edit window. This is done to help the comment stand out inside the Edit window.

Command line in the Edit window:

```
$$ NO,
          comment text
```

To type a document comment directly into the Edit window:

1. Type COMMENT and press Tab. PC-DMIS highlights the OPER field.

2. Type $$ and press Tab or Enter.

You select **Insert | Report Command | Comment** from the menu bar to show the **Comment** dialog box. Then, from the **Comment type** area, select **Document**.

When you open a measurement routine that was saved from a later version into the current version, any commands that are not supported in the current version show up as DOC comments. See "Save As" in the "Using Basic File Options" chapter.

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Yes / No

This option displays a message box with a question and **YES / NO** buttons when the measurement routine is executed.

1. Type the command `COMMENT/YESNO` in the desired location of the Edit window.
2. Type in the desired question that you want to display to the operator, and press Enter.

You can also access this option by selecting **Insert | Report Command | Comment** from the menu bar to show the **Comment** dialog box. Then select **Yes / No**.

The response to the YES / NO question appears in the measurement routine. The text 'YES' or 'NO' is associated with the comment identification and is available in any expression via a reference to the Comment ID (such as C1.INPUT for example).

Command line in the Edit window:

```
comment ID = COMMENT/YESNO,NO,Full Screen=NO,
             comment text
```

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Readouts

This option allows you to display text inside the Probe Readouts window when you execute the measurement routine. Type the comment `/READOUTS` in the desired location of the Edit window. Type in the desired text you want to display in the Probe Readouts window, and press Enter. You can also access this option by selecting **Insert | Report**

Command | Comment from the menu bar to show the **Comment** dialog box. Then select **Readouts**.

If you press the F9 key while the cursor is in a `COMMENT/READOUTS` command line, PC-DMIS displays the **Comments** dialog box, allowing you to alter the displayed message.

The Edit window command line for this option reads:

```
COMMENT/READOUTS,
                comment text
```

For more information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

Tracefields

PC-DMIS uses tracefields to send additional information into your statistical database along with your measurement results. The **Insert | Statistics Command | Trace Field** menu option opens the **Trace Field** dialog box.

For more information about tracefields, see "Using Tracefields".

Expressions

You can insert expressions into most of PC-DMIS's editable fields. See the "Using Expressions and Variables" chapter.

Screen Captures

You can insert a `DISPLAY/METAFILE` command to insert screen captures of the Graphic Display window into your report. See the "Inserting Screen Captures" topic in the "Inserting Report Commands" chapter.

Global Commands

The table below defines the list of global commands available in the Edit window. While you cannot alter these commands, the Edit window does allow you to select the minor commands following these major commands.

To select the minor commands, do the following:

1. Press the Tab key to select the minor command in the Edit window.
2. Press F7 or F8. This toggles through the available choices.

This list serves as a comprehensive summary of the commands that can be added to a measurement routine. See each specific section for more information as necessary.

The maximum number of characters that PC-DMIS can manage within any line of the Edit window is 280.

MAJOR COMMAND	MINOR COMMAND
2DANGLE (see "Dimensioning Angle")	A
2DDISTANCE (see "Dimensioning Distance")	M
3DANGLE (see "Dimensioning Angle")	A
3DDISTANCE (see "Dimensioning Distance")	M
ALIGNMENT (see the "Creating and Using Alignments")	BF2D, BF3D, ITERATE, LEVEL, ROTATE, TRANS, ROTATE_OFFSET, TRANS_OFFSET, RECALL, RECALL_EXTERNAL
ANGULARITY (see "Dimensioning Angularity")	M
ANALYSISVIEW (see "Analysis area")	
ARRAY INDICES (see "Arrays")	
ASSIGN (see "Using Variables with Expressions")	
ATTACH (see "IDs for features within subroutines, basic scripts, or external routines")	

AUTO (see "Sheet Metal Measurements")	VECTOR POINT, SURFACE POINT, EDGE POINT, CORNER POINT, ANGLE POINT, HIGH POINT, SPHERE, CIRCLE, CYLINDER, SQUARE SLOT, ROUND SLOT, CONE, ELLIPSE, NOTCH, CONE
AUTOTRIGGER (see "Parameter Settings: Probe Trigger Options tab")	
CAD EQUALS PART (see "CAD Equals Part")	
CALCULATION	
CALLSUB (see "Calling a Subroutine")	
CASE (see "Case / End Case")	
CHECK (see "Check Distance")	
CLEARP (see "Parameter Settings: Clearance Plane tab")	"WORK PLANE VALUE"
CLAMP (see "Clamping Value")	
COLUMN132 (see "Dimensions")	

COMMENT (see "Comments")	OPER, REPT, INPUT, $$, YESNO, READOUTS After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.
CONCENTRICITY (see "Dimensioning Concentricity")	M
CONST (see "Constructed Features")	CIRCLE, CONE, CURVE, CYLINDER, ELLIPSE, LINE, PLANE, POINT, SET, SURFACE, SPHERE
CIRCULARITY (see "Dimensioning Circularity")	
CYLINDRICITY (see "Dimensioning Cylindricity")	
DEFAULT_CASE (See "Default Case / End Default Case")	
DISPLAY (see "Screen Captures")	METAFILE
DO (see "Do / Until")	
ELSE (see "Else / End Else")	
END_ELSEIF (See "Else If / End Else If")	

ELSE_IF (see "Else If / End Else If")	
END_CASE (see "Case / End Case")	
END_DEFAULTCASE (see "Default Case / End Default Case")	
END_ELSE (see "Else / End Else")	
END_IF (see "If / End If")	
END_SELECT (see "Select / End Select")	
END WHILE (see "While / End While")	
ENDSUB (see "Branching with Subroutines")	
EQUATE (see "Equate Alignment")	
EXTERNALCOMMAND (see "Inserting External Objects")	
FILE (see "Using File Input / Output")	OPEN, MOVE, EXISTS, DIALOG, DELETE, COPY, CLOSE, WRITE_CHARACTER, WRITE_BLOCK, WRITELINE, SAVE_POSITION, REWIND, RECALL_POSITION, READ_UPTO, READ_CHARACTER, READ_BLOCK, READLINE
FLATNESS (see "Dimensioning Flatness")	M

FLY (see "Machine area")	
FORMFEED (see "Inserting a Form Feed Command")	
FORMAT (see "Dimension Format")	HEADINGS,STATS
GAPONLY (see "Dimensioning Location")	ON
GENERIC (see "Creating a Generic Feature")	POINT, PLANE, NONE, LINE, CYLINDER, CONE, CIRCLE, SQUARE_SLOT, SPHERE, ROUNDSLOT
GO TO (see "Jumping to a Label Using GoTo")	
HIT (see "Hits")	BASIC, ANGLE, VECTOR, SURFACE, EDGE, CORNER
IF (see "If / End If")	
KEYIN (see "Dimensioning through Keyboard Input")	M
LABEL (see "Using Labels")	
LEITZPROBE	
LOADPROBE (see "Load Probe")	"FILE NAME"
LOCATION (see "Dimensioning Location")	A, D, M, PA, PR, R, T, X, Y, Z
LOOP (see "Creating a Loop")	START, END

MEAS (see "Measured Features")	CIRCLE, CONE, CURVE, CYLINDER, LINE, PLANE, POINT, SET, SLOT, SPHERE
MODE (see "Probe Mode Toolbar")	DCC, MANUAL
MOVE (see "Move Commands")	POINT, ROTAB, CIRCULAR, CLEARPLANE, INCREMENT, PH9 (X,Y,Z)
MOVESPEED (see "Move Speed")	% of overall machine speed
ONERROR (see "Branching on an Error")	Unexpected_hit, probe_miss
OPTIONMOTION (see "Parameter Settings: Probe Options tab")	
PARALLELISM (see "Dimensioning Parallelism")	M
PD	
PERPENDICULARITY (see "Dimensioning Perpendicularity")	M
POSITION (see "Dimensioning Position")	A, D, M, PA, PR, R, T, V, X, Y, Z
PREHIT (see "Prehit Distance")	"distance"
PROBECOMP (see "Probe Comp")	"ON, OFF"
PROFILE (see "Dimensioning Profile - Line or Surface")	

ROUTINE	END "ends the measurement routine at the command's location"
READPOINT (see "Read Point")	"xyz value of probe location"
RECALL (see "Recalling an Existing Alignment")	VIEWSET, ALIGNMENT (INTERNAL/ EXTERNAL)
RETRACT (see "Retract Distance")	"distance"
RETROLINEARONLY (see "Location Options")	ON, OFF
RMEAS (see "Setting Up Relative Measure (RMEAS)")	
RS	
RT	
RUNOUT (see "Dimensioning Runout - Circular or Total")	M
S	
SAVE (see "Saving an Alignment")	ALIGNMENT
SCANSPEED (see "Scan Speed %")	% of overall machine speed
SCRIPT (see "Inserting BASIC Scripts")	
SELECT (see "Select / End Select")	

STATS (see the "Tracking Statistical Data")	ON, OFF
STRAIGHTNESS (see "Dimensioning Straightness")	M
SUBROUTINE (see "Branching with Subroutines")	
TIP (see "Tip")	"file name"
TEMPCOMP (see "Compensating for Temperature")	
TOUCHSPEED (see "Touch Speed %")	"% of overall machine speed"
TRACEFIELD (see "Tracefields")	(field name : value)
UNTIL (see "Do / Until")	
WHILE (see "While / End While")	
WORKPLANE (see "Settings Toolbar")	TOP, BACK, BOTTOM, FRONT, LEFT, RIGHT

Command Mode Shortcut Keys

The following list shows the various keyboard functions available within the Edit window's Command Mode.

- Alt + F3
  Displays the **Search** dialog box.
- Shift + Tab or Ctrl + Left Arrow
  Moves the cursor backward to the last editable field.

- Ctrl
  Hold Ctrl and click on a feature in the Graphic Display window to move the cursor to the corresponding feature line in the Edit window. Hold Ctrl and double-click on a feature to select the entire feature.
- Ctrl + A
  Selects and highlights all the text in the Edit window.
- Ctrl + C
  Copies text or objects to the Clipboard. If you highlight text from a single field, that text is copied to the Clipboard. If more than one field is highlighted, then the entire object is copied to the Clipboard.
- Ctrl + E
  Executes selected features or the feature or operator comment command on which the cursor rests.
- Ctrl + End
  Moves the cursor to the end of the measurement routine.
- Ctrl + Tab
  Minimizes or restores the Edit window.
- Ctrl + Home
  Moves the cursor to the beginning of the measurement routine.
- Ctrl + Page Down
  Moves the cursor to the end of the file.
- Ctrl + Page Up
  Moves the cursor to the beginning of the file.
- Ctrl + Q
  Displays the **Execution** dialog box so you can execute the measurement routine.
- Ctrl + T
  Assigns the current command to the selected active arm.
- Ctrl + V
  Pastes the Clipboard's contents at the insertion point. If you only copied text, this attempts to paste the text. If you copied an entire object to the Clipboard, this attempts to paste the entire object. In the latter case, if the insertion point is in the first column of the Edit window on the first line of a command, the Clipboard object is placed before this object. In all other cases, the insertion is after the object.
- Ctrl + X
  Cuts any highlighted characters.
- Ctrl + Y
  Displays the **Execution** dialog box and resumes a paused execution.

- Ctrl + Shift + F

  This option toggles the display of selected feature commands and their IDs in the Graphic Display window. You can also use this on a GROUP command to toggle the display of features within the group.

- Delete or Backspace
  Deletes any highlighted characters. If nothing is highlighted, then both keystrokes function as in a normal editor.

- Double-click
  Highlights the editable field that the cursor is on.

- Down Arrow
  Moves the cursor down one line.

- F1
  Displays the Help topic for the command.

- F2
  Inserts an expression.

- F3
  Marks a single feature for execution. Multiple features can be marked simultaneously by highlighting them before selecting this command.

- F4
  Prints the entire contents of the Edit window.

- F5
  Displays the **Setup Options** dialog box.

- F6
  Displays the **Change all the fonts** dialog box.

- F7
  If the cursor is placed in a toggle field, this key switches the entry in the field forward to the next entry in the alphabetical list of allowed entries. If the end of allowed entries is reached, it cycles to the beginning of the list.

- F8
  If the cursor is placed in a toggle field, this key switches the entry in the field backward to the previous entry in the alphabetical list of allowed entries. If the beginning of allowed entries is reached, it cycles to the end of the list.

- F9
  Opens the dialog box for the command on which the cursor is residing when you press this key.

- F10
  Displays the **Parameter Settings** dialog box.
- F12
  Displays the **Fixture Setup** dialog box.
- Left Arrow
  Moves the cursor left one character.
- Page Down
  Moves the cursor down one page.
- Page Up
  Moves the cursor up one page.
- Right Arrow
  Moves the cursor right one character.
- Shift
  If you hold Shift and an arrow key is pressed, the text is highlighted as the cursor moves. Holding Shift and double-clicking on a start block or end block selects the entire command block.
- Shift + F10
  Displays the **Jump To** dialog box.
- Tab + Ctrl + Right Arrow
  Moves the cursor forward to the next editable field.
- Up Arrow
  Moves the cursor up one line.
- - (minus key) or Alt + -
  Removes the last hit from the hit buffer.
- End
  Inserts the measured feature into the measurement routine.

Related Topics

Using Shortcut Keys and Shortcut Menus

Working in DMIS Mode

DMIS Mode displays the Edit window commands in DMIS format. This means you can use the DMIS syntax to edit your measurement routines. To place the Edit window in DMIS Mode, select **View | DMIS Mode** or from the **Edit Window** toolbar, click the **DMIS Mode** icon ( ).

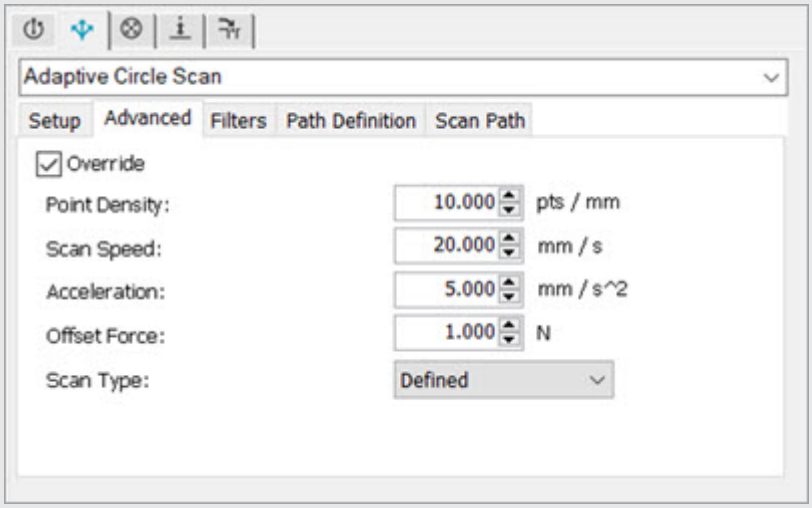
Most of the commands and features that you can add to your measurement routine from Command Mode you can also add from DMIS Mode. For information on how to insert or edit commands in DMIS Mode, see "Understanding Core Concepts".

For information on DMIS, see https://qifstandards.org/.

Copying and Pasting Parameters

The Copy Parameters and Paste Parameters functions help you copy parameters from one feature or dimension and paste them to another feature(s) or dimension(s) in the Edit window. For example, you can copy and paste the following:

- Feature parameters, such as NUMHITS and DEPTH
- Dimension parameters, such as OUTPUT, TEXT, and tolerances



Probe Toolbox Adaptive Scan Strategies tab for Adaptive Circle scan

For Auto features using Adaptive Scanning strategies, you can also use these scanning parameters for Copy and Paste:

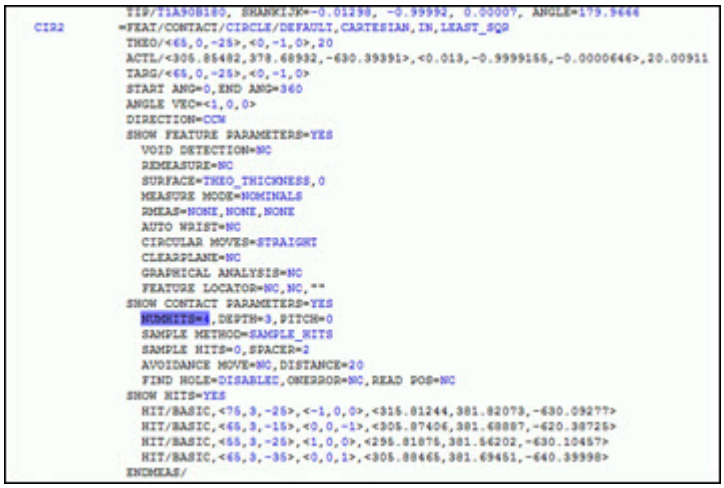
- Point Density
- Scan Speed
- Acceleration
- Offset Force

Nominal values, basic hit values, and vector component values are not parameters; therefore, you cannot copy and paste them.

Copying a Parameter

1. Use one of the following methods to select the parameter(s) that you want to copy:

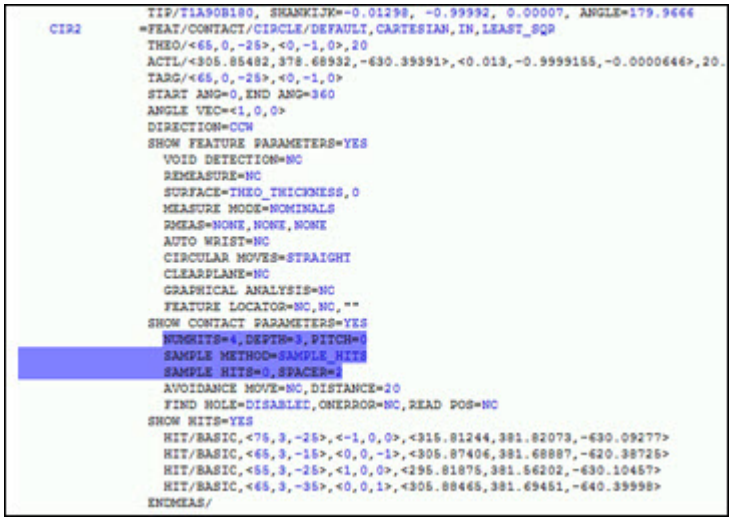
   - To select a single parameter, click anywhere on it or highlight it. For example:



Command Mode

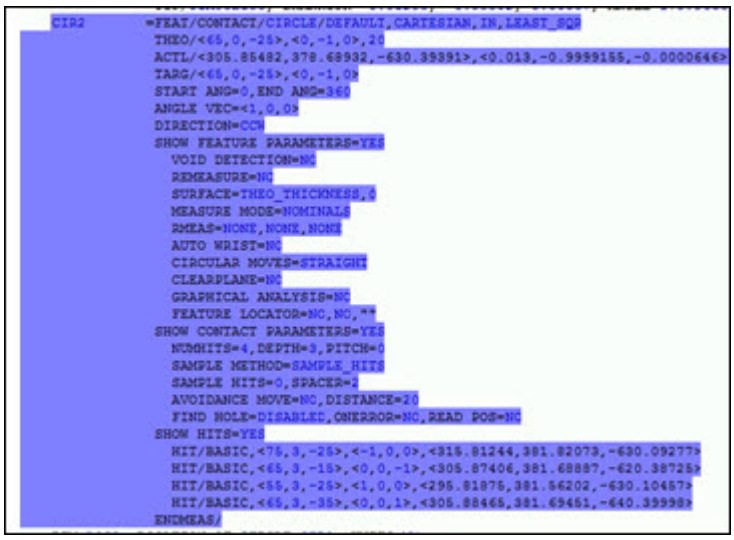
Summary Mode

   - To select multiple, continuous parameters for a single feature, click on the first parameter and then drag your mouse while holding the mouse button. For example:

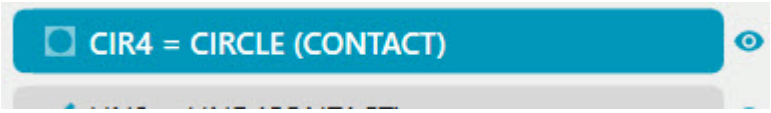


Command Mode

   - To select a complete feature to copy all of its parameters, hold down the Ctrl key and then drag your mouse. For example:



Command Mode



Summary Mode

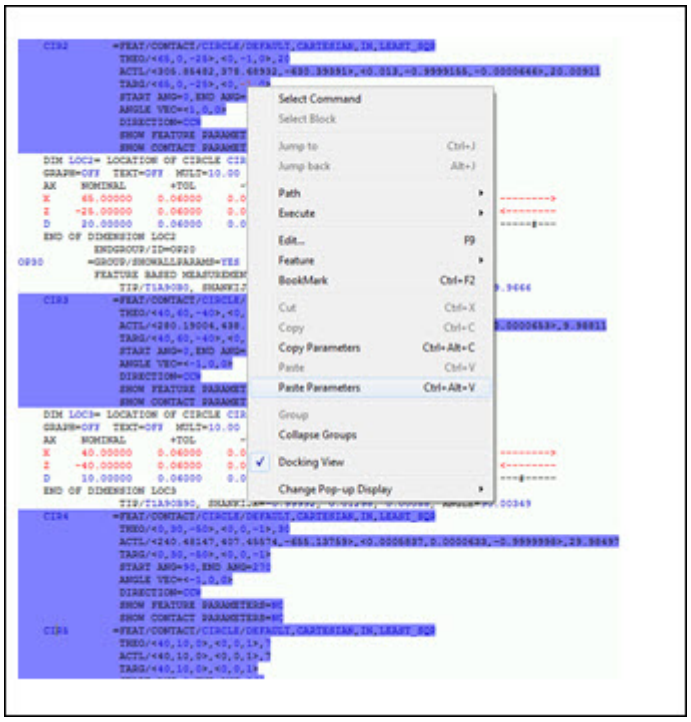
2. Use one of the following methods to copy the selected parameter(s):
    - Press Ctrl + Alt + C.
    - Right-click and select **Copy Parameters** on the shortcut menu.
    - Select **Edit | Copy Parameters**.

Pasting a Parameter

1. Use one of the following methods to select the feature to paste the copied parameter(s) to:
    - To select a single feature, click anywhere on the feature or dimension.
    - To select multiple, continuous features, click on the first feature and then drag your mouse while holding the mouse button. For example:

Command Mode

- To select multiple, non-continuous features, click anywhere on the feature, and then hold down the Ctrl key while clicking each non-continuous feature. For example:



Command Mode

2. Use one of the following methods to paste the selected parameter(s):

   - Press Ctrl + Alt + V.
   - Right-click and select **Paste Parameters** on the shortcut menu.
   - Select **Edit | Paste Parameters**.

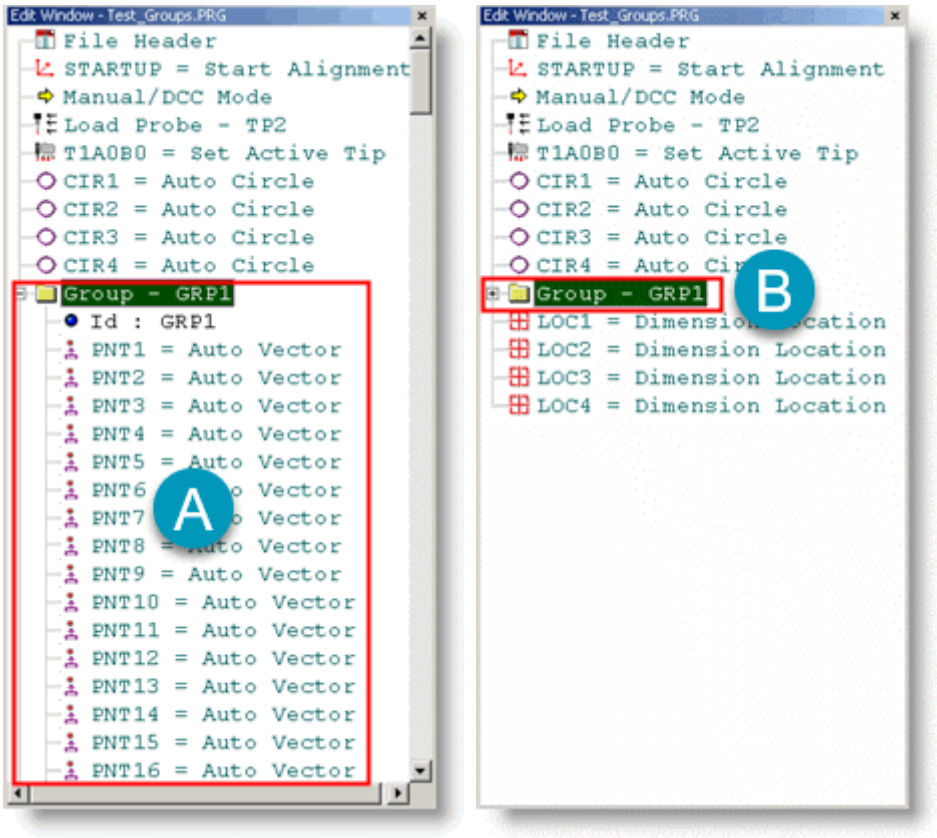
Working with User-Defined Groups

PC-DMIS lets you group contiguous Edit window commands together and place them inside of a user-defined `GROUP` / `ENDGROUP` command pair (in DMIS Mode, groups appear as DMISEW commands).

Grouped commands help simplify user interaction with a measurement routine by enabling you to apply operations on all commands inside the group, by applying them to the group itself. In addition, the measurement routine is visually less busy, enabling you to better visualize the overall structure of the measurement routine.

For example, suppose you have several hundred points in your measurement routine from a scan that you want to hide to make the overall structure of the measurement routine more visible and easier to work with. You can select all of the point commands as long as they are all in one list, and then select the **Insert | Group** menu item to group those commands together. In any mode of the Edit window, you can then show or hide the commands inside of the group as needed.

For example, in the images below, the left Edit window shows a group with the points expanded (A). The right Edit window shows a group with the points collapsed (B).



Groups appear in this format inside the Edit window in Command Mode:

```
GRP1 = GROUP/SHOWALLPARAMS = YES

    ... commands inside the group are listed
```

```
    ... between the two commands

ENDGROUP/
```

SHOWALLPARAMS = This toggle field let you show or hide commands inside of the group in Command Mode or DMIS Mode. By default, this command is set to YES, which shows all of the commands within the group. If you set it to NO, commands inside the group still exist, but are hidden.

Inserting Groups

When you insert your first group command into a measurement routine, PC-DMIS gives the command a default name of "GRP1" in Command Mode and DMIS Mode, and "GRP1 = GROUP" in Summary Mode. PC-DMIS then increments the number in the ID for each additional group. To easily modify the name of the group to something more descriptive, type a new ID in Command Mode or DMIS Mode, or press F9 and edit the GROUP command that way.

There are several different ways to insert groups into a measurement routine:

- **Insert | Group** menu item - This menu item inserts the `GROUP` and `ENDGROUP` commands. If you selected multiple commands before you selected this menu item, they would be placed inside of the group. In Command Mode and DMIS Mode, this means that the `GROUP` command would precede the very first selected command, and the `ENDGROUP` command would follow the last selected command. If you don't have any commands selected, PC-DMIS displays a dialog box that asks if you want to insert the command pair without any features inside it. This menu item works in any mode of the Edit window.
- Right-click shortcut menu -
  - Command Mode - Right-click inside Command Mode or DMIS Mode, and select **Group** from the shortcut menu. If you selected multiple commands before you selected this menu item, they would be placed inside of the group. In Command Mode and DMIS Mode, this means that the `GROUP` command precedes the very first selected command, and the `ENDGROUP` command follows the last selected command.
  - Summary Mode - Right-click inside Summary Mode, and select **Add Command**. From the list of commands that appears, select **Group**. This option inserts an empty `GROUP` / `ENDGROUP` command pair. Groups appear as folders inside Summary Mode. Commands appear indented inside those folders. To expand or collapse these groups, click the plus (+) or minus (-) buttons, respectively. The `ENDGROUP` is hidden in Summary Mode.
- Type the command - In Command Mode or DMIS Mode, type `GROUP` , and press Enter or Tab. PC-DMIS inserts an empty `GROUP` / `ENDGROUP` command pair.

PC-DMIS makes the **Group** menu item unavailable for selection if you try to group something that is not allowed, such as attempting to only group a portion of a command's block.

Feature Based Measurement

If your PC-DMIS license is programmed with the Feature Based Measurement (FBM) module, you will see an extended group command. For example:

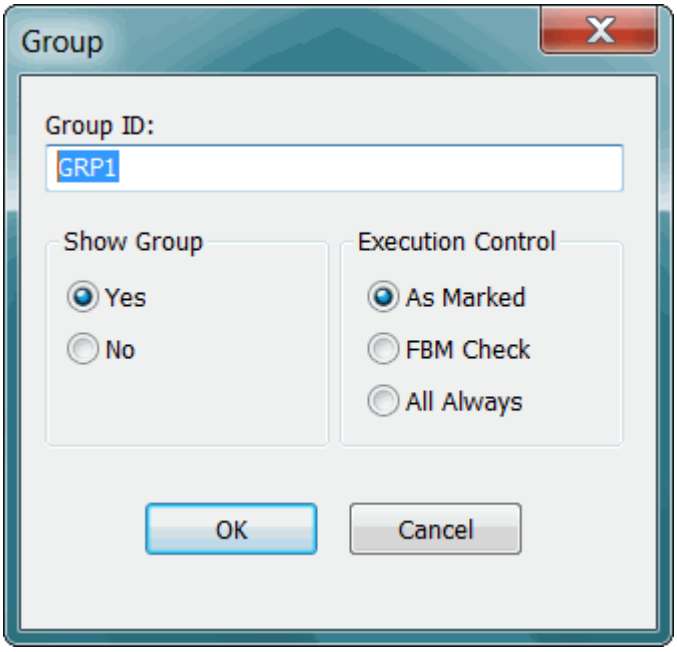
```
OP10          =GROUP/SHOWALLPARAMS=NO
               EXECUTION CONTROL=FBM CHECK
               ENDGROUP/ID=OP10
```

Example of the FBM extended group command.

PC-DMIS extends the `GROUP` command when you use the command with FBM. The group header displays an additional choice for execution control. For complete information, see "Mini Routines" in the PC-DMIS Core documentation.

Editing Groups

To edit a group, modify it directly in the Edit window. You can also press F9 on the `GROUP` command, or double-click on the command, and use the **Group** dialog box:



Group Dialog Box

- **Group ID** - Use this box to change the group's ID.
- **Show Group** area - If you select **Yes**, it sets the `SHOWALLPARAMS` entry to **TRUE** in the PC-DMIS Settings Editor. If you select **No**, it sets the `SHOWALLPARAMS` entry to **FALSE**.

- **Execution Control** area - This area of the dialog box is available only when the Feature Based Measurement (FBM) option is enabled on your PC-DMIS license. This area determines how PC-DMIS will execute the commands in a group. For more information, see "Mini Routines" in the PC-DMIS Core documentation.
    - **As Marked** - If you select **As Marked**, PC-DMIS executes all of the marked features. PC-DMIS skips all of the non-marked commands.
    - **FBM Check** - At the time of execution, PC-DMIS determines if a feature or dimension is marked in the group. If it finds a marked feature or dimension, it executes the entire group according to its marking status. If PC-DMIS determines that no feature or dimension is marked, it skips the entire group.
    - **All Always** - PC-DMIS always marks and executes the entire group and its nested subgroups. There may be several situations where the PC-DMIS application programmer would like to always measure certain features.

Removing Groups

Removing Only Group Commands

To remove the `GROUP` / `ENDGROUP` commands, *but retain the group's contents*, do one of the following:

- In the Edit window in Command Mode, ensure that all the items in the group are visible, select either the `GROUP` or `ENDGROUP` command (but not the entire group), and press Delete on your keyboard.
- In the Edit window in Summary Mode, expand the group, right-click on the `GROUP` command, and either select **Delete** from the shortcut menu or press Delete on your keyboard.

Removing both Group Commands and Contents

To remove the `GROUP` / `ENDGROUP` command *and its contents*, do one of the following:

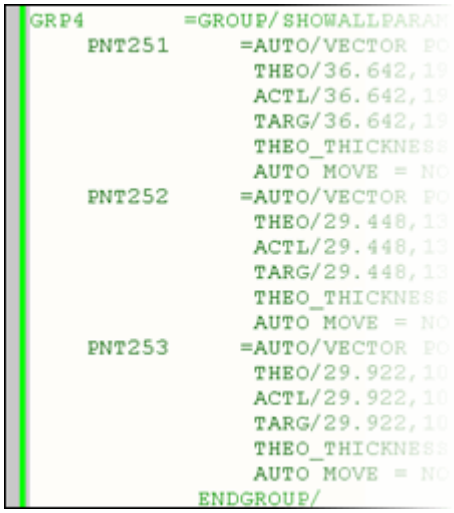
- In the Edit window in Command Mode, hide all of the features in the group, select the `GROUP` and `ENDGROUP` commands (ensure that you select both commands), and press Delete on your keyboard.
- In the Edit window in Summary Mode, collapse the group, right-click on the `GROUP` command, and either select **Delete** from the shortcut menu or press Delete on your keyboard.

If the items in the group are visible, you need to select the entire `GROUP` / `ENDGROUP` command block before you remove it.

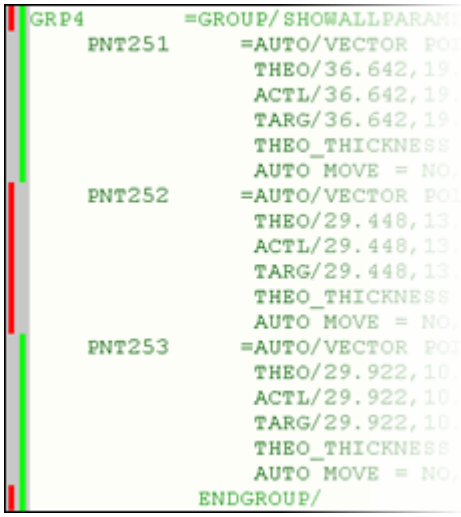
Using Multiple Arm Mode with Groups

In Command Mode or DMIS Mode, groups that contain commands set for execution by a specific arm will have the same red and/or green Multiple Arm Mode margin markings as those commands.

If all the commands in a group are tied to one arm, the `GROUP` command also has the same margin color markings. If you then change the arm mode of the `GROUP` command, all commands inside switch their margin color markings to the other arm.



If one or more commands are tied to both arms, the `GROUP` command also has the same margin color markings as both arms.



For information on Multiple Arm Mode, see the "Using Multiple Arm Mode" chapter.

Performing Operations on Groups

In general, all operations that can be performed on a single command in the Edit window, when performed on a group, are performed on every item contained within the group.

The various operations you can perform on a group are described in detail below. Most are taken from the Command Mode's shortcut menu. Right-click in the Command Mode to see this shortcut menu. This shortcut menu is discussed in the "Command Mode Shortcut Menu" topic in the "Using Shortcut Keys and Shortcut Menus" chapter. The information below only applies to how this menu works with `GROUP` commands or features inside of groups.

Shortcut Menu Operations on Groups

- **Select Command** - When the group is collapsed and you choose **Select Command**, PC-DMIS selects every command contained in the group as a block. When the group is expanded, **Select Command** selects only the `GROUP` command itself.
- **Select Block** - This menu item selects every command contained in the group as a block, regardless of the expanded/collapsed state of the group.
- **Execute From Cursor (Ctrl + U)** - This doesn't change from the usual operation.
- **Execute Block (Ctrl + L**) - If you select a block, PC-DMIS only executes the block. If you selected a group as the block, then PC-DMIS executes the group.
- **Jump To (Ctrl + J)** - This doesn't change from the usual operation.
- **Jump Back (Alt + J)** - This doesn't change from the usual operation.
- **Edit (F9)** - When the cursor is positioned on the GROUP command, a **Group** dialog box appears. You can modify the group's ID and display state. If you select **Edit** on the ENDGROUP command, nothing happens.
- **Mark (F3)** - If you place the cursor on the GROUP command and select this menu item, all items contained within the group are Marked or Unmarked as a whole. If you have any unmarked commands within the group when you select this menu item, those commands become marked, and all other commands remain marked. If all commands within the group are already marked when you select **Mark (F3)**, all commands become unmarked. The color of the GROUP command reflects the marked state of the commands contained within the group. If at least one command within the group is marked for execution, then the group displays as marked for execution. If no command is marked, then the group displays as unmarked.
- **Delete** - *This menu item only appears on the Summary Mode shortcut menu*. If you select a GROUP command and then select **Delete**, PC-DMIS deletes the GROUP and ENDGROUP commands and everything in between. You cannot undo a GROUP deletion.
- **Group** - If you select an existing GROUP command and then select the **Group** menu item, PC-DMIS creates a new GROUP command and nests the selected group inside it.
- **Collapse Groups** - This doesn't change from the usual operation.
- **Expand Groups** - This doesn't change from the usual operation.

- **Set as Start Point** - This doesn't change from the usual operation.
- **Cut** - Removes the GROUP command and ENDGROUP command. All items stored inside the group remain in the measurement routine.
- **Copy** and **Paste** - This copies and pastes a GROUP / ENDGROUP pair and everything inside the group.

Other Group Operations

- **File | Partial Execution | Execute Feature (Ctrl + E)** - Selecting this menu item while selecting the GROUP command executes all features inside the group.
- **Flow Control Statements** - The ID can be used to control the flow of measurement routine execution, similar to how you can send execution flow to a LABEL command. You can use GOTO or IF_GOTO flow control commands to send the execution flow to a group based on defined conditions. For example:

```
IF_GOTO/VAR > 0, GOTO = GRP1
```

or

```
GOTO/GRP1
```

Mini Routines

The topics in this chapter include:

[About Mini Routines]

[Using Mini Routines]

[Using Search ID box and Filter by list]

[Creating a Mini Routine]

[Creating a Sub Mini Routine]

[Adding Dimensions to Mini Routines]

[Shortcut Menus in the Mini Routine Dialog Box]

Using Group Property

[Creating a Measurement Routine Capable of Executing a Mini Routine]

Creating a Measurement Routine Capable of Measuring any Dimension Collision Free with ClearanceCube

Creating a Mini Routine using ClearanceCube: Key Concepts

Creating a Measurement Routine Capable of Measuring any Dimension Collision Free with FBM License

Creating a Mini Routine using FBM License: Key Concepts

Executing a Mini Routine or Sub Mini Routine

Examples of Creating Mini Routines and Sub Mini Routines

Important Notes

About Mini Routines

Mini routines enable you to measure a selected dimension or group of dimensions from a lengthy measurement routine.

With a mini routine, you can create a single measurement routine and measure the dimensions manufactured after every operation or only the dimensions that are affected by one or more tools or the critical dimensions. Mini routines also allow you to remeasure out-of-tolerance dimensions.

A mini routine provides you with the ability to create the structure to measure tool-based, cell-based, and operation-based features from your measurement routine.

Since mini routines can execute partial measurements in a measurement routine, it’s crucial to ensure there are no collisions when you execute a partial measurement routine. You can achieve this by using either of the following methods:

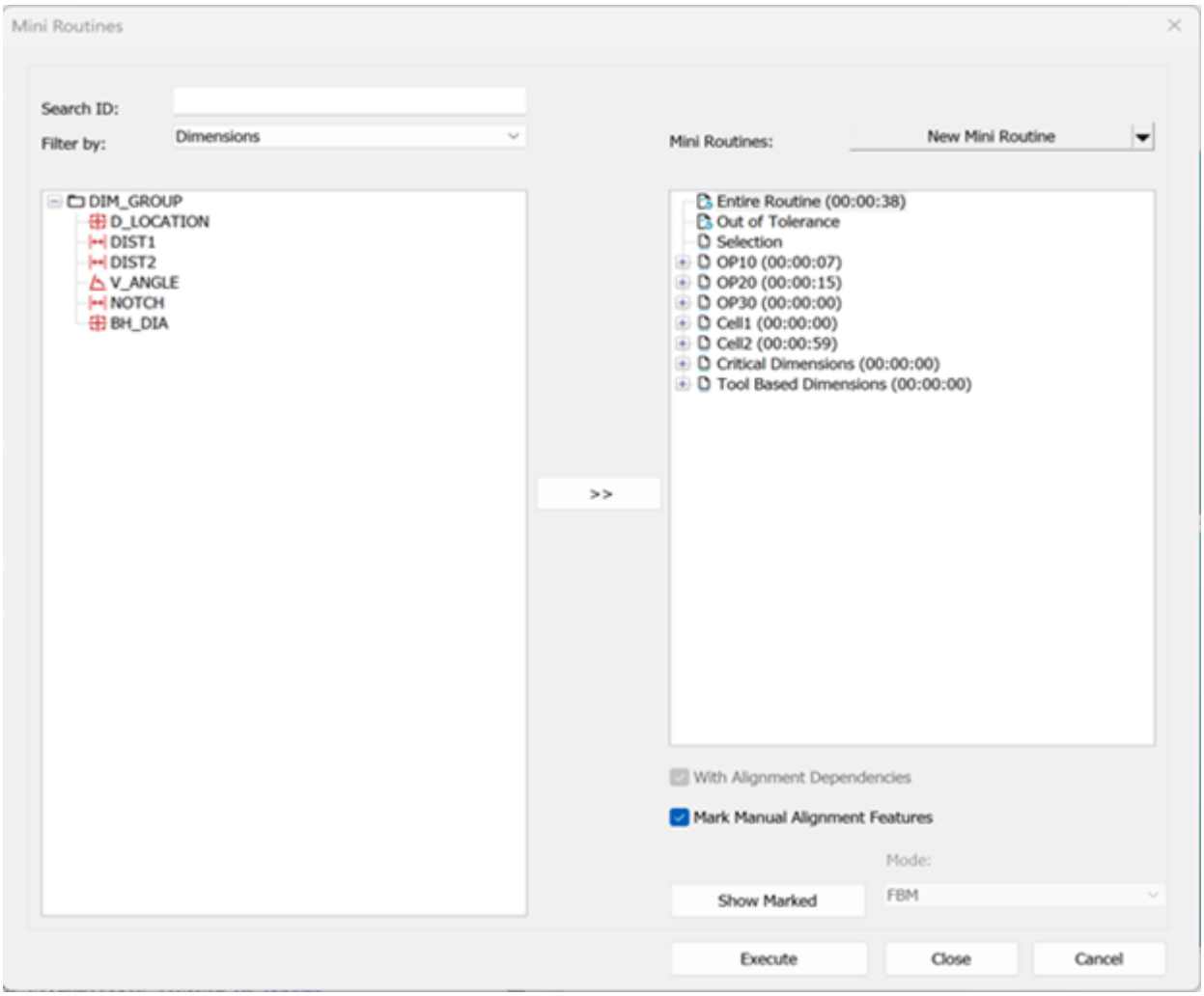
- Using ClearanceCube
- Using an FBM license

> The FBM license is an optional feature in PC-DMIS. FBM functionality is only available if your PC-DMIS license is programmed with the Feature Based Measurement module.

Both methods require you to use the `GROUP` command to create your measurement routine. For details, see "Creating a Measurement Routine Capable of Executing a Mini Routine" in the PC-DMIS Core documentation.

Using Mini Routines

To use mini routines, select the menu option **File | Partial Execution | Mini Routines** to open the **Mini Routines** dialog box.



Mini Routines Dialog Box

The left pane displays the dimensions available in the measurement routine, while the right pane displays the mini routines.

PC-DMIS automatically creates these mini routines:

- **Entire Routine** - Measures all the dimensions in the measurement routine.
- **Out of Tolerance** - Measures only the dimensions that are out of tolerance in the measurement routine.
- **Selection** - A "one-time" mini routine that measures items during the current session only. This mini routine will be empty when you re-open the **Mini Routines** dialog box.

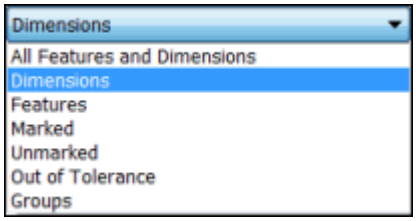
Using Search ID box and Filter by List

You can use filters to find the items you want to measure in your measurement routine. The filtered items are displayed in the Dimension pane.

1. To filter, do one of the following:

   - In the **Search ID** box, type the ID of the item that you want to find and then press Enter. You can also use meta characters in this box. For help, see "Selecting Features Using Meta-Character Matching" in the PC-DMIS Core documentation.

   - In the **Filter by** list, select a filtering option:



Filter by List

All Features and Dimensions - Displays the IDs of the features and dimensions.

Dimensions - Displays the IDs of the dimensions. This is the default.

Features - Displays the IDs of the features.

Marked - Displays the IDs of the features that are marked in the measurement routine.

Unmarked - Displays the IDs of the features that are not marked in the measurement routine.

Out of Tolerance - Displays the IDs of the features that are out of tolerance.

Groups - Displays the names of the groups. All dimensions that belong to a group are included in a mini routine.

2. Tooltips in the Dimension pane provide detailed information about dimensions. To display a tooltip, hover your mouse pointer over an item. The tooltip remains visible until you move the pointer off the item.

If you change the name of an item in the Edit window, PC-DMIS automatically changes its name in the filtered list as well as in the mini routine.

Creating a Mini Routine

To create your own mini routine:

1. In the **Mini Routines** list, click **New Mini Routine**. You can also right-click in the Mini Routine pane and select **New Mini Routine** on the shortcut menu. Click on the default name that appears and type the name of your choice.

2. To create another mini routine, click on the icon of any existing mini routine to select it and then select **New Mini Routine**. This places the mini routine at the root level of the tree structure.

Creating a Sub Mini Routine

You can create a sub mini routine for a mini routine that has no items.

1. Select the desired mini routine.

2. In the **Mini Routines** list, click **New Sub Mini Routine**. You can also right-click in the Mini Routine pane and select **New Sub Mini Routine** on the shortcut menu.

3. Click on the default name that appears and type a name of your choice.

Adding Dimensions to Mini Routines

1. In the filtered list in the Dimension pane, select each item that you want to add to a mini routine or sub mini routine.

   To display additional selection options, right-click in the pane. A shortcut menu appears:

   Filtering Shortcut Menu

   Select All - Selects all of the items in the filtered list.

   Unselect All - Deselects all of the items in the filtered list.

   Expand All - Displays all features and dimensions in all groups.

   Collapse All - Hides all features and dimensions in all groups.

2. With **Filter by** set to **Groups**, select **Expand All** and then click on each item in a group that you want to add to a mini routine or sub mini routine.

3. In the Mini Routine pane, click on the icon for the desired mini routine or sub mini routine to select it.
4. Click **>>** to move the selected items from the Dimension pane to the selected mini routine.
5. To save your changes to mini routines at any time, click **Close**. To discard your changes to mini routines, click **Cancel**. The **Mini Routines** dialog box closes.

When you select the **Close** button to close the dialog box, PC-DMIS automatically saves the mini routines as an .xml file with this naming convention:

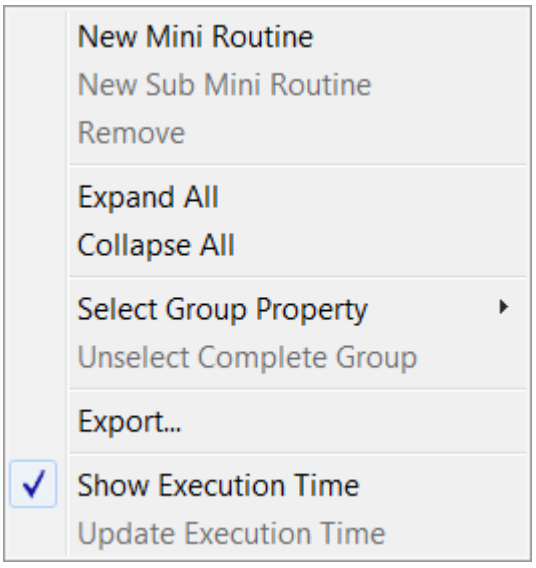
<name of measurement routine>.MiniRoutines.xml

This file is saved in the folder in which the measurement routine resides. When you open the measurement routine, the data in the .xml file displays in the **Mini Routines** dialog box.

If you send the measurement routine to another machine, you must include the *<name of measurement routine>*.MiniRoutines.xml file to see the mini routines on the other machine.

Shortcut Menu in the Mini Routine Dialog Box

If you right-click in the right pane of the Mini Routine dialog box, PC-DMIS displays a shortcut menu. The menu has these options:



Mini Routine Shortcut Menu

New Mini Routine - Creates a new mini routine.

New Sub Mini Routine - Creates a new sub mini routine.

Remove - Deletes the selected mini routine or sub mini routine.

Expand All - Displays all items in all mini routines and sub mini routines.

Collapse All - Hides all items in all mini routines and sub mini routines.

Select Group Property - Displays additional options:

- **Features and Dimensions Group** - This group property indicates that all features and dimensions in the group will be marked. The parents of all dimensions will also be marked.
- **Whole Group** - Marks all of the commands in the group regardless of the type of command. This includes commands that are not features and dimensions, or commands that do not get marked by dependency relations. This property is similar to the marked `GROUP` command header in PC-DMIS. For more information, see "Performing Operations on Groups".

Unselect Complete Group - Deselects the complete group and displays all of the dimensions in the group.

Export - Saves the tree structure for each mini routine in an .xml file, which you can use with automation. A dialog box appears and prompts you for the name and location of the file.

Show Execution Time - Displays the execution time in <hours>:<minutes>:<seconds> to the right of the **Entire Routine** option, the mini routine name, and the sub mini routine name.

Update Execution Time - This option is selected by default. If this option is selected, the execution time will be updated every time it is executed. This update function is available for the entire routine as well as for each separate mini routine. If you do not want the time to be updated after every execution, clear this option.

The **Update Execution Time** option is available when:

- The **Record and display execution time** check box on the **General** tab in the **Setup Options** dialog box is selected (for more information about the **Record and display execution time** check box, see "Execution Area").
- The **Entire Routine** option is selected.
- A mini routine or sub mini routine is selected.

Creating a Measurement Routine Capable of Executing a Mini Routine

PC-DMIS offers the following methods for collision-free measurement of any dimension in a measurement routine:

- Using ClearanceCube

- Using FBM License (Optional License)

> The FBM license is an optional feature in PC-DMIS. FBM functionality is available only if your PC-DMIS license includes the Feature Based Measurement module.

Creating a Measurement Routine Capable of Measuring any Dimension Collision Free with ClearanceCube

You can use the ClearanceCube and mini routine concepts to create measurement routines to provide a collision-free path for measuring any selected dimensions.

For information on how to use ClearanceCube, see the "Using the ClearanceCube" topic in the PC-DMIS Core documentation.

Creating a Mini Routine Using ClearanceCube: Key Concepts

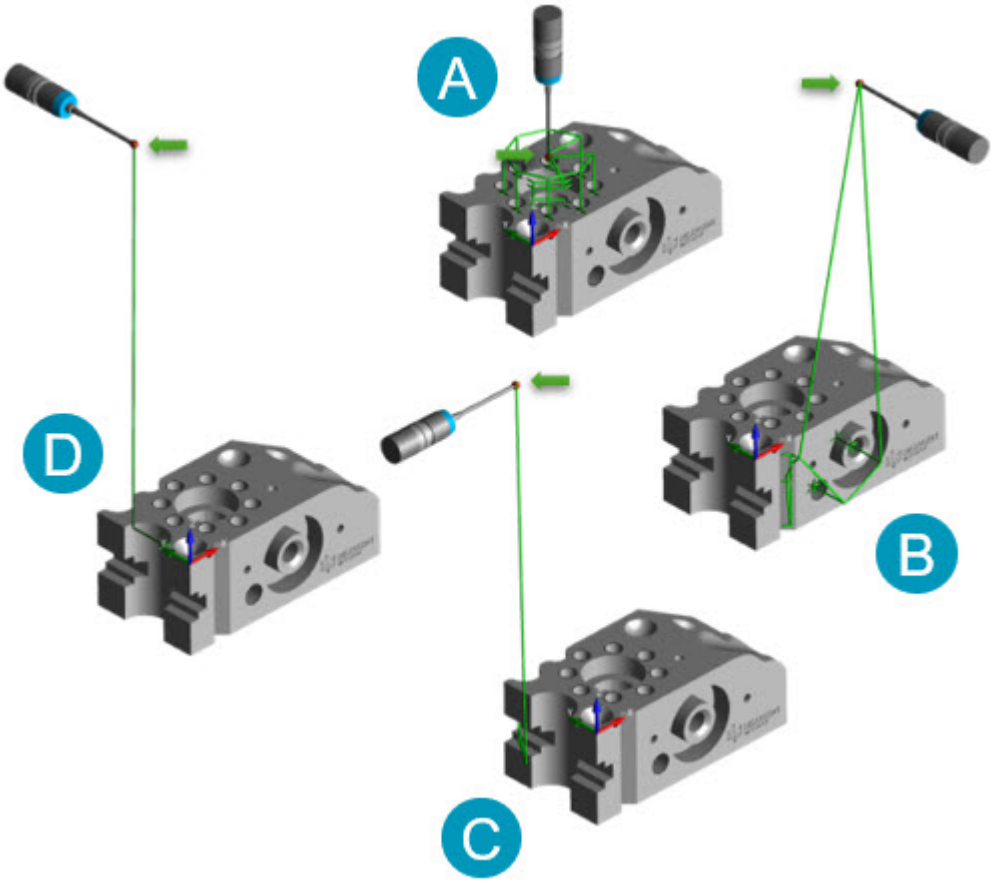
Organization of a Measurement Routine

- **Features and commands are measured always** - As a PC-DMIS application programmer, if you would like to always (with every execution) measure certain features then you must create them outside of a group.
- **Features and commands are measured only if required for selected dimensions** - As a PC-DMIS application programmer, if you would like to measure a feature only if the selected dimension requires it, you must create that feature inside a group.

Use Groups

Typically, CMM measures all features possible with one tip unless some features are required explicitly for alignment purposes. Features are measured logically based on the use of the tip or face of the part. Continue to create routines in your usual style, and, as an additional step, group the features measured in this logical manner.

The following image explains the concept.



A. *ZPLUS Group*

B. *YMINUS Group*

C. *XMINUS Group*

D. *YPLUS Group*

For information about how to insert groups into a measurement routine, see "Inserting Groups" in the PC-DMIS Core documentation.

Using a MOVESET Command

Each group can measure features on one side of a part. You can use a `MOVESET` command at the group's beginning and end.

When you insert the `MOVESET` command at the beginning and end of a group, it is like how a car enters and exits a freeway. After entering the freeway, a car can move at high speed but needs to avoid accidents. Similarly, after executing the beginning Moveset, the tip can move freely and avoid collisions by using avoidance moves.

For the Moveset at the beginning of the group, assign the ClearanceCube motion to the starting face and turn off the ClearanceCube motion at the ending face.

Similarly, for the Moveset at the end of the group, turn off the ClearanceCube motion at the starting face and assign the ClearanceCube motion to the ending face.

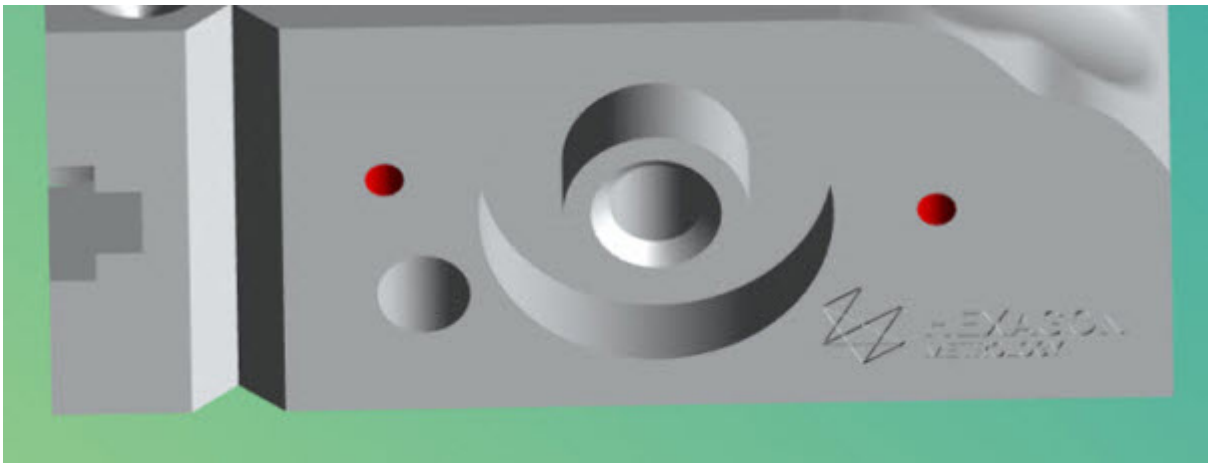
Now, you can use any method in PC-DMIS to measure features on this side (face) of the part. Use the Avoidance Move to navigate between features. You do not need to activate the ClearanceCube motion for these features unless required for a specific reason.

Alternatively, you can activate ClearanceCube for all features in a group for parts with simple geometry instead of using `MOVESET` commands.

Using Nested Groups

You can use nested groups where additional move commands are required to safely measure features without collision.

For example, you can create a nested group to measure the planes on the V groove. This nested group includes entry and exit `MOVESET` commands. Use the `MOVESET` command inside the nested group to position the tip in front of the V groove and ensure collision-free measurement of the planes within the groove.



Example showing nested groups.

Datum Assignments

If you assign datum feature reference letters to the datum features, that assignment should be within the group that measures the features.

Dimensions

You can create dimensions in the group where they are measured or you create them in a separate group. You can also group the dimensions into operation, cell, or manufacturing tool categories. This helps you select a whole group of dimensions property in a mini routine.

Create Alignment to Constrain 6 Degrees of Freedom

In PC-DMIS, you can base alignments on the recall of a previous alignment. If you create alignment in this way, the mini routine looks at recalled alignment dependencies and marks everything required to create the recalled alignment.

If an alignment constrains all 6 Degrees of Freedom (DOF), we recommend you recall the STARTUP alignment rather than the previous one. This breaks the dependency chain, and PC-DMIS only measures those features.

Using LOADPROBE and TIP Commands

If your measurement routine requires a tool or tip change, define this within the group where it is needed. Ensure that you select the necessary probe and tip in each group.

Using Variables

Create variables outside of the group if you need input from the user. Create other variables within the group if you want to use that variable for measuring a feature or dimension.

Commands Outside of Groups

When used with ClearanceCube mode, a mini routine always marks and executes every command that is not part of any group. You must place anything that must be executed (tracefields, operator responses, and generating strings for output file names) outside of a group.

Creating a Measurement Routine Capable of Measuring any Dimension Collision Free with FBM License

You can use FBM license and mini-routine concepts to create measurement routines to provide a collision-free path for measuring any selected dimension.

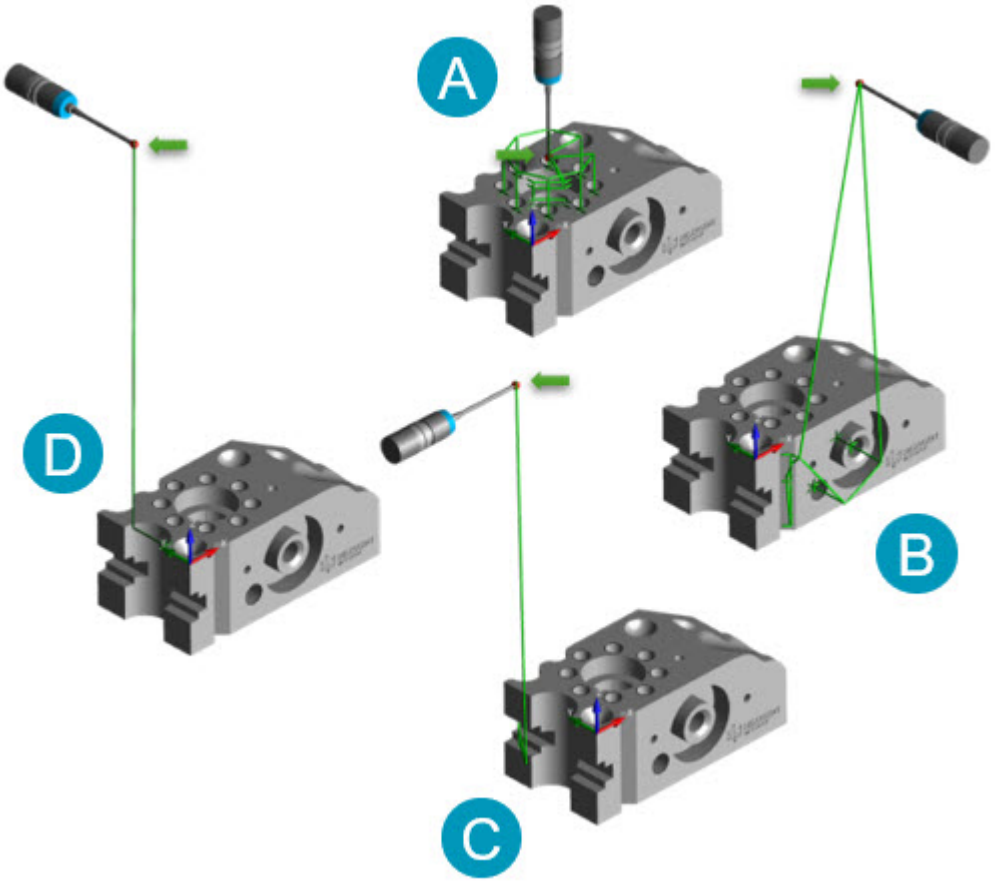
Use the FBM license when the ClearanceCube method does not support the complexity of your requirement or the part’s geometry.

Creating a Mini Routine using FBM License: Key Concepts

Use Groups

Typically, CMM measures all features possible with one tip unless some features are required explicitly for alignment purposes. Features are measured logically based on the use of the tip or face of the part. Continue to create routines in your usual style, and, as an additional step, group the features measured in this logical manner.

The following image explains the concept.



A. *ZPLUS Group*

B. *YMINUS Group*

C. *XMINUS Group*

D. *YPLUS Group*

A mini routine with an FBM license is tightly coupled with the `GROUP` command. The following is an example of a measurement routine where all measured features and dimensions are organized into groups.

```
STARTUP    =ALIGNMENT/START,RECALL:USE_PART_SETUP,LIST
            ALIGNMENT/END
            MODE/MANUAL
            PREHIT/3
            RETRACT/3
            MOVESPEED/ 150
            TOUCHSPEED/ 5
            FORMAT/TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;M
            LOADPROBE/LSPX3BY50
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
            RECALL/ALIGNMENT,EXTERNAL,MANUAL FIXTURE
            MODE/DCC
START_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=ALL ALWAYS
            ENDGROUP/ID=START_GROUP
ALIGN_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=ALIGN_GROUP
ZPLUS_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=ZPLUS_GROUP
YMINUS_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=YMINUS_GROUP
XMINUS_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=XMINUS_GROUP
YPLUS_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=YPLUS_GROUP
CONST_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=AS MARKED
            ENDGROUP/ID=CONST_GROUP
DIM_GROUP  =GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=AS MARKED
            ENDGROUP/ID=DIM_GROUP
VIEWSET1   =VIEWSET/
                              END OF MEASUREMENT FOR
```

Example of a measurement routine with groups

Group Command Concepts

PC-DMIS extends the `GROUP` command when you use the command with FBM enabled in the license.

```
START_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=ALL ALWAYS
            ENDGROUP/ID=START_GROUP
ALIGN_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=AS MARKED
            ENDGROUP/ID=ALIGN_GROUP
ZPLUS_GROUP=GROUP/SHOWALLPARAMS=NO
            EXECUTION CONTROL=FBM CHECK
            ENDGROUP/ID=ZPLUS_GROUP
```

Example of group commands with FBM

The group header displays an additional choice for execution control. Three options are available in the group control:

- **ALL ALWAYS** – The mini routine always marks and executes the entire group and its nested subgroups. There may be several situations where the PC-DMIS application programmer wants to always measure certain features.

- **AS MARKED** - The mini routine marking method with the FBM license marks the commands in the measurement routine to measure the selected dimensions. All parent features of every dimension are selected, and any required alignment is marked. If you select **AS MARKED**, the mini routine executes all marked features. Mini routine skips all the non-marked commands.
- **FBM CHECK** - At the time of execution, PC-DMIS determines if a feature or dimension is marked in the group. If it finds a marked feature or dimension, it executes the entire group according to its marking status. If PC-DMIS determines that no feature or dimension is marked, it skips the whole group.

For information about how to insert groups into a PC-DMIS measurement routine, see "Inserting Groups" in the PC-DMIS Core documentation.

Using a Move Point Command

Each group can measure features on a given face of a part. There may be move point(s) commands at the beginning of the group. These move points position the tip on top of the face of interest. You can use any method in PC-DMIS to measure all features on this face.

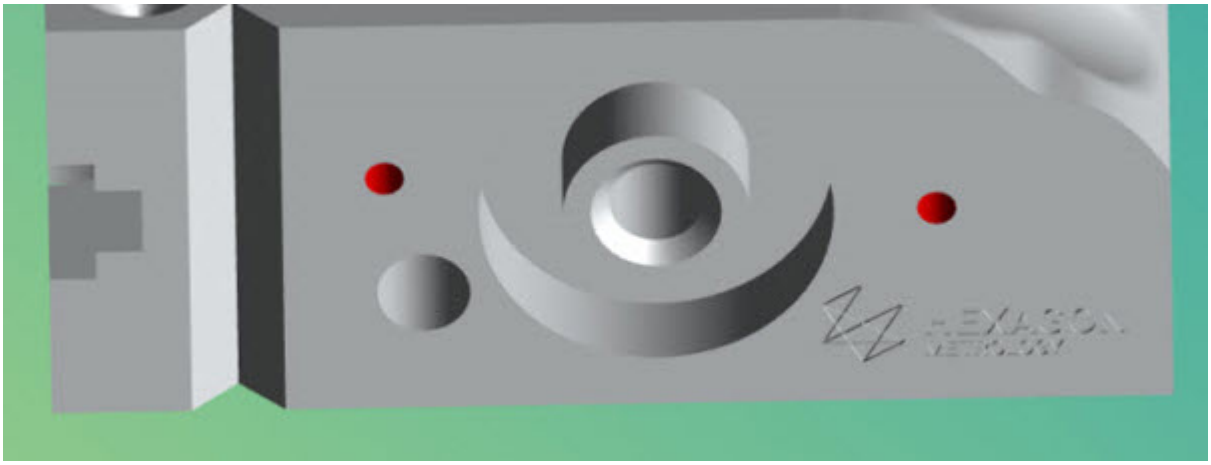
When all measurements are complete, the probe safely moves back to a position using move point(s). It can then safely move to any other group.

When you insert the `MOVE/POINT` command at the beginning and end of a group, it is like how a car enters and exits a freeway. After entering the freeway, a car moves at high speed but needs to avoid accidents. Similarly, after entering the move point, the tip uses avoidance moves so it can move freely and avoid collisions.

Using Nested Groups

You can use nested groups where additional move points are required to measure features without collision.

For example, you can create a nested group to measure the planes on the V groove. This nested group includes entry and exit move commands in front of the V groove. This ensures collision-free measurement of the planes within the groove.



Example of nested groups

Datum Assignments

If you assign datum feature reference letters to the datum features, that assignment should be within the group that measures the features.

Dimensions

You can create dimensions in the group where they are measured or you create them in a separate group. You can also group the dimensions into operation, cell, or manufacturing tool categories. This helps you select a whole group of dimensions property in a mini routine.

Create Alignment to Constrain 6 Degrees of Freedom

In PC-DMIS, you can base alignments on the recall of a previous alignment. If you create alignment in this way, the mini routine looks at recalled alignment dependencies and marks everything required to create the recalled alignment.

If an alignment constrains all 6 Degrees of Freedom (DOF), we recommend you recall the STARTUP alignment rather than the previous one. This breaks the dependency chain, and PC-DMIS only measures those features.

Using LOADPROBE and TIP Commands

If your measurement routine requires a tool or tip change, define this within the group where it is needed. Ensure that you select the necessary probe and tip in each group.

Using Variables

Create variables in a group with execution control set as “All Always” if you need to get input from the user.

Create other types of variables in the group in which you want to use that variable for measuring a feature or a dimension.

Commands Required to be Executed Always

A mini routine always marks and executes every command that is in a group with execution control set to ALL ALWAYS. You must place anything that must be executed (tracefields, operator responses, and generating strings for output file names) inside of a group.

Commands Outside of Groups

Any command that exists outside of groups has a global scope. PC-DMIS always executes these commands according to their marked status.

Executing a Mini Routine or Sub Mini Routine

1. Select **File | Partial Execution | Mini Routines** to open the **Mini Routines** dialog box.

2. Click the icon for the mini routine or sub-mini routine to select it.

3. If you want to measure the mini routine with alignment dependencies, select the **With Alignment Dependencies** check box. To measure it without alignment dependencies, clear the check box.

   The availability of the **With Alignment Dependencies** check box depends on the `MiniRoutineWithAlignDependencyEnabled` setting in the Settings Editor application.

   - If the entry is enabled, the **With Alignment Dependencies** check box is available.
   - If the entry is disabled, the check box is dimmed and unavailable to prevent unauthorized or unintentional use.

   "Alignment dependencies" means that the features required to create an alignment for the selected items will be marked and measured. For example:

   - Suppose that PC-DMIS has measured the part. The part has not moved yet, and you want to remeasure certain features. You can measure the mini routines and sub mini routines without measuring the alignment features again.
   - Suppose that PC-DMIS has measured the part. You then notice that one hole is dirty, and you need to measure only the dimensions associated with that hole. You can measure the dimensions with alignment dependency because the part has moved and needs to be realigned.

4. To mark manual alignment features for measurement, click **Mark Manual Alignment Features**.
5. To mark the features and dimensions to be measured, select the desired mini routine or sub mini routine and click **Show Marked**. PC-DMIS marks the features and dimensions in the Edit window.
6. In the **Mode** list, select **ClearanceCube** or **FBM**, depending on whether you created the measurement routine with ClearanceCube or FBM license concept. If your license does not include the FBM option, ClearanceCube is the only available choice.
7. Click **Execute**. PC-DMIS closes the **Mini Routines** dialog box and displays the **Execution** dialog box.

Marking Method for Routines using ClearanceCube

When using ClearanceCube, a mini routine marks the selected dimensions and dependent commands through the following steps:

1. The mini routine marks the complete measurement routine.

2. The mini routine unmarks all groups or all commands in each group.
3. The mini routine marks the selected dimensions with parent mode dependency. This includes marking all features on which they depend.
4. The mini routine identifies each group where one or more features are marked. It then marks all the commands in that group that are not features or dimensions. For example, load probe, tip, recall of alignment, prehit, retract, comments, move, moveset, move speed, display precision, format, mode, etc.

Marking Method for Routines using FBM License

When you use the FBM license, a mini routine marks the selected dimensions and dependent commands through the following steps:

1. The mini routine marks the complete measurement routine.
2. The mini routine unmarks all features and dimensions.
3. The mini routine marks the selected dimensions with parent mode dependency. This includes marking all features on which they depend.
4. The mini routine marks all commands of groups that have the execution control set to **ALL ALWAYS**.
5. When PC-DMIS executes a measurement routine with one or more groups with the execution control set to **FBM CHECK**, PC-DMIS determines if any features or dimensions in the group need to be measured.
    - PC-DMIS does not execute the entire group if a group has no marked features. It does not execute any other commands in the group (such as move points and tip change commands), whether they are marked or not.
    - If a group has marked features, PC-DMIS executes the group, as marked by the mini routine.
6. For groups with the execution control set to **AS MARKED**, PC-DMIS fully executes the group according to the normal marking rule.

Example of Creating Mini Routines and Sub Mini Routines

This example creates mini routines and sub mini routines for a part. The part is manufactured in two operations: Operation 10 and Operation 20. Operation 10 manufactures the part in Station 1 and Station 2. This example assumes that a measurement routine has been created to measure the entire part.

1. In the Mini Routine pane, find the items to measure in the measurement routine.
2. Create the mini routines and sub mini routines:

a. Click **New Mini Routine**. **MiniRoutine1** appears.

b. Click **New Mini Routine** again. **MiniRoutine2** appears.

c. Rename **MiniRoutine1** to **Op10**, and **MiniRoutine2** to **Op20**.

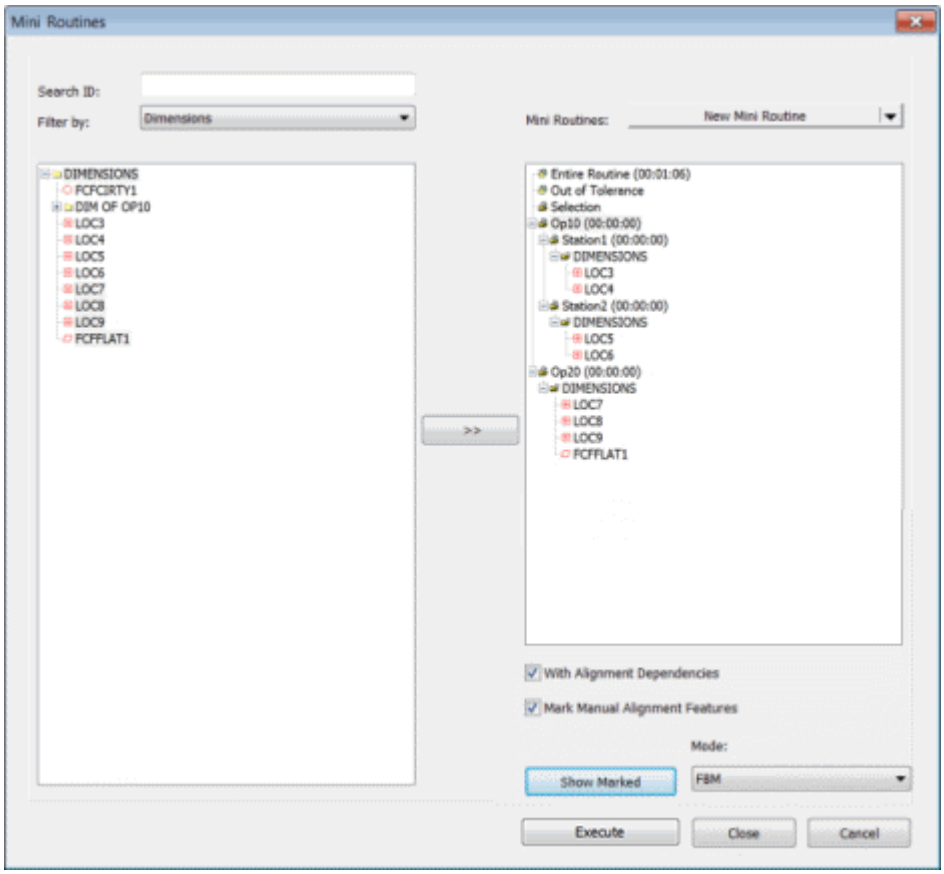
d. Select **Op10** and then click **New Sub Mini Routine**. **Op101** appears.

e. Select **Op10** again and then click **New Sub Mini Routine**. **Op102** appears.

f. Rename **Op101** to **Station 1**, and **Op102** to **Station 2**.

3. In the Dimension pane, select the items to measure.

4. Move the selected items to the **Station 1** and **Station 2** sub mini routines, and the **Op20** mini routine.



Example of mini routines and sub mini routines

Station 1 has the dimensions of the features manufactured by Station 1. **Station 2** has the dimensions of the features manufactured by Station 2.

Click **Close**. PC-DMIS saves the data as a .xml file in the measurement routine's folder.

5. To execute the **Op10** mini routine, select it and then click **Execute**. PC-DMIS measures the dimensions of the features manufactured in Operation 10.

6. To execute the **Op20** mini routine, select it and then click **Execute**. PC-DMIS measures the dimensions of the features manufactured in Operation 20.

Important Notes

- Mini routines provide a way to measure selected dimensions and features with or without alignment dependency.
- Recall the necessary alignment at the beginning of every group to ensure that PC-DMIS uses the correct alignment to measure all features. This functionality is provided for specific industries that may want to quickly re-measure certain dimensions after measuring the complete part without moving it from its fixture.
- Any command outside of all groups (that is, it does not belong to a group) has a global scope. PC-DMIS always executes these commands according to their marked status.
- Nested groups follow the same execution rules. If an inner group contains a measurement command, PC-DMIS also executes its outer group.

  For example, suppose you have GROUP A and GROUP B. GROUP B is nested inside GROUP A. If execution control is set to **FBM CHECK** for both GROUP commands, but only GROUP B has features or a marked dimension, GROUP A also executes its marked commands.
- If you delete a dimension and recreate it, remember to edit the mini routine to include it.
- If you change the ID of a dimension, PC-DMIS automatically updates it in the mini routine.

Using Other Windows, Editors, and Tools

Using Other Windows, Editors, and Tools: Overview

PC-DMIS provides you with an assortment of windows, editors, and other aids that can help make your measurement routine more productive.

These include:

- The Edit window, which has already been discussed in the "Using the Edit Window" chapter
- A Report window to display measurement results (see "Using the Report Window")
- The PC-DMIS FUSION application brings all your measurement data in one place (see "Using PC-DMIS FUSION")
- A script editor that creates scripts in the BASIC programming language (see "Using the Basic Script Editor")
- An editor that creates and displays interactive forms and dialog boxes (see "Using the Form Editor")
- An inspection report editor that enables you to quickly view and make small editing changes to your automatically-generated inspection reports (see "Viewing an Inspection Report")
- An interface for quick generation of simple measurement routines (see "Using the Quick Start Interface")
- A Settings window that you can use to make quick edits to frequently-used values (see "Using the Settings Window")
- A Preview window that previews your measurements before you accept them (see "Using the Preview Window")
- A window that creates and stores marked sets of features for future execution (see "Using the Marked Sets Window")
- A Probe Readouts window that shows the current probe's location and other information (see "Using the Probe Readouts Window")

- A Status window that displays the current status of an operation or feature information (see "Using the Status Window")
- A Probe Toolbox that performs probe manipulations (see "Using the Probe Toolbox")
- A dockable color bar that shows the different tolerance zones and dimension colors (see "Using the Dimension Colors Window")
- A dockable Q-DAS Data Editor window that modifies K-field data for their Q-DAS statistical results (see "Using the Q-DAS Data Editor Window")
- A dialog box that you can use to select an active tip (see "Selecting a Tip")
- A path viewer that shows the path your probe takes during measurement routine execution (see "Viewing Path Lines")
- A **CAD Information** dialog box that displays information about a CAD element in the Graphic Display window (see "Viewing CAD Information" in the "Editing the CAD Display" chapter)
- A **CAD Import History** dialog box that displays a history of CAD models imported into the current measurement routine (see "Viewing CAD Import History")

Using the Report Window

Select the **View | Report Window** menu option to display the Report Window. This window, after measurement routine execution, displays your measurement results and automatically configures the output according to a default report template. For detailed information, see the "About the Report Window" topic in the "Reporting Measurement Results" chapter.

Using PC-DMIS FUSION

PC-DMIS FUSION brings all your metrology data together so you can:

- Review PC-DMIS results and reports
- See sample-level details over time
- Explore SPC trends
- View part information in a central dashboard

If you have PC-DMIS FUSION installed, you can choose **View | PC-DMIS FUSION Reporting** to open the latest report data from your current measurement routine in the PC-DMIS FUSION application. This starts PC-DMIS FUSION if it's not already running.

For more information about PC-DMIS FUSION, see the **Welcome** tab of the FUSION view in the PC-DMIS Home page.

Using the Basic Script Editor

The Basic Script Editor can be used to create and edit BASIC scripts that can be used in Basic Script objects during execution or from the **Basic Script Editor** toolbar.

The **View | Basic Script Editor** menu option opens the Basic Script Editor and replaces PC-DMIS's main menu bar with these menus: **File**, **Edit**, **Run**, and **Help**. You can restore PC-DMIS's normal menu bar by minimizing or closing the Basic Script Editor.

The Basic Script Editor consists of the following:

- **Basic Script Editor** toolbar
- **File** menu
- **Edit** menu
- **Run** menu
- **Help** menu

These topics are discussed below.

Basic Script Editor Toolbar

The **Basic Script Editor** toolbar supports the following functions:

New

This button allows you to create a new basic script in the editor.

Open

This button displays the **Open File** dialog box used to open an existing basic script into the editor.

Save

This button saves the current basic script. If you have not already named the current script, a **Save As** dialog box asking for the name of the script will appear.

Print

This button prints the current basic script.

Print Preview

This button allows you to see the current basic script in the Print Preview window as it appears when printed.

Find

This button allows you to search for text in the current basic script.

Cut

This button cuts currently selected text and put text on the clipboard.

Copy

This button copies currently selected text and put text on the clipboard.

Paste

This button pastes text from the clipboard into the editor at the current insertion point.

Undo

This button allows you to undo the last editing change.

Compile

The **Compile** icon compiles (makes the script understandable and ready to run on the computer system) the current BASIC script. You must compile a script before running it.

Start

This button compiles and runs the current basic script.

Scripts run from the editor using the PC-DMIS basic commands can insert objects into the current measurement routine.

File Menu

The Basic Script Editor's **File** menu gives you the following commands and options:

New

The **File | New** menu option opens a new Basic Script Editor in which you can write a new script.

Open

The **File | Open** menu option allows you to navigate to and open an existing script. In order for files to appear in the Basic Script Editor, files must be of file type *.bas.

Save

The **File | Save** menu option is used to save a script. With a new script, the first time you select this option, the **Save As** dialog box appears, and you can use it to choose what to name your script and where you want to save it.

Save As

The **File | Save As** menu option saves a new script, or an already existing script with a new file name. When you choose it, the **Save As** dialog box appears for you to enter the file name and select the folder where you want to save the script.

Print

The **File | Print** menu option allows you to print the script in the Basic Script Editor from your system's printer.

Print Preview

The **File | Print Preview** menu option allows you to preview what PC-DMIS sends to the printer when you choose **Print** from the Basic Script Editor's **File** menu.

Exit

The **File | Exit** menu option allows you to exit out of the Basic Script Editor without saving any changes you have made to any open scripts. Choosing **File | Exit** will return you the main user interface. The menu bar will return to normal PC-DMIS functions.

Unicode

The **File | Unicode** menu item specifies whether your basic script is a Unicode script or not. If it isn't a Unicode script, then the Basic Script Editor interprets the script as ASCII text.

The Basic Script Editor needs to know the format of your script in order to correctly display and interpret it. Unicode formatting allows the editor to handle more complex characters (such as those displayed in Chinese or Japanese).

Unless you're working in a language that uses multi-byte characters, you generally won't need to select this menu item.

Edit Menu

The **Edit** menu of the Basic Script Editor allows you to use basic Edit functions to manipulate the text displayed in the Basic Script Editor.

Undo

The **Edit | Undo** menu option allows you to undo the most recent action taken in the Basic Script Editor.

Cut

The **Edit | Cut** menu option allows you to cut selected text from the Basic Script Editor. Cut text is stored in the Windows clipboard to later be pasted elsewhere.

Copy

The **Edit | Copy** menu option allows you to copy selected text. Copied text is stored in the Windows clipboard to later be pasted elsewhere.

Paste

The **Edit | Paste** command allows you to paste text that is stored in the Windows clipboard.

Delete

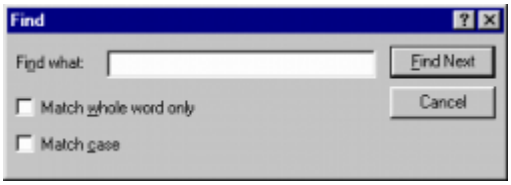
The **Edit | Delete** command allows you to delete highlighted text.

Select All

The **Edit | Select All** menu option automatically selects all the text within the Basic Script Editor. You can then **Cut**, **Copy**, or **Delete** the selected text.

Find

The **Edit | Find** menu option brings up the **Find** dialog box.



Find Dialog Box

This dialog box allows you to search for a specific word, or term within the Basic Script Editor.

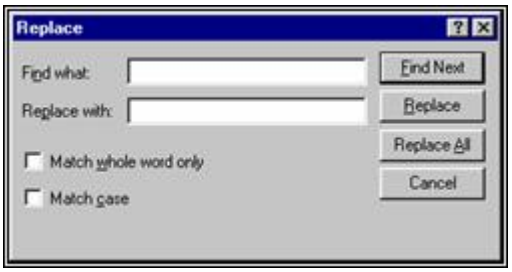
- If you choose the **Match Whole Word Only** check box, the dialog displays only those words that match the entire word.
- If you choose the **Match Case** check box, the dialog displays only those terms that match the case (Uppercase or Lowercase) that you used in the **Find what** box.

Find Next

The **Edit | Find Next** searches in the Basic Script Editor for the next term that meets the qualifications specified in the **Find** dialog box (See **Edit | Find** topic for details.)

Replace

The **Edit | Replace** menu option displays the **Replace** dialog box.



Replace Dialog Box

The **Replace** dialog box is an extension of the **Edit | Find** command. It is used to search for a specific term and then replace it with the term entered in the **Replace with** box.

The **Match whole word only** check box only matches whole words, not partial matches. For example, if you are looking for "point" and you do not have this check box selected, it may find the sequence of characters inside the words "points" or "pointer".

The **Match case check box** only finds instances that match the exact case typed into the **Find what** box. If you typed "point", it would not find "Point" or "POINT" because of their case differences.

The **Find Next** button searches through the Basic Script Editor and displays the first instance that meets the qualifications entered in the dialog box.

The **Replace** button is used to replace what has been found (using the **Find Next** button) with what is in the **Replace with** box.

The **Replace All** button is used to replace all instances in the Basic Script Editor that meet the search qualifications with what is in the **Replace with** box.

The **Cancel** button closes the **Replace** dialog box.

Dialog Editor

The **Edit | Dialog Editor** option opens a window with a grid named **Dialog One** and also the **MasQ Enable Dialog Designer** toolbar. This toolbar and the **Dialog One** grid provide the necessary tools to design dialog boxes that can later be programmed for your scripts.

Clicking the "X" in the upper right-hand corner of the **MasQ Enable Dialog Designer** toolbar closes these tools.

View Menu

The **View** menu allows you to choose if the **Basic Script Editor** toolbar and Status Bar are being displayed.

- Select **View | Toolbars** to display or hide the various toolbars.
- Select **View | Status Bar** to display or hide the Status bar.

With this menu, you can also determine where to set tab stops. This option allows you to indent BASIC language program statements a set number of characters, to help to improve the script's readability. To do this, select **View | Set Tab Stops** and type a number. PC-DMIS then indents by the indicated number of characters whenever you press the Tab key.

For example, if you want a tab stop at every five characters, type "5" in the **Set Tab Stops** dialog box.

You can use the `TabStops` entry in the PC-DMIS Settings Editor to set the number of characters that appear in the **Set Tab Stops** dialog box.

Run Menu

The **Run** menu allows you to select the **Compile** or **Start** command. The compile command compiles the script—checking it for syntax errors—while the start command executes the script.

Help Menu

The **Help** menu provides a **Basic Help** menu option. This option shows commands from the BASIC language that you can use with the Basic Script Editor.

Using the Form Editor

You can select **File | Reporting | New | Form Report** to open the Form Editor. This editor provides you with powerful tools to aid you in building interactive forms and dialog boxes that get activated during execution. For detailed information, see the "Creating Forms" topic in the "Reporting Measurement Results" chapter.

Viewing an Inspection Report

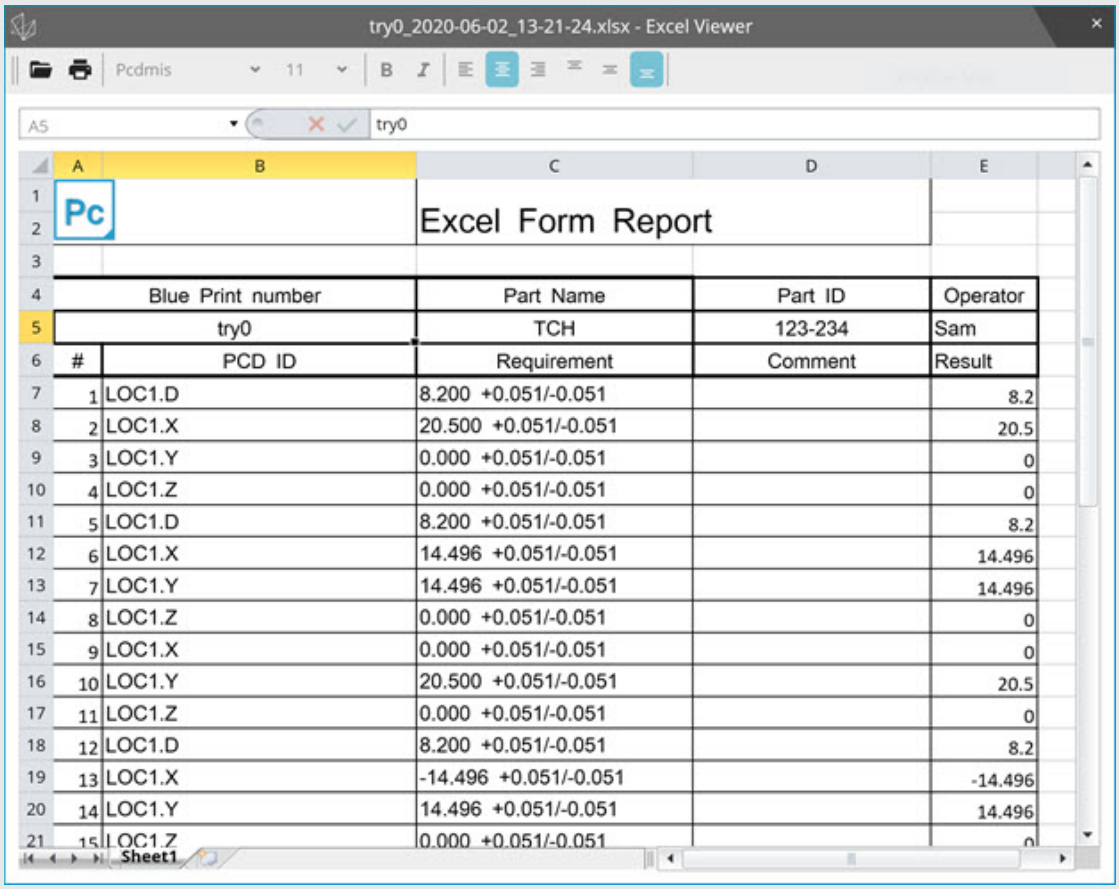
You can select **View | Inspection Report** to open an inspection report that was previously saved in one of these file types:

- .rtf
- .pdf
- .xls
- .xlsx
- .csv

To open an inspection report:

1. Select **View | Inspection Report** to display the **Open** dialog box.
2. From the **Files of type** list, select the file type that filters the current folder.
3. From the **Open** dialog box, navigate to the folder that contains the report file. Then select the file.
4. Click **Open**. PC-DMIS uses the default software installed on your computer to open the selected file.

If you select **Excel Files (*.XLS;*.XLSX;*.CSV;)**, and you want to use the internal Excel viewer that comes with PC-DMIS, select the **Use internal Excel viewer** check box. PC-DMIS shows the report in an internal Excel viewer.



Excel Viewer

Open ( ) - This icon displays the **Open** dialog box that enables you to open an Excel report.

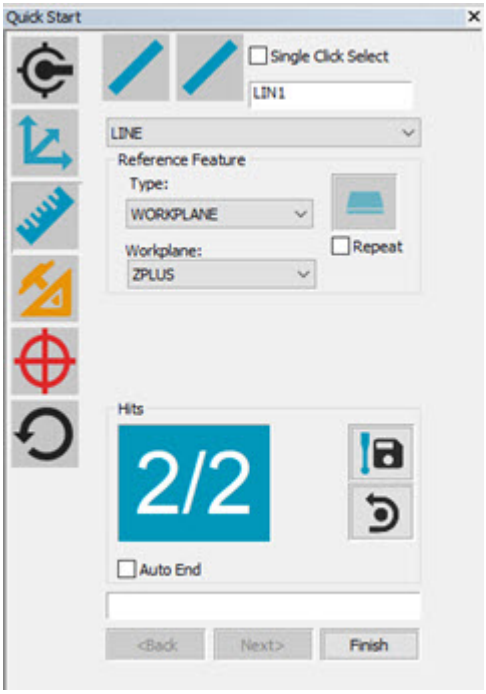
Print ( ) - This icon opens a standard **Print Setup** dialog box that enables you to send the print job to the printer.

Exit ( ) - This icon closes the **Excel Viewer**.

To maximize the window, double-click the title bar. To return the window to its previous size, double-click the title bar again.

Using the Quick Start Interface

The **Quick Start** interface (**View | Other Windows | Quick Start**) is a dialog box with a connected toolbar down the side.



Quick Start Dialog Box

This interface helps you create a measurement routine. It does this by providing dialog boxes or procedures that help you define or calibrate a probe, align your part, measure features, construct additional features, and dimension existing features.

To access an item, click on the desired toolbar icon. If the icon contains additional procedures, another toolbar appears to the right of the selected icon. You can select a specific procedure from the new toolbar.

Quick Start Toolbar Icons

The **Quick Start** toolbar contains these icons:

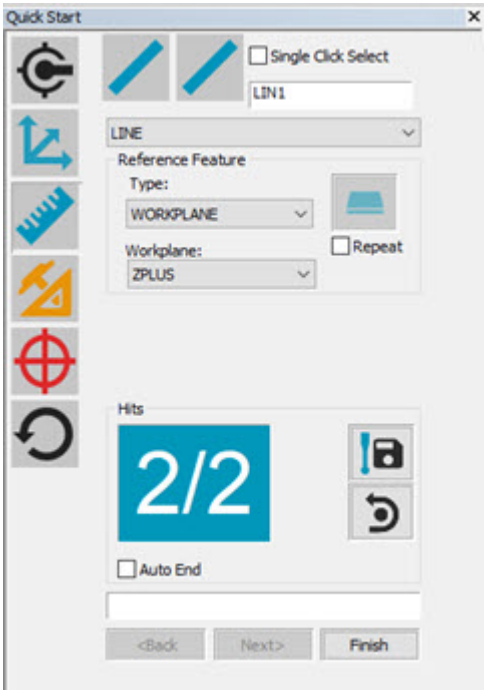
Icon	Description
	Calibrate Probes - This icon does not have a toolbar. It opens the **Probe Utilities** dialog box. You can use this dialog box to define a probe and calibrate probe tips.
	Alignments - Displays the **Quick Start Align** toolbar. You can select an alignment procedure from this toolbar.
	Measure - Displays the **Quick Start Measure** toolbar. You can select a measurement procedure from this toolbar.
	Construct - Displays the **Quick Start Construct** toolbar. You can select a construction procedure from this toolbar.
	Dimension - Displays the **Quick Start Dimension** toolbar. You can select a desired dimension procedure from this toolbar.
	Reset - This icon does not have a toolbar. It resets the **Quick Start** dialog box to Guess Mode.

Understanding the Quick Start Interface

When you enable the **Quick Start** interface, PC-DMIS disables the QuickFeature functionality. Also, if you are in the process of creating a Quick Start feature, you cannot perform any editing in the Edit window.

For example, you cannot use QuickFeatures if you enable the **Quick Start** interface. Also, you cannot delete, copy, or mark features in the Edit window; neither can you execute your measurement routine. To perform these and other actions, you need to first close the **Quick Start** interface.

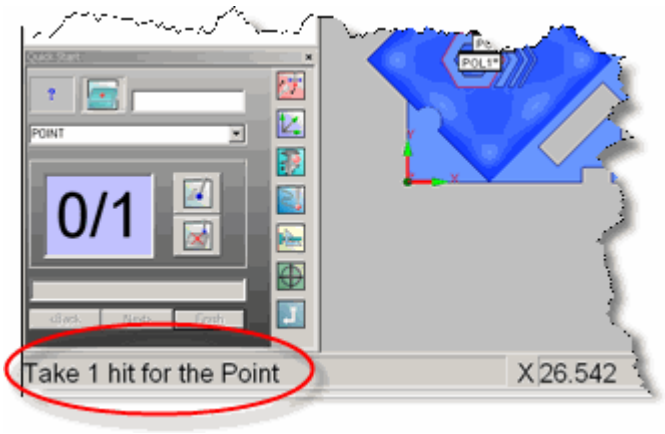
The **Quick Start** toolbar is tied to the **Quick Start** dialog box. This means that whenever you select the **Quick Start** interface, both the toolbar down the left side and the dialog box automatically appear and are inseparable.



Quick Start Dialog Box

In addition:

- Hover your mouse over a particular item on the dialog box or toolbar to display a tooltip with information on what that item does.
- To dock or undock the **Quick Start** dialog box to or from the left side of the screen, click the title bar and then drag and drop it. For details on docking and undocking user interface elements, see "Docking and Undocking User Interface Elements".
- Instructions for the various Quick Start procedures appear on the status bar at the bottom of your screen. If the instructions are too long for the Status bar area, PC-DMIS scrolls the instructions from right to left. To reset the scrolling of the current instructions to the beginning, move your mouse over the **Quick Start** dialog box.



Example showing the Quick Start instruction on the left side of the Status bar.

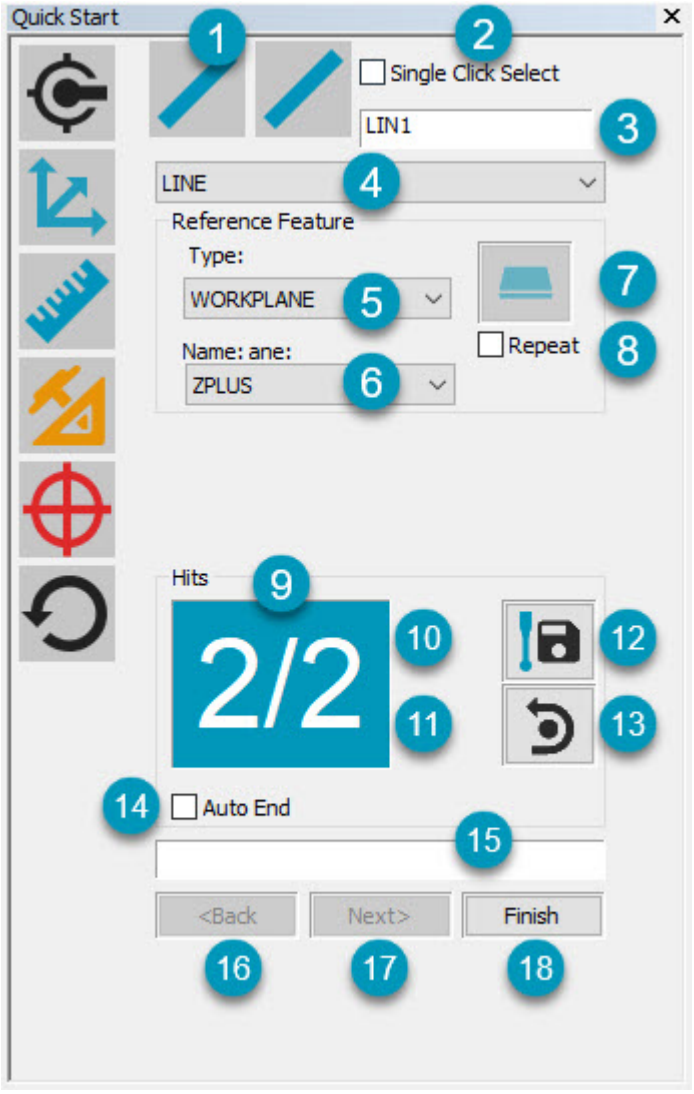
- To select input features used in the different procedures, click the feature from the Edit window or Graphic Display window.

- PC-DMIS includes a Status window that you can use to preview a dimension. If the dialog box is in Guess Mode, the preview is useful for reviewing a measured feature before you click the **Finish** button to insert the dimension or feature into the measurement routine. See "Using the Status Window" in the "Using Other Windows, Editors, and Tools" chapter.

Using the Quick Start Dialog Box

The **Quick Start** interface contains a toolbar attached to the **Quick Start** dialog box (**View | Other Windows | Quick Start**). Many of the procedures on this toolbar use the **Quick Start** dialog box to perform their operations. The toolbar icons are discussed in the "Using the Quick Start Interface" topic.

This topic focuses on describing the items on the dialog box portion of the interface and how to use the dialog box to perform various operations.



Quick Start Dialog Box

1 - **Graphical Description**

For all operations that use the **Quick Start** dialog box, PC-DMIS displays two icons at the top of the dialog box. A current procedure on the left and the current step in that procedure, or the current guessed feature, on the right.

If you see an asterisk icon, that means PC-DMIS is in a Guess Mode operation.

For example, in the above image, the Point icon on the right shows that with a single hit, a point feature gets created. The icon on the right changes to a line if you take another hit. PC-DMIS returns to this mode whenever you finish creating a Constructed feature or Alignment using the **Quick Start** dialog box. Click the **Guess Mode** button from the **Quick Start: Measure** toolbar to place PC-DMIS in Guess Mode.

Depending on the number of points taken, this mode guesses the feature type you're attempting to measure and dynamically updates the **Quick Start** dialog box to reflect this. For example:

- If you are in **Guess Mode** and you take two points, the **Quick Start** dialog box updates to a Line feature.
- If you take four points, it updates to a Circle feature.
- If you take eight points, it updates to a Cylinder, and so on.

See "Guessing a Measured Feature Type" in the "Creating Measured Features" chapter.

2 - **Single Click Select**

You can use this check box to create a measured feature with a single click of the mouse on the CAD model.

See "Creating Quick Start Measured Features" below for information.

3 - **ID**

The unique ID for the feature. An ID appears in the box once you select an appropriate procedure.

See "ID" in the "Navigating the User Interface" chapter.

See "Editing Values and IDs" in the "Using the Edit Window" chapter.

4 - **Feature Override**

This list is used to override the guessed feature measurement to the selected feature type. For example, if you take four hits and PC-DMIS guesses a plane, you can select **Circle** from this list to force it to create a measured circle instead.

See "Overriding a Guessed Measured Feature" in the "Creating Measured Features" chapter.

5, 6, 7, 8 - **Reference Feature**

You can project some measured features (circles, ellipses, lines, polygons, and slots) relative to a reference plane. The **Reference Feature** area appears in the **Quick Start** dialog box for these feature types where you can determine whether or not the measured feature is 3D, projected relative to the current workplane, or another user-defined plane.

You can select one of these types of reference features from the **Type** list (5):

- **3D** - The measured feature is created in 3D space directly from the hits probed on the part. It is not constrained to fit relative to a plane.

  For measured lines, the **3D** item is not available because there is no way for PC-DMIS to know which way to compensate them.

- **Workplane** - The measured feature is created as a 2D feature that is snapped onto a plane parallel to the referenced workplane and located at the average distance between the points.

- **Feature** - The measured feature is created as a 2D feature that is snapped onto a referenced user-defined plane.

  If you want the feature created at the average distance between the points on a plane parallel to the referenced user-defined plane, you must clear the **Move Feature to Reference Plane** check box in the list of check boxes on the **General tab** of the **Setup Options** dialog box. See "Setup Options: General Tab" for more information.

  If you need a reference plane feature that doesn't exist in the **Name** list (6), click the plane icon (7). The **Quick Start** dialog box walks you through the process of measuring the plane feature and then returns to the current line, circle, or slot measurement.

Repeat - This functionality ties into the plane icon discussed in the previous paragraph. In some instances, you may want to create a new reference plane for each feature. Rather than clicking the plane icon before each feature measurement, you can mark this check box to have PC-DMIS repeat the plane creation sequence before measuring each feature. PC-DMIS prompts you to take three hits to create the reference plane first. When you click **Finish**, it then prompts you to take the hits for the actual feature.

You can use the **Move Feature to Reference Plane** check box on the **General** tab of the **Set Up Options** dialog box to have 2D features snapped directly onto the reference plane instead of onto a parallel plane.

9 - **Number of Hits Taken / Required Hits**

This display shows two numbers.

- The number on the left of the slash indicates the number of hits you have currently taken.
- The number on the right of the slash indicates the minimum hits you need to measure the feature. You can take more than the minimum number of hits, in which case, the number to the left of the slash will be greater than the number on the right. You can also increase the number to a user-defined minimum.

10 - **Increase Hits**

The up arrow increases the feature's defined minimum number of hits (the number on the right of the slash) by one.

11 - **Decrease Hits**

The down arrow decreases the feature's defined minimum number of hits (the number on the right of the slash) by one.

12 - **Store a Move**

The **Store a Move** icon is used to easily store move points into the measurement routine. When you click this icon, PC-DMIS reads the current location of the probe and inserts a `MOVE/POINT` command into the Edit window.

See "Inserting a Move Point Command" in the "Inserting Move Commands" chapter.

13 - **Remove a Hit**

The **Remove a Hit** icon removes the last hit from the hit buffer.

14 - **Auto End**

You can use the **Auto End** check box to have the software automatically finish or end a feature in learn mode once it detects that you have taken the required number of hits. This means you don't need to press a key or click a button to learn the feature. You can use the **Increase Hits** or **Decrease Hits** arrow buttons to set the required number of hits.

Notes on laser Trackers

- If you mark **Auto End**, and from the **Compensation** area, you mark **Predefined**, the software finishes the feature when you measure the defined number of hits.

- If you clear **Auto End**, and from the **Compensation** area, you mark **Predefined**, the feature may finish early. Suppose you defined the required hits above the internal minimum needed for a feature (such as seven hits for a circle feature which internally only requires three), you can press END to learn the feature early as long as the number of hits you take meets or exceeds the internal required minimum.

- If you clear both **Auto End** and **Predefined**, you must always press **END** or click **Finish** to complete the feature.

For more information on the **Compensation** area, see "Predefined Compensation" in the Portable documentation.

15 - **Results**

The **Results** box shows the results of all steps in a measurement procedure taken thus far. For example, if you want to do a Plane - Line - Line alignment, the results box shows the following when you select or measure the second line:

Step 1:PLN1=Measured Plane

Step 2:LIN1=Measured Line

Step 3:LIN2=Measured Line

The **Results** box is tied to the buttons at the bottom of the dialog box. The buttons become enabled whenever you fulfill the requirements for a certain step in a procedure.

16, 17 - **<Back and Next>**

The **<Back** and **Next>** buttons cycle between lists of the required features or inputs. These buttons become available when procedures used on the toolbars require the selection or creation of multiple features (such as the **Dimension** and **Align** toolbars) or user input (such as upper and lower tolerance values for dimensions).

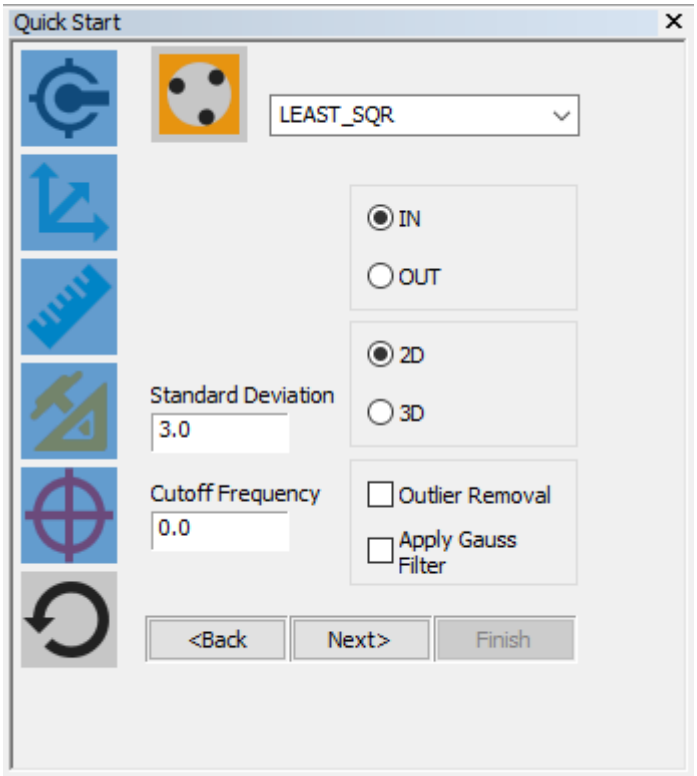
Clicking the **Done** button on your jog box acts the same as clicking the **Next>** button when using the **Quick Start** dialog box.

18 - **Finish**

The **Finish** button completes the procedure, inserts the appropriate command into the Edit window, and then, in most cases, PC-DMIS returns to the initial step of the current procedure. For alignments or constructed features, however, after you click **Finish**, PC-DMIS returns to the default Guess Mode.

Constructed Features Interface

For some constructed features, when working through the steps to create the features, the Quick Start interface displays additional options with a green background, similar to this:



These options are not described in this chapter. For information on these options, consult the appropriate topic in the "Constructing New Features from Existing Features" chapter.

Quick Start: Measure Toolbar

Quick Start Measure Toolbar

This toolbar contains the icons relative to the following measuring functions:

Point	Line	Plane
Circle	Cylinder	Cone
Sphere	Torus	Round Slot
Square Slot	Guess Mode	

For more information on creating measured features, see the "Creating Measured Features" chapter.

Scanning

The **Scan** icon lets you quickly perform a Fixed Time / Distance Manual Scan (Variable Delta). PC-DMIS prompts you with instructions in the Status bar. For more information on this type of scan, see one of the following:

- For a CMM, see "Performing a Fixed Time / Distance Manual Scan" in the "Scanning" chapter of the PC-DMIS CMM help.
- For a portable device, see "Performing a Fixed Time / Distance Manual Scan" in the "Portable Hard Probe Scanning" chapter of the PC-DMIS Portable help.

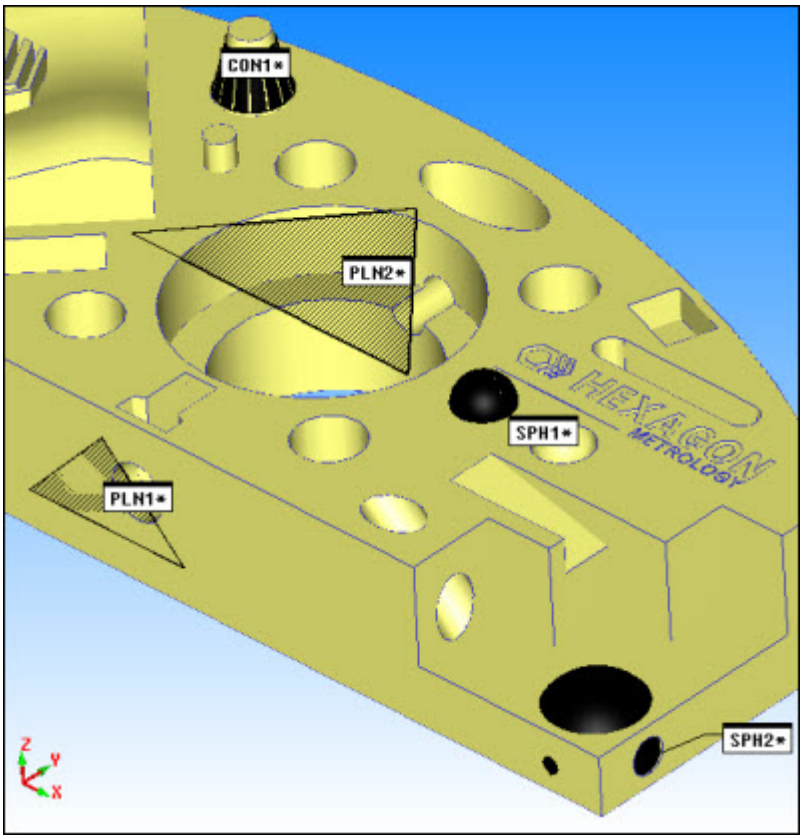
Creating Quick Start Measured Features

1. From the **Quick Start** toolbar, select the **Measure** toolbar. For details, see the "Using the Quick Start Interface" chapter.

2. Click the icon of the feature to measure. The top-left icon in the dialog box changes to display the features that PC-DMIS measures in this procedure. Instructions appear on the status bar.
3. With PC-DMIS in Offline mode, click the **Program Mode** icon on the **Graphic Modes** toolbar (for details, see "Graphic Modes Toolbar"). PC-DMIS draws a simulated depiction of the probe near the part in the Graphic Display window.
4. With PC-DMIS in Offline mode, right-click to set the depth for the probe. With PC-DMIS in Online mode, move the probe to the desired depth.
5. Take the minimum number of hits on the part to measure the feature.
6. Click **Finish** or press **Done** on your jog box when the measurement is complete. The **Quick Start** dialog box remains open. PC-DMIS automatically compensates for the probe radius and places the measured feature on the part in the Graphic Display window.
7. Follow steps four through six until you create all of the measured features you need.

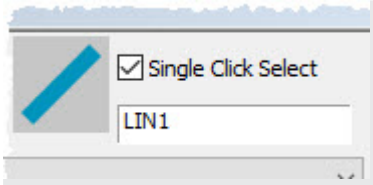
8. Click **Close** when finished. The **Quick Start** dialog box closes.

As PC-DMIS takes hits and creates the feature, it draws the measured feature on the screen. For 3D measured features (Torus, Cylinder, Sphere, Cone, and Plane), PC-DMIS draws the feature with a shaded surface.



Example showing measured features with shaded surfaces.

Creating Measured Features from CAD Using Single Click Select Option



The **Single Click Select** option works with all measured features except Torus and scans. PC-DMIS disables this option internally during Guess Mode. See the note below.

The **Single Click Select** check box controls how PC-DMIS creates features when you use CAD. If you select the check box, you can now learn a measured feature with a single mouse-click on the CAD model. To do this:

1. Select the type of feature that you want to create from the **Quick Start** dialog box's **Measure** toolbar.

2. Mark the **Single Click Select** check box.

3. Click, with your mouse, on the CAD model near the feature that you want to create. PC-DMIS locates the feature in the CAD model and automatically generates the necessary hits to create the feature. The hit count increases and displays the predefined number of hits for the feature type you chose.

Feature Type	Number of Hits
Point	1
Line	2
Plane	4
Circle	4
Cylinder	8
Cone	8
Sphere	5
Round Slot	6
Square Slot	5

4. Once PC-DMIS generates the hits, click **Finish** to insert the feature into the measurement routine.

Single Click Select and Guess Mode

This icon places PC-DMIS into Guess mode. Single Click Select does not work in Guess mode even though the check box remains selected. For Guess mode to function, PC-DMIS attempts to detect the feature type at each successive hit. To maintain consistency with existing functionality, PC-DMIS disables Single Click Select internally if you use Guess mode. If the Quick Start window is closed, PC-DMIS automatically reverts to the legacy Guess mode even if Single Click Select was selected when the Quick Start window was closed. Single Click Select requires you to identify, ahead of time, what feature is being measured. This capability is unique to the Quick Start window.

An asterisk icon appears in the top-left corner of the **Quick Start** dialog box to show that the software is in Guess mode.

Measuring a Point

	You can use the **Point** icon to measure the position of a point belonging to a plane aligned with a reference plane (shoulder) or a point in space.
To create a measured point you must take one hit on the part.	
For information on the associated Edit window command, see the "Basic Measurement Format for a Point" topic in the "Creating Measured Features" chapter.	

Measuring a Line

	You can use the **Line** icon to measure the orientation and linearity of a line belonging to a plane aligned with a reference plane, or a line in space.

To create a measured line, you must take two hits on the part.

Measured Lines and Workplanes

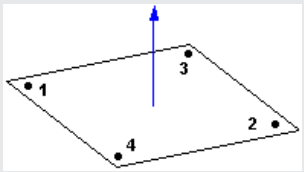
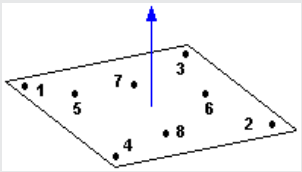
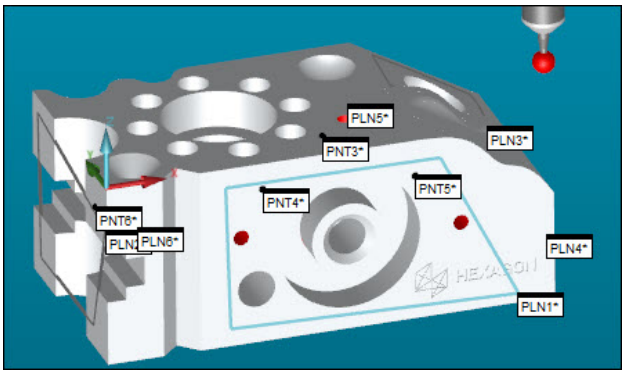
When creating a measured line, PC-DMIS expects the hits for the line to be taken at a vector perpendicular to the current workplane.

For example, if your current workplane is ZPLUS (with a vector 0,0,1), and you have a block-like part, the hits for the measured line must be on a vertical wall of that part, such as the front or side.

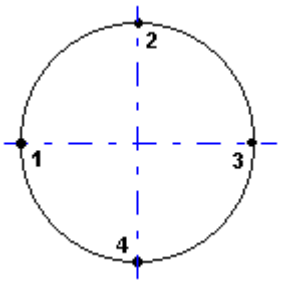
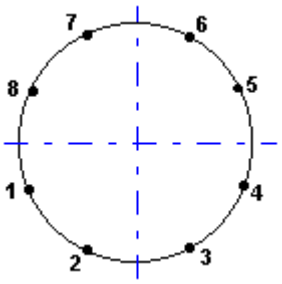
If you then wanted to measure a line feature on the top surface of the part, you would need to switch the workplane to XPLUS, XMINUS, YPLUS, or YMINUS, depending on the direction of the line.

For information on the associated Edit window command, see the "Basic Measurement Format for a Line" topic in the "Creating Measured Features" chapter.

Measuring a Plane

	You can use the **Plane** icon to measure any flat or planar surface.
To create a measured plane, you must take a minimum of three hits on any flat surface. If you only use the minimum of three hits, it's best to select the points in a large triangular pattern that cover the widest area of the surface.	
Example Plane with 4 Points	Example Plane with 8 Points
	
When you create a plane feature, PC-DMIS shows the outline of the plane from the plane's hits in the Graphic Display window:  *Sample plane from four hits taken on the Y-face* For information on the associated Edit window command, see the "Basic Measurement Format for a Plane" topic in the "Creating Measured Features" chapter.	

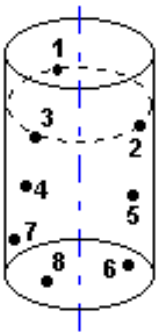
Measuring a Circle

<table>
<tr><td> then </td><td>You can use these Circle icons to measure the diameter, roundness, and position of the center of a hole or stud parallel to a reference plane, i.e. the perpendicular section of a cylinder aligned with a reference axis.</td></tr>
<tr><td colspan="2">To create a measured hole or stud, you must take a minimum of three hits. The plane is automatically recognized and set by the system during measurement. You must distribute the points to be picked uniformly around the circumference of the circle.</td></tr>
<tr><td>Example Circle with 4 Points</td><td>Example Circle with 8 Points</td></tr>
<tr><td> 
</td><td> 
</td></tr>
<tr><td colspan="2">For information on the associated Edit window command, see the "Basic Measurement Format for a Circle" topic in the "Creating Measured Features" chapter.

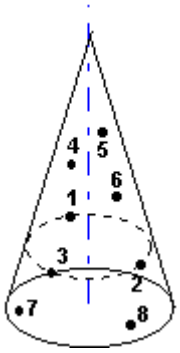
 You can also create circles from a single point by using the Measure Single Point Circle toolbar item. This is useful when you attempt to measure a hole with a probe whose sphere size is larger than the hole's diameter and therefore cannot fit entirely into the hole to take the usual minimum three hits required. For more information, see the PC-DMIS Portable documentation.

For details on how to create single-point circle features with PC-DMIS Portable, see "Creating Single-Point Circle Features" topic in the PC-DMIS Portable documentation.</td></tr>
</table>

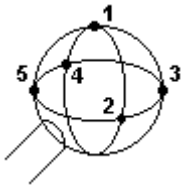
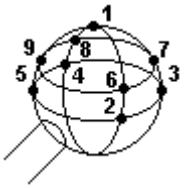
Measuring a Cylinder

	You can use the **Cylinder** icon to measure the diameter, cylindricity, and orientation of the axis of a cylinder oriented in space. PC-DMIS also calculates the position of the baricenter of the points picked.
To create a measured cylinder, you must take a minimum of six hits on the cylinder. You must distribute the points to be picked uniformly over the surface of the cylinder. The first three points picked must lie on a plane perpendicular to the main axis.	
 *Example showing a cylinder feature with eight points.*	
For information on the associated Edit window command, see the "Basic Measurement Format for a Cylinder" topic in the "Creating Measured Features" chapter.	

Measuring a Cone

	You can use the **Cone** icon to measure conicity, angle at the tip, and orientation in space of the axis of a cone. The position of the baricenter of the points picked is also calculated.
To create a measured cone, you must take a minimum of six hits. The points to be picked must be uniformly distributed on the surface. The first three points picked must lie on a plane perpendicular to the main axis.	
 *Example showing a cone feature with eight points.*	
For information on the associated Edit window command, see the "Basic Measurement Format for a Cone" topic in the "Creating Measured Features" chapter.	

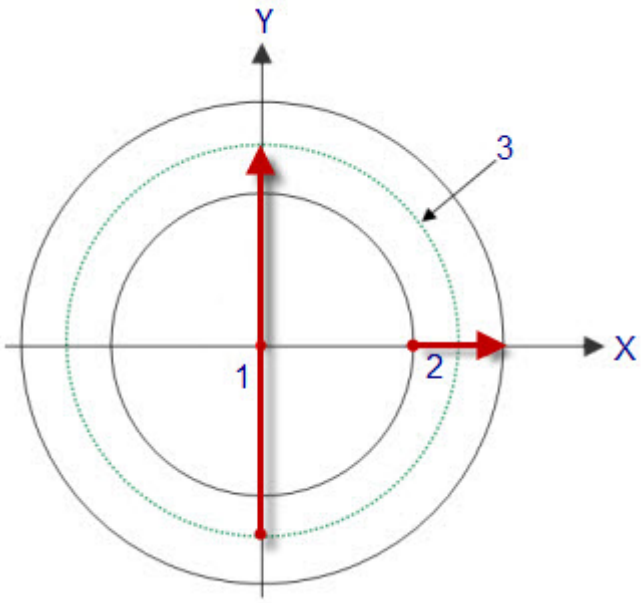
Measuring a Sphere

	You can use the **Sphere** icon to measure the diameter, sphericity, and position of the center of a sphere.
To create a measured sphere, you must take a minimum of four hits. You must distribute the points to be picked uniformly over the surface of the sphere. The first four points picked must not lie on the same circumference. The first point should be taken on the pole of the sphere's cup. The other three points are taken on a circumference.	
Example Sphere with 5 Points	Example Sphere with 9 Points
	
For information on the associated Edit window command, see the "Basic Measurement Format for a Sphere" topic in the "Creating Measured Features" chapter.	

Measuring a Torus

	You can use the **Torus** icon to measure the center diameter and ring diameter of the torus feature. The position of the barycenter of the points picked is also calculated.

To create a measured torus, you must take a minimum of seven hits. Take the first three hits on one level of the torus's center line circle (see the diagrams below). These hits must represent the orientation of the torus so that an imaginary circle generated through these three hits would have roughly the same vector as the torus.

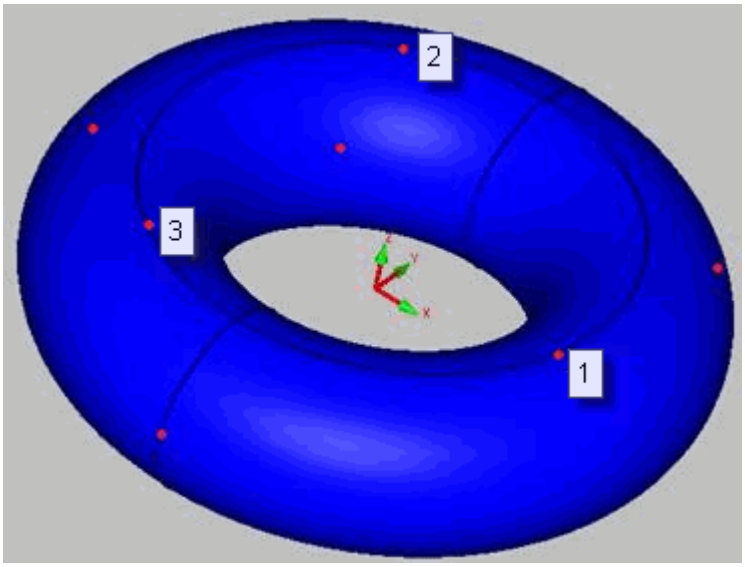


Top-down view of a torus. Note the major diameter (1), the minor diameter (2), and the center line circle (3).

If you orient the torus and are looking down on it from a bird's eye view, with Z+ pointing up towards you, take the first three hits in a counter-clockwise direction to give the torus a vector of 0,0,1. If you take the hits in a clockwise direction, the torus will have a vector of (0,0,-1).

You can probe the remaining 4 hits in any random location as long as they all don't lie in the same plane.

Example Torus with 7 Points



Example Torus created from 7 points, with the first three taken in a counter clockwise direction

For information on the associated Edit window command, see the "Basic Measurement Format for a Torus" topic in the "Creating Measured Features" chapter.

Measuring a Round Slot

then	You can use these **Round Slot** icons to create a measured round slot.

To create a measured round slot, you must take at least six hits on the slot, usually two points on each straight side and one point on each curve. Alternately, you could take three points on each curve.

Example showing a round slot feature with six points.

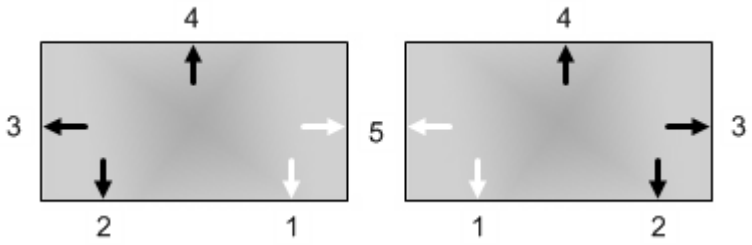
You can also create measured slots from two points. This is useful when the probe sphere is larger than the slot's diameter, and you're not able to take the required hits. For more information, see the PC-DMIS Portable documentation.

For information on the associated Edit window command, see the "Basic Measurement Format for a Round Slot" topic in the "Creating Measured Features" chapter.

Measuring a Square Slot

then	You can use these **Square Slot** icons to create a measured square slot.

To create a measured square slot, you must take five hits on the slot, two on one of the long sides of the slot and then one hit on each of the three remaining sides. The hits must be taken in a strictly Clockwise (CW) or Counterclockwise (CCW) direction.



Example showing a square slot feature with five points in CW (left) and CCW (right) directions.

With this icon, you can also create measured slots from two points. This is useful when the probe sphere is larger than the slot's diameter, and you're not able to take the required hits. For more information, see the PC-DMIS Portable documentation.

For information on the associated Edit window command, see the "Basic Measurement Format for a Square Slot" topic in the "Creating Measured Features" chapter.

Using Guess Mode

	Use the **Guess Mode** icon to have PC-DMIS guess the type of measured feature to create based on the hits taken.

The following table shows the minimum number of hits PC-DMIS needs to guess a particular Measured Feature and that feature's dimension type.

Minimum # of Hits	Feature	Dimension
1	Point	1D
2	Line	2D
3	Plane	2D
3	Circle	2D
6	Cylinder	3D
6	Cone	3D
4	Sphere	3D
7	Torus	3D
6	Round Slot	2D
5	Square Slot	2D

See the "Guessing a Measured Feature Type" topic in the "Creating Measured Features" chapter for rules and additional information on using guess mode.

Quick Start: Construct Toolbar

Quick Start Construct Toolbar

When you construct elements, instead of processing picked points, PC-DMIS processes the characteristic points of already measured elements or elements you will measure.

The range of elements that you can construct is identical to the range of the elements that you can measure using the **Quick Start Measure** toolbar. The most frequent application is the construction of the circle passing through the centers of a circular pattern of holes or bosses.

Typically, the elements used to construct other elements are points and circles (or spheres) which you may or may not have measured and stored already. If you have not already measured a needed feature, you can measure it during the construction procedure.

The features you can construct are:

Point Features		Line Features	
	• Intersect Point • Mid Point • Projected Point		• Best Fit Line • Intersect Line • Mid Line
Plane Features		**Circle Features**	
	• Best Fit Plane • Mid Plane • Alignment Plane		• Best Fit Circle • Intersect Circle • Height Circle • Diameter Circle
Slot Features		**Cylinder Features**	
	• Slot Feature		• Cylinder Feature
Cone Feature			
	• Cone Feature		

See the topics in the "Constructing New Features from Existing Features" chapter of the PC-DMIS Core documentation for in-depth information on these feature constructions.

Creating Constructed Features

This procedure below contains very basic information on constructing features. For in-depth information, see the "Constructing New Features from Existing Features" chapter.

1. From the **Quick Start** toolbar, select the **Construct** toolbar. See "Using the Quick Start Interface" chapter for details.
2. Click the icon of the feature to be measured. The **Quick Start** dialog box shows an icon of the procedure and a second icon for the first needed feature type.
3. Follow the instructions at the bottom of the toolbar or on the Status bar. You can either fill in the data in the dialog or select (or measure) the needed features to use in the construction.
4. Continue following the instructions until the **Finish** button becomes available. If you find items that appear in the Quick Start user interface that are not covered in the "Using the Quick Start Dialog Box" topic, consult the "Constructing New Features from Existing Features" chapter.
5. Click **Finish** when ready. PC-DMIS places the newly constructed feature on the part in the Graphic Display window and also in the Edit window.

Quick Start: Dimension Toolbar

Quick Start Dimension Toolbar

The **Dimension** toolbar lets you perform geometric measurements and geometric tolerance checks. With the exception of the **Key-In** dimension, this toolbar contains all the dimensions from the normal PC-DMIS **Dimension** toolbar. See "Dimension Toolbar" in this chapter.

> The **Quick Start Dimension** toolbar only creates legacy dimensions. It does not create the new Geometric Tolerance dimension commands covered in the "Using Geometric Tolerances" chapter.

About Geometric Measurements	About Geometric Tolerance Checks
You can use geometric measurements in the following circumstances: • When dimensions involve two elements (such as distances and angles). • When dimension elements are not measurable (such as corners). • When you can obtain dimension measurements from geometric calculation procedures (intersections, projections, and / or middle elements). In each geometric relationship between two elements, PC-DMIS creates a third element (point, circle, line, or plane). The default output format of the calculated element includes the most meaningful dimensions. Do not confuse *Geometric relationships* with *geometric tolerances*. • *Geometric tolerances* are used to check the *functional requirements* of a feature. *Functional requirements* are feature characteristics that ensure sufficient assembly, safety, appearance, performance etc. • *Geometric relationships*, as used in PC-DMIS, are not sufficient to ensure functional requirements because they do not utilize the full extent of datum theory.	Geometric tolerance checks test the location, orientation, profile, runout, or form error of a feature (checked feature) relative to a datum reference frame (as applicable). The following checks are available: • **Orientation checks:** test for errors of parallelism, perpendicularity, and angularity. • **Location checks:** test for errors of coaxiality, and concentricity, location, and position. • **Profile checks:** test for errors of profile form or profile, relative to the current alignment. • **Runout checks:** test for errors of total runout of a circle, cylinder, or plane. PC-DMIS inserts a dimension of the resulting geometric tolerance check between two elements into the Edit window.

Icon	Description	Quick Start Procedure
	Creates a legacy Location dimension.	Select or measure the feature and click **Next**. Specify the plus and minus tolerances and then click **Finish** to insert the Location dimension.
	Creates a legacy Position dimension.	Select or measure the feature and click **Next**. Specify the plus and minus tolerances and then click **Finish** to insert the Position dimension.
	Creates a legacy Distance dimension.	Select or measure the first feature and click **Next**. Select or measure the second feature and click **Next**. Specify the plus and minus tolerances, type a nominal value, select either 2D or 3D, and then click **Finish** to insert the Distance dimension.
	Creates a legacy Angle dimension.	Select or measure the first feature and click **Next**. Select or measure the second feature and click **Next**. Specify the plus and minus tolerances, type a nominal value, select either 2D or 3D, and then click **Finish** to insert the Angle dimension.
	Creates a legacy Concentricity dimension.	Select or measure the first circular feature and click **Next**. Select or measure the second circular feature and click **Next**. Specify the plus tolerance and click **Next**. Click **Finish** to insert the Concentricity dimension.
	Creates a legacy Coaxiality dimension.	Select or measure the first feature (cylinder, cone, or line) and click **Next**. Select or measure the second feature (cylinder, cone, or line) and click **Next**. Specify the plus tolerance and click **Next**. Click **Finish** to insert the Coaxiality dimension.
	Creates a legacy Circularity dimension.	Select or measure the first circular feature and click **Next**. Select or measure the second circular feature and click **Next**. Specify the plus tolerance and click **Next**. Click **Finish** to insert the Roundness dimension.

	Creates a legacy Cylindricity dimension.	Select or measure a Cylinder and click **Next**. Select or measure the second feature and click **Next**. Specify the plus tolerance and click **Next**. Click **Finish** to insert the Cylindricity dimension.
	Creates a legacy Straightness dimension.	Select or measure a Cone, Line, or Cylinder and click **Next**. Specify a plus tolerance and click **Next**. Click **Finish** to insert the Straightness dimension.
	Creates a legacy Flatness dimension.	Select or measure a Plane and click **Next**. Specify a plus tolerance and click **Next**. Click **Finish** to insert the Flatness dimension.
	Creates a legacy Perpendicularity dimension.	Select or measure the first feature and click **Next**. Select or measure the second feature. Specify a plus tolerance and a projection distance and click **Next**. Click **Finish** to insert the Perpendicularity dimension.
	Creates a legacy Parallelism dimension.	Select or measure a non-point feature and click **Next**. Select or measure the second non-point feature. Specify a plus tolerance and a projection distance and click **Next**. Click **Finish** to insert the Parallelism dimension.
	Creates a legacy Total Runout dimension.	Select or measure a Cone, Cylinder, Line, or Plane and click **Next**. Specify a plus and minus tolerance and click **Next**. Click **Finish** to insert the Total Runout dimension.
	Creates a legacy Circular Runout dimension.	Select or measure a Circle, Cone, Cylinder, Line, or Sphere and click **Next**. Specify a plus and minus tolerance and click **Next**. Click **Finish** to insert the Circular Runout dimension.
	Creates a legacy Surface Profile dimension.	Select or measure a feature and click **Next**. Specify a plus and minus tolerance. Choose whether the profile is Form Only or Form and Location and click **Next**. Click **Finish** to insert the Surface Profile dimension.

	Creates a legacy Line Profile dimension.	Select or measure a feature, and click **Next**. Specify a plus and minus tolerance and click **Next**. Click **Finish** to insert the Line Profile dimension.
	Creates a legacy Angularity dimension.	Select or measure any non-point or non-sphere feature and click **Next**. Select a Cone, Cylinder, Line, or Plane and click **Next**. Specify a plus tolerance, a distance, and angle and click **Next**. Click **Finish** to insert the Angularity dimension.
	Creates a legacy Symmetry dimension.	Select or measure a line or set for the first feature and click **Next**. Select the second feature and click **Next**. Select the third (or datum) feature and click **Next**. Type a plus tolerance and click **Next**. Click **Finish** to insert the Symmetry dimension.

Dimensioning Features Using the Quick Start Interface

The procedure below provides basic instructions on how to use the Quick Start interface to create dimensions:

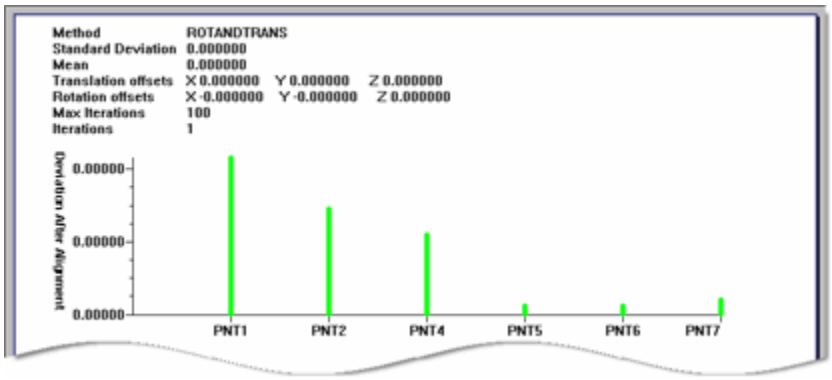
1. From the **Quick Start** toolbar, select the **Dimension** toolbar. See "Using the Quick Start Interface" chapter for details.
2. Select the desired dimension from the toolbar. The icon for the selected dimension procedure appears in the **Quick Start** dialog box and instructions appear in the Status Bar at the bottom of your screen.
3. Follow the instructions on the Status Bar by selecting features from the Edit window or Graphic Display window (or by measuring them if they don't exist).
4. Follow the instructions on the Status Bar to type values into the **Quick Start** dialog box.
5. Continue following instructions and clicking **Next** until the **Finish** button becomes available.
6. Click **Finish**. PC-DMIS inserts the dimension into the measurement routine.

Quick Start: Align Toolbar

Quick Start Align Toolbar

The **Align** toolbar is used to create alignments from specific feature types by using these procedures.

Icon	Description	Quick Start Procedure
	Plane, Line, Line alignment	Select or measure a Plane to level to, and click **Next**. Select or measure a Line to rotate to, and click **Next**. Select or measure a second Line. The origin is created by translating the part to the two lines. Click **Finish** to insert the alignment.
	Plane, Line, Circle alignment	Select or measure a Plane to level to, and click **Next**. Select or measure a Line to rotate to, and click **Next**. Select or measure a Circle. The circle's center, projected onto the line, is where PC-DMIS sets the alignment's origin. Click **Finish** to insert the alignment.
	Plane, Circle, Circle alignment	Select or measure a Plane to level to, and click **Next**. Select or measure the first Circle feature, and click **Next**. Select or measure the second Circle feature. PC-DMIS rotates the alignment to the line created from the centers of the two circles. It then uses the first circle's center to set the alignment's origin. Click **Finish** to insert the alignment.
	Plane, Line, Point alignment	Select or measure a Plane to level to, and click **Next**. Select or measure a Line to rotate to, and click **Next**. Select or measure a Point for the alignment's origin. Click **Finish** to insert the alignment.
	Cylinder, Line, Point alignment	Select or measure a Cylinder, and click **Next**. Select or measure a Line, and click **Next**. Select or measure a Point. PC-DMIS levels the alignment to the surface where you clicked the point, rotates the alignment to the line, and sets its XY origin in the center of the cylinder. Click **Finish** to insert the alignment.
	Align Free alignment	This opens up the **Alignment Utilities** dialog box. See "Description of the Alignment Utilities Dialog Box" in the "Creating and Using Alignments" chapter for information on using this dialog box to create an alignment.

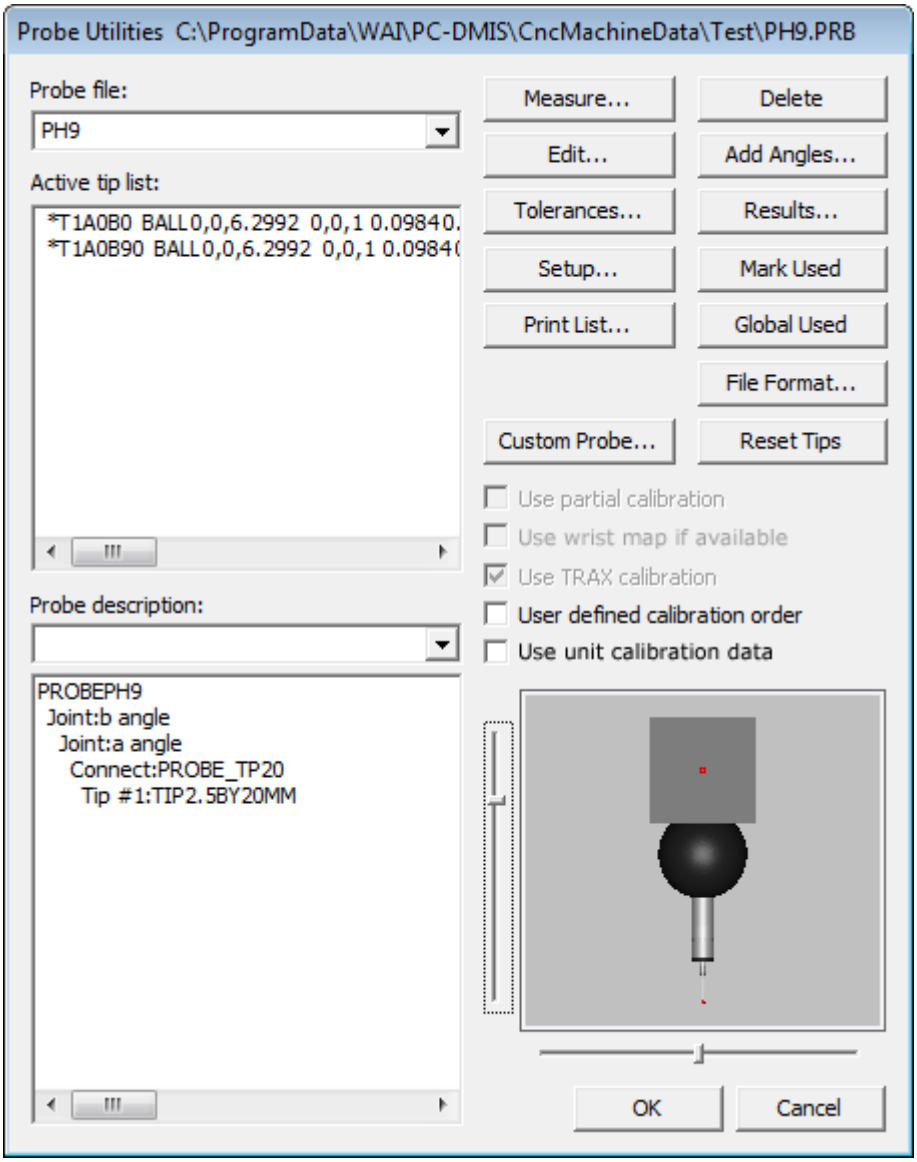
	Six Points Best Fit alignment	Follow the on-screen prompts to select or measure six points. A typical procedure is to measure three points on the top surface to level to the Z Axis. Measure two points on the front surface to rotate to the X Axis. Then measure one point to define the origin for the Y axis. Click **Finish**. This establishes the correct origin for the alignment. PC-DMIS inserts the Best Fit 3D Alignment. Following execution, PC-DMIS displays a 3D Alignment Best Fit Graphical Analysis in the Report window.  *Example showing the graphical analysis of a Best Fit alignment.* This graphical analysis of the 3D Best Fit alignment displays this information in the Report window: • **Header** - This contains various values used in the Best Fit alignment: Method, Standard Deviation, Mean, Translation offsets, Rotation offsets, Max iterations, Iterations. • **Vertical Axis** - This shows the amount of deviation after the alignment. • **Horizontal Axis** - This displays the IDs of the points used in the alignment.

For additional ways of creating alignments see the "Creating and Using Alignments" chapter.

Quick Start: CALIBRATE PROBES

CALIBRATE PROBES Button

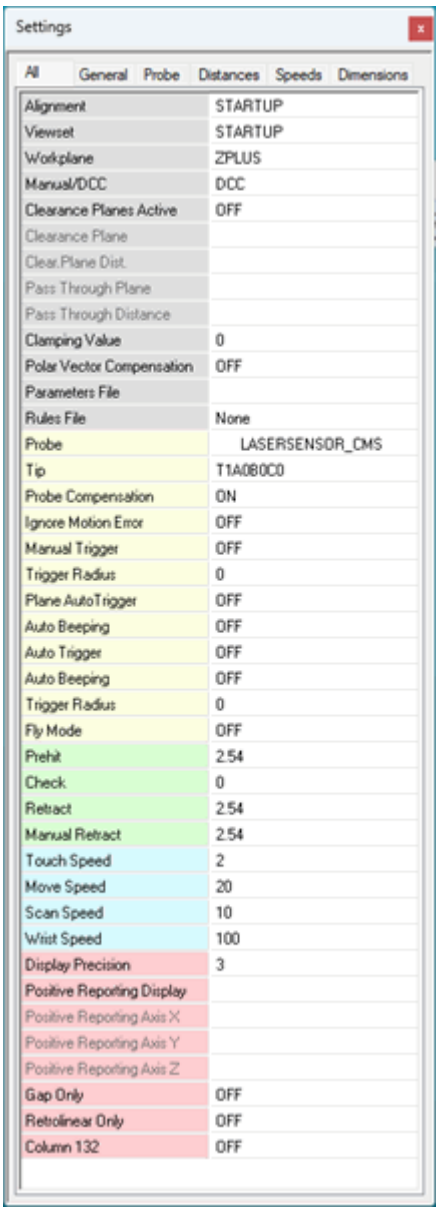
Click the **CALIBRATE PROBES** button to open the **Probe Utilities** dialog box. You can use this dialog box to create probes and calibrate their angles. For information on how to use this dialog box, see the "Defining Probes" topic in the "Defining Hardware" chapter of the PC-DMIS Core documentation.



Probe Utilities Dialog Box

Using the Settings Window

Select **View | Other Windows | Settings Window** to open the Settings window.



Settings Window

This dockable window contains several tabs of frequently modified settings. Each tab contains color-coded, editable settings, specific to that tab. The settings are linked to your cursor's current location in the measurement routine and updates their display to match the setting in the measurement routine at the cursor's location. For example, if you have multiple `LOADPROBE` commands, and you select each `LOADPROBE` command in the Edit window, PC-DMIS updates the Settings window to display the selected `LOADPROBE` command.

Editing a Setting

To edit a setting, simply click on the field to the right of the setting, and select a different option, or type a new value. When you change a setting, PC-DMIS inserts the appropriate parameter change into the measurement routine at the cursor's location.

Available Settings

The following tabs and settings are available. The shaded background on the cells in the table below indicates the default color associated with those settings:

All tab

This tab contains all the settings from all the tabs

General tab

Alignment - This inserts a `RECALL/ALIGNMENT` command, to recall the alignment you select from the list.

Viewset - This inserts a `RECALL/VIEWSET` command for the saved view you selected from the list.

Manual/DCC* - This inserts a `MODE/DCC` or `MODE/MANUAL` to place the measuring machine into either DCC or Manual mode respectively.

Workplane - This changes your current workplane by inserting a new `WORKPLANE` command into the measurement routine.

Clearance Plane Active - This activates a clearance plane by inserting a `CLEARP` command. Pressing F9 on the inserted command allows you to change the affected plane.

Clearance Plane - Allows you to define what plane has the clearance plane. This remains unavailable for selection until you activate the Clearance Plane Active setting or until a cursor rests on a `CLEARP` command.

Clear Plane Dist. - Lets you set the distance for the defined Clearance Plane.

Pass Through Plane - This indicates the plane that the probe passes through to get to the next feature plane.

Pass Through Distance - This defines the distance for the pass through plane.

Clamping Value - This value tells the controller how firmly to hold the probe on a Leitz CMM. It only works with the Leitz controller and the Leitz brand TTP.

Polar Vector Compensation - This allows the measurement of Vector and Surface points to always be compensated along a Polar vector by inserting a `POLARVECTORCOMP` command.

Probe tab

Probe - This inserts a `LOADPROBE` command for the selected probe into the measurement routine.

Tip - This defines the AB angle to use for the `LOADPROBE` command by inserting the selected `TIP` command into the measurement routine.

Probe Compensation - This turns probe compensation on or off by inserting the appropriate `PROBECOM` command.

Ignore Motion Error - This inserts an `IGNOREMOTIONERROR/ON` or `IGNOREMOTIONERROR/OFF` command into the measurement routine. If you turn this setting on, PC-DMIS will not stop when the probe encounters a collision.

Manual Trigger - This setting inserts a `POINT MANUAL TRIGGER` command into your measurement routine. For details, see the "Point Manual Trigger area" topic in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation. This option is disabled in DCC mode.

Trigger Tolerance - This setting inserts a `TRIGGERTOLERANCE` command. This command tells PC-DMIS to only accept a hit if the hit is within the defined tolerance zone. You can use this with manual machines to take accurate hits.

Trigger Tol. Zone - This value determines the tolerance radius for the Trigger Tolerance setting.

Trigger Plane - This setting inserts a `TRIGGERPLANE` command. This tells PC-DMIS to automatically take a hit when the probe passes the plane defined by the surface normal of an Auto feature at the level of the defined depth. You can use this command with manual machines; instead of pressing a button to take a hit, you can place `TRIGGERPLANE` commands at any standard location within the Edit window.

Auto Trigger - This setting inserts an `AUTOTRIGGER` command into your measurement routine. When PC-DMIS detects that the probe has moved into the defined tolerance zone, it automatically takes a hit. For details, see the "Point AutoTrigger area" topic in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.

Auto Beeping - This setting determines whether or not PC-DMIS uses a beeping with your `AUTOTRIGGER` command. If this is ON, the closer the probe gets to the target, the more frequent the beeps.

Auto Tolerance Zone - This setting defines the radius for the Auto Trigger tolerance zone.

Fly Mode* - This inserts a `FLY` command and sets it to either ON or OFF. The `FLY` command works with a `MOVEPOINT` command and moves the probe around the part in a smooth, non-stopping motion.

Distances tab

Prehit* - This inserts a `LOADPROBE` command for the selected probe into the measurement routine.

Check* - This setting inserts a `CHECK` command that sets the distance past the theoretical hit location that the machine will continue to search for the surface of the part.

Retract* - This setting inserts a `RETRACT` command that sets the distance that the machine will move away from the actual hit location before going from Touch Speed to Move Speed.

Manual Retract* - This setting inserts a `MANRETRACT` command that sets the retract distance your CMM will automatically travel whenever you take a manual hit.

Speeds tab

Touch Speed* - This setting inserts a `TOUCHSPEED` command that changes the speed at which the CMM takes hits. Values include 1 - 20 percent.

Move Speed* - This inserts a `MOVESPEED` command that determines how fast the CMM moves in between taking hits. Values include 1 - 100 percent.

Scan Speed* - This setting inserts a `SCANSPEED` command that determines how fast your machine scans your part. Possible values include 1 - 100 percent.

Dimensions tab

Display Precision - This setting inserts a `DISPLAYPRECISION` command with a value equal to the value in the setting. Any features following this command shows decimal places to the specified value.

Display Positive Reporting - This setting inserts a `POSITIVEREPORTING` command into the measurement routine. This command displays features on the negative side of the origin with positive values. You can choose this reporting on all data or only on deviations.

Positive Reporting Axis X - This setting determines whether or not PC-DMIS displays the positive reporting on the X axis.

Positive Reporting Axis Y - This setting determines whether or not PC-DMIS displays the positive reporting on the Y axis.

Positive Reporting Axis Z - This setting determines whether or not PC-DMIS displays the positive reporting on the Z axis.

Gap Only - This setting inserts a `GAPONLY` command into the measurement routine. When set to ON, and dimensioning an edge point location, PC-DMIS calculates the location axes by projecting the measured point to the theoretical surface and then projecting this new point onto the theoretical approach vector. PC-DMIS then calculates any location axes from this new point.

Retrolinear Only - This setting inserts a `RETROLINEARONLY` command into the measurement routine. When set to ON, and dimensioning surface or vector point locations, the location axes are calculated in this manner:

- First, finding the largest component of the theoretical surface normal vector (largest in the X, Y, or Z direction).

- Second, projecting the measured point to this largest component vector in a manner that the projection is perpendicular to the original theoretical surface normal vector.
- The location axes are then calculated from this new projected point.

*These items are not available when using a portable device.

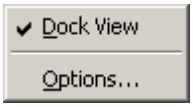
For in-depth documentation, find the specific options in one of these topics:

- Settings Toolbar
- Probe Mode Toolbar
- Selecting Setup Options
- Modifying Report and Motion Parameters

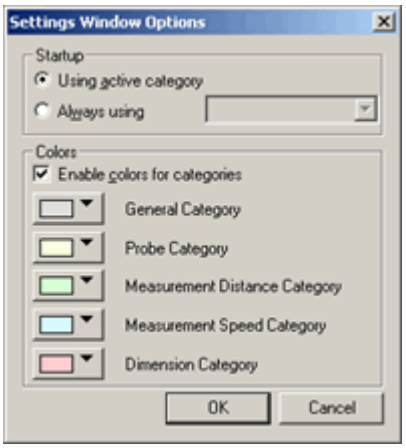
You can also insert many of these settings from the **Insert | Parameter Change** submenu.

Changing Settings Window Options

You can change your Settings window options by right-clicking in the window and selecting **Options** from the shortcut menu.



The **Settings Window Options** dialog box appears.



Settings Window Options Dialog Box

You can use the **Settings Window Options** dialog box to modify the colors for the different categories. It is also used to control the categories that appear when the Settings window is started.

Startup Area

This area contains two startup options for the Settings window:

Using active category - The Settings window displays the last used category when it starts.

Always using - The Settings window displays the category selected from the drop-down list when it starts.

Colors Area

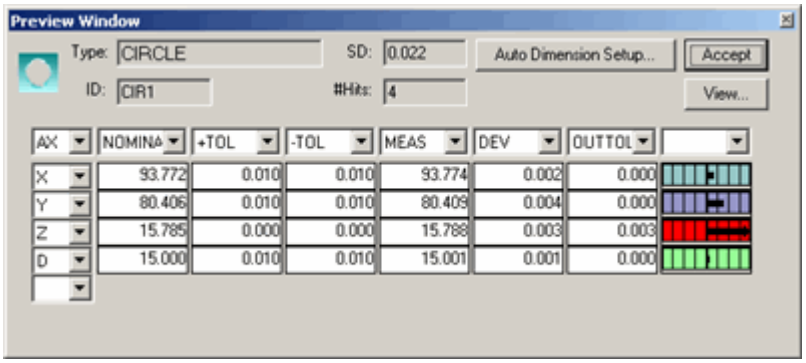
You can use the **Enable colors for categories** check box to disable or enable the color-coded display for the categories of settings.

The drop down arrows on the colored boxes are used to modify the default color-coded display for the available categories.

Using the Preview Window

> The Preview window functionality is deprecated and is no longer developed. While it remains in the software for any who still use it, the Status window functionality and elements of the Using the Probe Readouts window ultimately replace it. Please consider using those tools instead.

Selecting the **View | Other Windows | Preview Window** menu option opens a window that allows you to preview the results of a feature measurement before actually accepting the measurement. After taking probe hits and pressing the End key on your keyboard (or DONE on the jog box) PC-DMIS displays the feature in the Graphic Display window and dimension information for the feature in the Preview window.



Preview Window dialog box showing the X, Y, Z, and diameter values of a measured circle feature.

The Preview window uses the same color scheme used for dimensions to show the deviations from acceptable tolerances; these colors allow you to quickly determine whether or not a particular measurement falls within acceptable tolerances or not (see

"Editing Dimension Colors" in the "Editing the CAD Display" chapter for information on how to change the tolerance colors used for dimensions).

To determine which feature information is displayed in the Preview window, simply place your Edit window cursor on the command line for any feature in the Edit window. Generally the cursor rests on the last feature in the Edit window.

Below are the available options for the Preview window.

Preview Window Options

The Preview window displays general information about a feature as well as more specific dimensional information. It contains these options:

ID - This box determines the name label for the measured feature or dimension.

SD - This box determines the Standard Deviation for the measured feature.

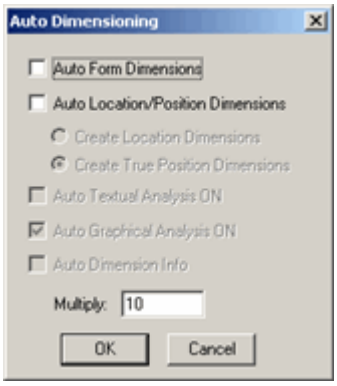
#Hits - This box determines the number of probe hits taken to measure the feature.

Hide - The **Hide** button closes the Preview window. You can open the Preview window by selecting **View | Preview Window**.

Accept - The **Accept** button accepts the measurement and creates an automatic dimension for the feature. You can only select this functionality using the **Auto Dimension Setup** button (see "Auto Dimension Setup" for details).

View - The **View** button displays the Analysis window for the appropriate dimension for the measured feature. See "Inserting Commands Related to the Analysis Window" for a discussion on the Analysis window.

Auto Dimension Setup - Selecting the **Auto Dimension Setup** button displays the **Auto Dimensioning** dialog box:



For information on this dialog box, see "Auto Dimension Setup" under "Setup Options: Dimension tab" in the "Setting Your Preferences" chapter.

AX - This list displays the selected axes in the Preview window. You can select these axes: **X, Y, Z, D, R, A, T, PR, PA, M, V, L, PD, RS, RT, S, H.** You can view information on up to six axes at one time by selecting them from the various **AX** lists.

For information on these axes, see "Default Axes" and "Sheet Metal Axes" in the "Using Legacy Dimensions" chapter.

NOMINAL - This box determines the nominal (or ideal) values for each axis.

+ TOL - This box determines a tolerance value above the nominal value for which the measurement is still acceptable.

- TOL - This box determines a tolerance value below the nominal value for which the measurement is still acceptable.

MEAS - This box determines the actual measured values.

MAX - This box determines the maximum value you can have before your measurement goes out of tolerance.

MIN - This box determines the minimum value you can have before your measurement goes out of tolerance.

DEV - This box determines the measurement's deviation from the nominal value.

OUTTOL - This box determines the value by which the measured value is out of tolerance.

If a sound file is associated with the **Preview Window Feature In Tolerance** event, a sound plays when a dimension is within tolerance. If a sound file is associated with the **Preview Window Feature Out Of Tolerance** event, a sound plays when a dimension is out of tolerance. For help with associating sound files with events, see "Setup Options: Sound Events tab".

Sizing the Preview Window

Like most windows, the Preview window can be sized to fit your current needs. If you need it larger or smaller, simply drag the edge of the window with the mouse and make it the desired size.

The Preview window in Learn mode differs from the Preview window in Execute mode, and you can change the sizes for each.

Using the Marked Sets Window

Selecting the **View | Other Windows | Marked Sets Window** menu option accesses the Marked Sets window. You can use this window to store a group of marked features that you want to execute. See the "Creating and Executing Marked Sets" topic in the "Editing a Measurement Routine" chapter.

Using the Probe Readouts Window

The **View | Other Windows | Probe Readouts** menu option displays a readout of the current CMM position and other useful information.

Sample Probe Readouts Window	Explanation
 Probe Readouts Window	A. Feature ID (CIR1) and Feature type (CIRCLE) B. Current XYZ probe position C. Form error D. XYZ position of the last hit E. Total deviation value F. Distance to next location shown in out-of-tolerance dimension colors G. Error of last measured feature H. Rotary table angle I. Screen counters that show the mouse location by using the CAD model's coordinate system J. Screen counters that show the mouse location by using the current part coordinate system K. Number of hits taken AB axis angles appear below the number of hits taken if you have an active wrist probe (such as a PHS or the CW43 light). L. Probe Readouts comments. This comes from the Probe Readouts comment type. See the "Inserting Programmer Comments" in the "Inserting Report Commands" chapter.

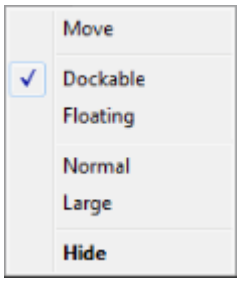
The Probe Readouts window changes depending on the options that you selected from the **Probe Readouts Setup** dialog box (see "Setting Up the Probe Readouts Window" in the "Setting your Preferences" chapter).

To access the **Probe Readouts Setup** dialog box, right-click in the Probe Readouts window and click Setup.

Sizing the Probe Readouts Window

You can alter the size of this window in two ways. If the window is docked, first float the readouts window (see below):

- **Method 1:** With the mouse, drag any of the window borders until you reach the desired size.
- **Method 2:** Right-click on the window's title bar to display the Probe Readouts menu. Select the **Large** option to display the Probe Readouts window centered on the screen and maximized to 95% of the screen's size. Select the **Normal** option to display the Probe Readouts window in the previous size prior to maximizing. Click anywhere outside the menu to close it.



Probe Readouts Menu

To close the Probe Readouts window, click on the **X** button in the upper right-hand corner.

Each time you change the location or size of the Probe Readouts window, PC-DMIS updates the proper entry. The next time you select the option, the window displays precisely as it did in the previous viewing.

On machines with an external DRO, this option is not available.

Docking or Floating the Probe Readouts Window

You can dock the Probe Readouts window to the sides of the screen or against other windows by right-clicking on the title bar. From the Probe Readouts menu (shown above), select the **Docking** option. To make the window float on top of other windows again, choose the **Floating** option from the same menu. You may need to drag the window on top of the Graphic Display window if the title bar is not visible.

You can also hold down the Ctrl key while you drag the window to prevent it from docking.

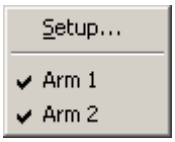
For details on docking and undocking user interface elements, see "Docking and Undocking User Interface Elements".

Multiple Arm Mode and the Probe Readouts Window

If you have more than one arm measuring your part, PC-DMIS displays the probe readout information for each additional arm in a new column.

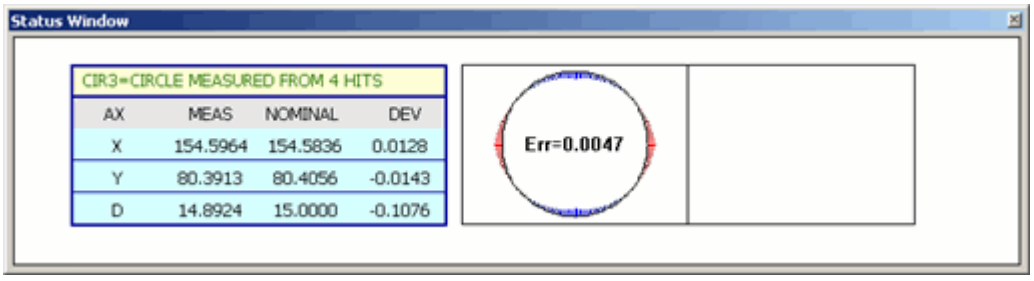
Example of the Probe Readouts window showing multiple arms in two columns.

You can right-click on this window to display a shortcut menu that you can use to show or hide information about a particular probe.



If a check mark appears next to a menu item, its information appears in the Probe Readouts window.

Using the Status Window



Status Window

The **View | Other Windows | Status Window** menu option displays the Status window. You can use this window to preview commands and features while you create them from the **Quick Start** toolbar, during feature execution, when you create or edit dimensions, and when you click the item in the Edit window with the Status window open. See "Using the Quick Start Interface" chapter for details.

While similar in some ways to the Preview window, the Status window differs in these ways:

- You don't specify tolerances or nominal values in this window. The window is for display only.
- It provides a flexible way to display commands by tying into the template-based Reporting functionality.
- It displays a dynamic preview for any command or dimension type, not just Location dimensions, as long as those commands are properly assigned inside your report and label templates.

This window functions the same as other dockable and scrollable windows. It displays scroll bars if the content within the window exceeds the window's size. Additionally, you can dock and undock this window at the bottom or top of the Graphic Display window when you double-click the window's title bar. You can also drag the window and then release it at the desired location. For details on docking and undocking user interface elements, see "Docking and Undocking User Interface Elements".

You can also hold down the Ctrl key while you drag the window to prevent it from docking.

The Status Window with the Quick Start Toolbar

The Status window displays the current status of any Guess Mode measurements, feature selections, and dimensions when you use the **Quick Start** toolbar. In the case of the screen shot shown above, this dimension has not yet been created, yet PC-DMIS provides a preview of the dimension in this window before you click **Finish** from the **Quick Start** toolbar.

The Status Window during Feature Creation or Execution

The Status window displays preview information of features not yet displayed in the Report window. This window utilizes the reporting template functionality to generate dynamic previews of commands during execution and creation. During feature creation, it only previews the feature if PC-DMIS is in Guess Mode.

The Status Window during Dimensioning

When you have one of the dimension dialog boxes open (either for legacy dimensions or for Geometric Tolerance dimensions), the Status window provides you with a preview of what the dimension will look like based on the items currently selected in the dialog box. For example, if you are creating a Distance dimension and want to preview the distances as you select different items, that information appears in the Status window as you click on the items in the dialog box. If a dialog box isn't active, it reverts to its default functionality of displaying information for the last item that you clicked on in the Edit window.

Changing the Status Window Template

Since the Status window uses reporting templates to display its information, you can change what template it uses if you would like it to display different information. You can display any command in the Status window as long as a label template is assigned to that command in the report template (.rtp) file. You assign what report template this uses from

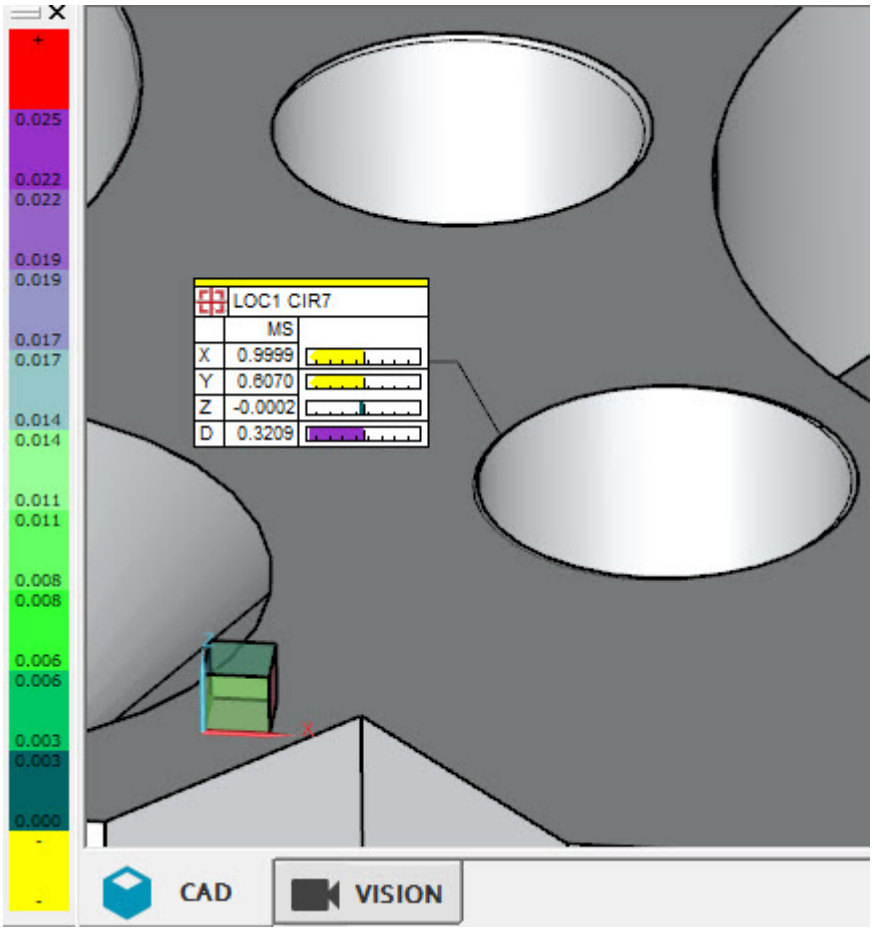
the `ReportTemplate` entry in the **StatusWindow** section of the PC-DMIS Settings Editor. The default template it uses is TEXTONLY.RTP.

Using the Status Window in Accumulation Mode

During execution, the Status window usually only displays the most recently executed feature and dimension. However, you can have PC-DMIS show a list of accumulated executed items by setting the `AccumulateStatusWindowResults` entry in the **Reporting** section to True in the PC-DMIS Settings Editor. With this mode turned on, the contents in the Status window don't get removed until execution finishes. This allows you to show a running list of information during measurement routine execution. Also, while in this mode, PC-DMIS follows the TextReportObject's properties in the Status window's template to determine what information gets sent to the Status window (features, dimensions, alignments, and so forth).

Using the Dimension Colors Window (Dimension Colors Bar)

The **View | Other Windows | Dimension Colors** menu option displays the Dimension Colors Bar. This dockable, non-sizable, bar-like window shows the colors for dimension tolerances and their associated scale values.



Example showing the Dimension Colors bar for a location dimension and associated dimensions.

You can drag and dock the color bar on either the right-side or left-side of the screen.

For details on how to dock and undock user interface elements, see "Docking and Undocking User Interface Elements".

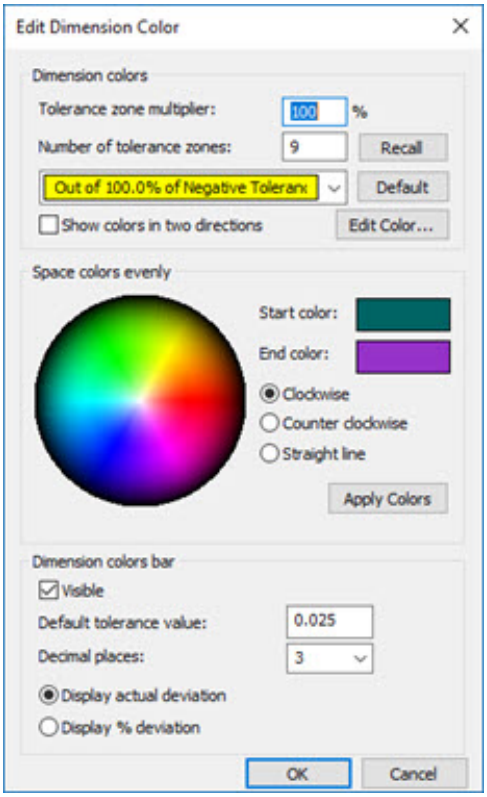
How it Displays

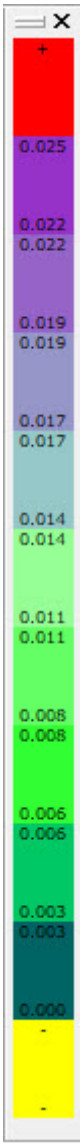
The top of the bar displays the current default tolerance value, which is represented by a label in a white band with its numerical scale value.

The remaining colored bands on the bar consist of the same number of defined tolerance zones as in the **Edit Dimension Colors** dialog box. An additional colored band on the bottom displays "Out of Tolerance (Negative)" (labeled with a "-" sign), and a colored band on the top displays "Out of Tolerance (Positive)", (labeled with a "+" sign).

Access the Edit Dimension Colors Dialog Box

To open the **Edit Dimension Colors** dialog box, right-click on the color bar:

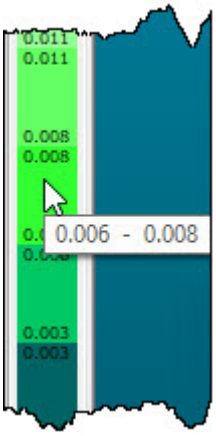




This dialog box defines the dimension colors and tolerance zones. It also contains some settings that determine how the color bar displays. For information on this dialog box, see the "Editing Dimension Colors" topic in the "Editing the CAD Display: Introduction" chapter.

Tooltips for Complete Information

If you move your mouse over a tolerance band, a tooltip appears. It displays the range of values for that tolerance band (lower to higher). The tooltip displays the "Out of Tolerance (Positive)” or "Out of Tolerance (Negative)" label when you position your mouse pointer over the "+” or "-" band, respectively. You cannot modify the decimal places.



Using the Q-DAS Data Editor Window

The Q-DAS Data Editor lets you modify the K-field data in the Q-DAS data file (QdasData.xml) through PC-DMIS.

The **View | Other Windows | Q-DAS Data Editor** menu option displays the Q-DAS Data Editor window.

This window functions the same as other dockable and scrollable windows. It displays scroll bars if the content within the window exceeds the window's size. Additionally, you can dock and undock this window at the bottom or top of the Graphic Display window when you double-click the window's title bar. You can also drag the window and then release it at the desired location. For details on docking and undocking user interface elements, see "Docking and Undocking User Interface Elements".

Q-DAS Data Editor Window

Prerequisites

To modify the K-field data of the dimension in the Q-DAS Data Editor, you need to do the following:

- You need a Q-DAS data file for the measurement routine. The file name has the following format:

  <name of measurement routine>.QdasData.xml

<name of measurement routine> represents the name of the PC-DMIS measurement routine.

For information on how to create a Q-DAS data file, see "Creating the Q-DAS Data File" in the Q-DAS Configurator help file.

- You need to place the cursor on the dimension in the Edit window. If you place your cursor on any command other than the dimension command, Q-DAS Data Editor displays this message:

  "The cursor is not placed on a dimension command."

  When you place the cursor on a dimension command in the Edit window and if no Q-DAS data file is available for the measurement routine, Q-DAS Data Editor displays this message:

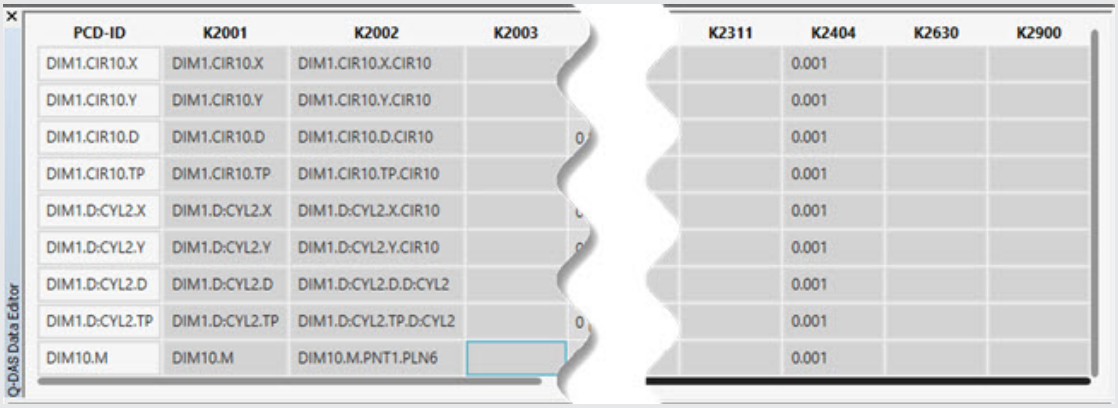
  "No QdasData.xml file exists for the measurement routine. Use Q-DAS Configurator to create a QdasData.xml file."

When you save the measurement routine, the changes you made in the Q-DAS Data Editor are also saved.

Suppose that you have the following position dimension `DIM1` command in your measurement routine:

```
DIM1 =POSITION : CIR10

        FEATCTRLFRAME/
SHOWNOMS=NO,SHOWPARAMS=YES,SHOWEXPANDED=YES


CADGRAPH=OFF,REPORTGRAPH=OFF,TEXT=OFF,MULT=10.00,ARROW
DENSITY=100,OUTPUT=BOTH,UNITS=MM

        COMPOSITE=NO,FIT TO DATUMS=YES,DEV PERPEN
CENTERLINE=ON,OUTPUT ALIGNMENT=Datum Reference Frame

        CUSTOMIZED DRF=NO

        STANDARDTYPE=ASME_Y14_5

                SIZE TOLERANCES/1DIAMETER,10,0.2,0.2

                PRIMARY DIMENSION/
POSITION,DIAMETER,0.2,MMC,D,MMC,,,

                SECONDARY DIMENSION/,0.2,MMC,A,D,MMC,,

                NOTE/DIM1

                FEATURES/CIR10,,
```

When you place the cursor on the above dimension command, Q-DAS Data Editor displays this data:



ClearanceCube Settings

You can use the ClearanceCube Settings window to define the ClearanceCube motion for each feature and tip in the measurement routine.

ClearanceCube Settings Window

PC-DMIS displays the **ClearanceCube Settings** window automatically when you activate ClearanceCube for the measurement routine.

From the **ClearanceCube** toolbar, you can use the **ClearanceCube Settings Window** icon to show or hide this window. You can also open this window from the **View | Other Windows | ClearanceCube Settings** menu option.

You can dock the ClearanceCube Settings window at the top or bottom of the Graphic Display window. To float the ClearanceCube Settings window, press the Ctrl button and then click and drag the window to the desired location.

- **ID(s)** - This list displays one or more feature IDs based on your selection of features in the Edit window. To do this, follow these steps:

  1. Place your cursor on a feature in Command Mode or select a feature in Summary Mode to display or edit the settings for the ClearanceCube.
  2. To edit the ClearanceCube settings for more than one feature at the same time, select them in the Edit window and then make the necessary changes.
  3. From the **ID(s)** list, select Features or Tips. This box appears when you select multiple features and tips in the Edit window.
  4. Based on your selection from the **ID(s)** list, the list shows the features or tips. Select the item in the list for configuration.

If you position your mouse cursor over a command in Command Mode or select a command in Summary Mode that does not require ClearanceCube settings, PC-DMIS displays the message “ClearanceCube is not required for the selected commands”.

- **Activate ClearanceCube Moves** - This switch determines whether you want to use ClearanceCube motion for the features you selected in the **ID(s)** list.

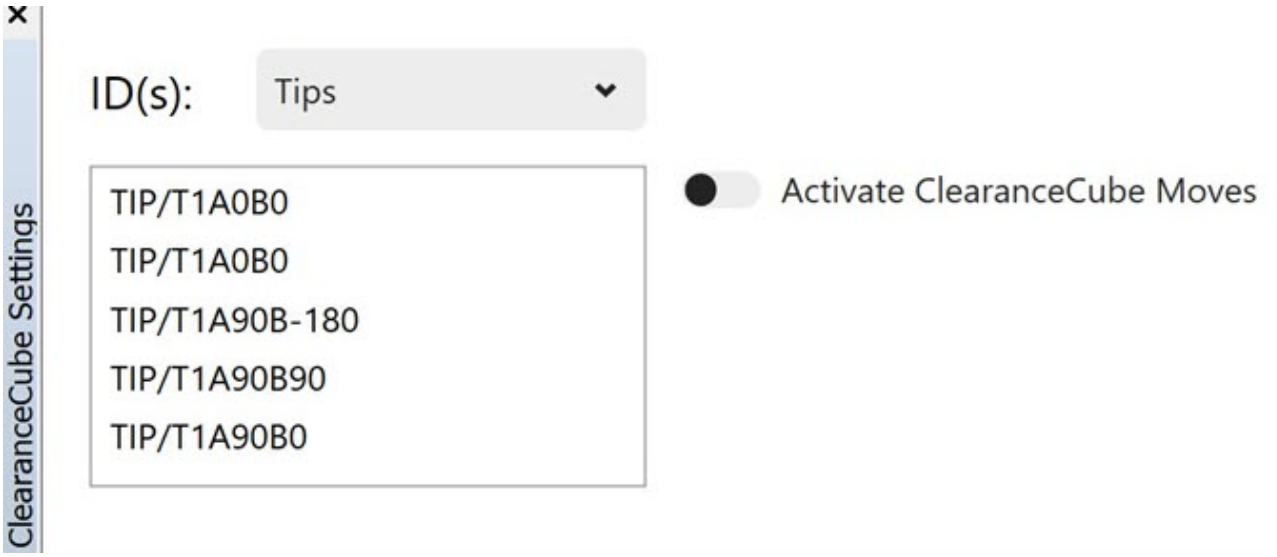
- **Starting Face** - This determines the ClearanceCube face that PC-DMIS moves to before it measures the feature. You have these options:
  - Select a face
  - Use Tip Vector
  - Set the Starting Face to OFF if you only want the ending motion to occur for the feature.
- **Ending Face** - This determines the ClearanceCube face that PC-DMIS moves to after it measures the feature. You have these options:
  - Select a face
  - Use Tip Vector
  - Set the Ending Face to OFF if you only want the starting motion to occur for the feature.

The feature vector determines the default setting for the ClearanceCube **Starting Face** and **Ending Face** options. PC-DMIS marks the **ClearanceCube uses Tip Vector for Start/End Face** check box in the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) by default. It is recommended to use the tip vector.

Based on constraints set in the **Constraints** tab, the selection of some axes for the starting and ending faces may not be available.

For Tip Commands

From the **ID(s)** list, select the features or tips. This box appears only when you select multiple features and tips in the Edit window.



ClearanceCube Settings Window - Tips Option

When you select the **Tips** option from the **ID(s)** list, the **ClearanceCube Setting** window only shows the **Activate ClearanceCube Moves** setting.

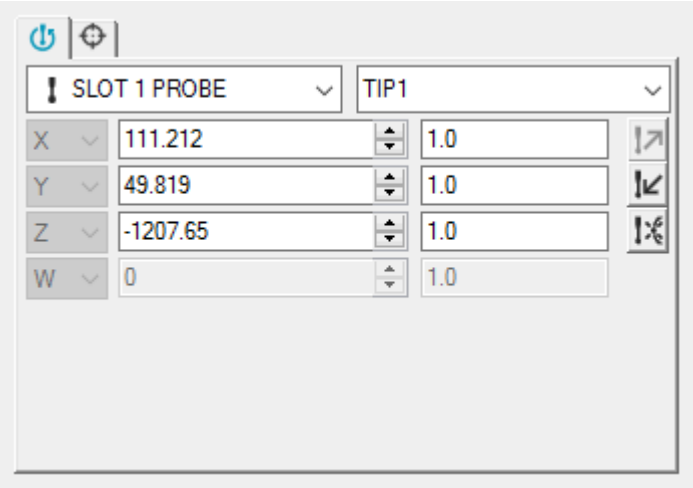
The ClearanceCube Shortcut Menu

In addition to the ClearanceCube Settings dialog box, PC-DMIS provides a right-click menu. To access this menu, follow these steps:

1. If you are in Command Mode, place your cursor on a feature. If you are in Summary Mode, click a feature to select it.
2. Right-click the feature and select the **Path | ClearanceCube** option.
3. From the **Starting Face** and **Ending Face** list, select the face to align the ClearanceCube to. The available options are -X, +X, -Y, +Y, -Z, +Z, Use Tip Vector, and Off.

Using the Probe Toolbox

The **View | Other Windows | Probe Toolbox** menu option displays the Probe Toolbox.



Probe Toolbox showing the Position Probe tab.

The Probe Toolbox lets you perform various probe-related manipulations. It presents tabs and information relative to the type of probe currently used. If you use an optical probe or a laser probe, for example, the tabs that appear are different than for contact probes, providing you the functionality of manipulating parameters specific to those probe types.

Most of the tabs in the Probe Toolbox appear when you view the toolbox embedded within the **Auto Feature** dialog box.

See the documentation sets that apply to your particular configuration for specific information on the Probe Toolbox:

- PC-DMIS CMM (and Portable)
- PC-DMIS Vision
- PC-DMIS Laser

Because the items in the Probe Toolbox are so frequently used when creating Auto features, the Probe Toolbox automatically appears as an embedded portion of the **Auto Feature** dialog box.

Placing and Sizing the Toolbox

Much like the Edit window or the various toolbars available in PC-DMIS, you can dock the Probe Toolbox to the sides of the PC-DMIS application window. You can also float it above other elements.

To dock the Probe Toolbox:

1. Ensure that the Probe Toolbox is in docking mode. (Right-click on the title bar and select **Dockable** from the shortcut menu.)
2. Select the toolbox's title bar with the mouse.

3. Drag the toolbox onto the left or right edge of the PC-DMIS application window.
4. Release the mouse. PC-DMIS docks the toolbox at that location.

To float the Probe Toolbox:

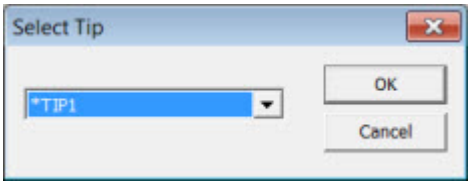
1. Select the toolbox's title bar with the mouse.
2. Drag the toolbox out of its docked position on top of the Graphic Display window and release the mouse. While it floats above the Graphic Display window, it is still in docking mode.
3. Right-click on the title and select **Floating** from the shortcut menu.
4. Drag the toolbox to where you want to place it.
5. Release the mouse. PC-DMIS floats the toolbox at that location.

You can also hold down the Ctrl key while you drag the window to prevent it from docking.

To size the toolbox, select an edge of the toolbox and drag the mouse to a new location.

Selecting a Tip

The **Insert | Parameter Change | Probe | Active Tip** menu item is used to select an active tip. To select a tip, highlight the desired tip ID in the list and select **OK**. The tip is inserted in the Edit window.



Select Tip Dialog Box

Noncalibrated tips have an asterisk before their tip ID # in the list.

Viewing Path Lines

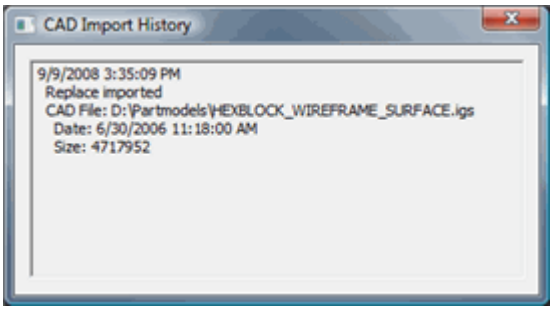
You can select the **View | Path Lines** menu option to draw the current path of the probe in the Graphic Display window.

The software only generates the probe's path lines for features that follow a `MODE/DCC` command. If your measurement routine doesn't have a `MODE/DCC` command, you won't see any path lines.

For more information on working with path lines, see "Showing, Animating, and Moving Path Lines" in the "Editing the CAD Display" chapter.

Viewing CAD Import History

The **View | CAD Import History** menu option displays the **CAD Import History** dialog box.



CAD Import History Dialog Box

This dialog shows information for all the CAD files that have been imported into the current measurement routine. This information includes the following:

- The date and time of the import operation
- The type of import operation (Merge or Replace)
- Whether or not a Direct CAD Interface (DCI) was used to import the model
- The CAD file's name, date, and size in bytes

This information is stored in the measurement routine file, not the CAD model file.

This only tracks import operations taken within PC-DMIS. For example, if you manually replace a measurement routine's CAD file, it is not tracked by the **CAD Import History** dialog box.

Using Toolbars

Using Toolbars: Introduction

In an effort to decrease the time it takes to program your measurement routine, PC-DMIS offers you a variety of toolbars composed of frequently used commands. You can access most toolbars in two ways.

- Select **View | Toolbars** and select a toolbar from the menu.
- Right-click on PC-DMIS's **Toolbar** area and select a toolbar from the shortcut menu provided.

The toolbars provided are discussed in these topics:

- File Operations Toolbar
- Graphic Modes Toolbar
- Graphic View Toolbar
- Graphic Items Toolbar
- Edit Window Toolbar
- Window Layouts Toolbar
- CAD Setup Toolbar
- Measurement Strategy Toolbar
- Auto Features Toolbar
- Measured Features Toolbar
- Constructed Features Toolbar
- Pointcloud Toolbar
- Mesh Toolbar
- Dimension Toolbar
- Alignment Toolbar
- Settings Toolbar

- Probe Mode Toolbar
- Select Tip Toolbar
- Active Arms Toolbar
- Active Rotary Table Toolbar
- ClearanceCube Toolbar
- Wizards Toolbar
- Build and Inspect Toolbar
- QuickMeasure Toolbar
- QuickCloud Toolbar
- Quick Fixture Toolbar

You can also customize and create your own toolbars. For more information, see "Customizing the User Interface" in the "Navigating the User Interface" chapter.

File Operations Toolbar

File Operations Toolbar

Before Certification

File Operations toolbar for PC-DMIS-User

File Operations toolbar for PC-DMIS-Programmer

After Certification

File Operations toolbar for PC-DMIS-User

File Operations toolbar for PC-DMIS-Programmer

You can use the **File Operations** toolbar (**View | Toolbars | File Operations**) to perform these operations:

New File - Click this option to create a new measurement routine.

Open File - Click this option to open an existing measurement routine.

Close File - Click this option to close the active measurement routine.

Quit - Click this option to quit PC-DMIS.

Save File - Click this option to save the current measurement routine.

Save As - Click this option to save the current measurement routine under a different name.

View Certified - Click this option to open the certified version of the measurement routine in Protected Mode.

> This button is available only for certified measurement routines.

Submit - Click this option to submit the current measurement routine for certification and generate a .protect file.

Cancel Submission - Click this option to cancel the submission of the current measurement routine for certification.

> The **Submit** and **Cancel Submission** buttons are available only if you belong to the **PC-DMIS-User** group and you have the Protect Viewer option added to your license.

Certify - Click this option to certify the current measurement routine.

Reject - Click this option to reject the certification for the current measurement routine.

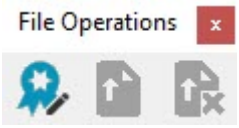
Delete - Click this option to delete all the changes you made after the last certification of the current measurement routine. You can then edit the measurement routine from the beginning.

> The **Certify**, **Reject**, and **Delete** buttons are available only if you belong to the **PC-DMIS-Programmer** group and have the Protect Viewer license enabled.

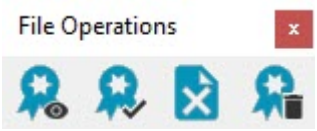
Recertify - Click this option to recertify the current measurement routine.

For complete information about these file operations, see the "Using Basic File Options" and "Using Advanced File Options" chapters in the PC-DMIS Core documentation.

Protected Mode File Operations Toolbar



File Operations toolbar for PC-DMIS-User



File Operations toolbar for PC-DMIS-Programmer

Edit Certified Routine - This button opens the certified measurement routine:

- If you are the creator of the measurement routine, this button opens the certified measurement routine in Edit Mode.
- If you are a member of the **PC-DMIS-Programmer** group and a measurement is submitted for recertification, this button opens the certified measurement routine in Protected Mode.
- If no measurement routine is available for recertification, PC-DMIS displays a message stating that you are not the creator of the measurement routine and it has not been submitted to you for certification.

Related Topics

Certify

Adding a User or a Group to the PC-DMIS-Programmer Group

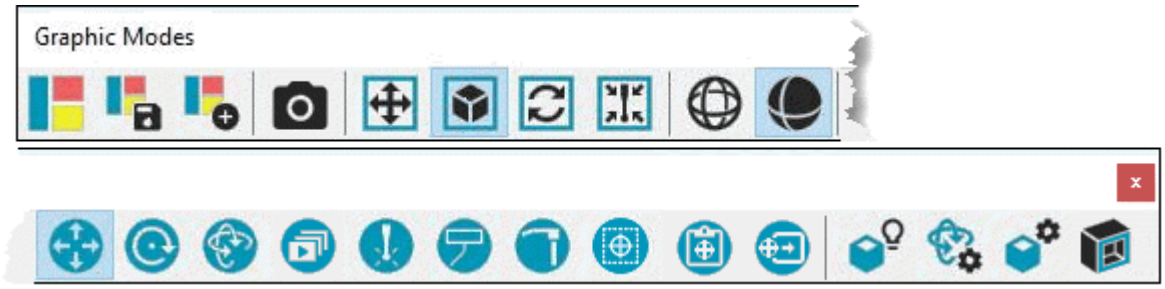
Single Level Authentication

Two Level Authentication

Edit Certified Measurement Routines

Procedure to certify and view the .protect file

Graphic Modes Toolbar



Graphic Modes Toolbar

You can use the **Graphic Modes** toolbar (**View | Toolbars | Graphic Modes**) to alter the display of the part on the screen. It also allows access to the available modes. This toolbar contains the following icons:

View Setup - Alters the display of the part in the Graphic Display window. See "Setting up the Screen View" in the "Editing the CAD Display" chapter.

Save Views - Saves the view of the part as the current viewset which can later be recalled. See "Working with Viewsets" in the "Inserting Report Commands" chapter.

Create Views - Saves the view of the part as a separate viewset command which can later be recalled. See "Working with Viewsets" in the "Inserting Report Commands" chapter.

Snapshot - Inserts a `SNAPSHOT` command of the current Graphic Display window state into the Edit window. When you execute this command, it inserts an image capture at that state into your report. See "Inserting Snapshots" in the "Inserting Report Commands" chapter.

Scale to Fit - Enlarges or shrinks the views of the graphic in the Graphic Display window to fit your screen. See "Scaling the Drawing" in the "Editing the CAD Display" chapter.

Draw Surfaces - Turns on or off the display of solid CAD surfaces. See "Drawing Surfaces" in the "Editing the CAD Display" chapter.

Repaint the Screen - Refreshes and redraws all the views of the part. See "Repainting the Screen" in the "Editing the CAD Display" chapter.

Center Probe - Centers the probe on the Graphic Display window screen so that the part is animated and the probe is always centered. See "Center Probe" in the "Editing the CAD Display" chapter.

Curve Mode - Places PC-DMIS into Curve Mode (for use with wireframe). See "Switching Between Curve and Surface Modes" in the "Editing the CAD Display" chapter.

Surface Mode - Places PC-DMIS into Surface Mode (for use with surface data). See "Switching Between Curve and Surface Modes" in the "Editing the CAD Display" chapter.

Translate Mode - Places PC-DMIS into Translate Mode. See "Translate Mode" in the "Editing the CAD Display" chapter.

2D Rotate Mode - Two dimensionally rotates the part for the selected view. See "2D Rotate Mode" in the "Editing the CAD Display" chapter.

3D Rotate Mode - Three dimensionally rotates the part for the selected view. See "3D Rotate Mode" in the "Editing the CAD Display" chapter.

2D Section Slide Show - If you have a Pointcloud or Mesh cross section in your measurement routine, this button toggles the 2D section slide show on and off. See "2D View of Cross Sections" in the PC-DMIS Laser documentation.

Program Mode - Uses the probe in either Offline or Online mode to learn and edit your measurement routine. See "Program Mode" in the "Editing the CAD Display" chapter.

Text Box Mode - Places PC-DMIS into a mode that lets you manipulate feature IDs, feature control frames, scan points, and CAD GD&T callouts. You can also use this mode to quickly create and modify Dimension Info and Point Info text boxes. See "Text Box Mode" in the "Editing the CAD Display" chapter.

Quick Fixture Mode - Enters Quick Fixture Mode so that you can move quick fixtures and the part by clicking and dragging them with the mouse. See "Inserting Quick Fixtures" in the "Defining Hardware" chapter.

GD&T Selection Mode (from CAD) - Provides the ability to box-select specific GD&T callouts that are embedded inside the part model and import them into your measurement routine. See "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.

It also provides a right-click shortcut menu to perform other operations on a callout. See "CAD GD&T Callouts Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" chapter.

GD&T Selection Mode (from File) - Shows an **Open** dialog box so that you can choose a drawing to import. You can then decide what GD&T tolerances to import using OCR. The software cycles through each GD&T tolerance on the drawing. See "Using GD&T Selection Mode to Create FCFs " in the "Using Geometric Tolerances" chapter.

CAD Lighting - Applies lighting, textures, and transparencies to CAD models. See "Setting up CAD and Graphics" in the "Editing the CAD Display" chapter.

Rotate Options - Changes how PC-DMIS displays items in the Graphic Display window during part rotation. See "Changing Rotation and Other Motion Options" in the "Editing the CAD Display" chapter.

Display Symbols - Defines the symbol style and size used to represent CAD points, scanned points, and feature points in the Graphic Display window. This opens the **Symbols** tab in the **CAD and Graphic Setup** dialog box. See "Modifying Display Symbols" in the "Editing the CAD Display" chapter.

Clipping Planes - Shows cross sections of your part model in the Graphic Display window. This opens the **Clip Planes** tab in the **CAD and Graphic Setup** dialog box. See "Clip Planes Tab" in the "Editing the CAD Display" chapter.

Graphic View Toolbar

Graphic View Toolbar

The **Graphic View** toolbar (**View | Toolbars | Graphic View**) changes how your part is displayed inside the Graphic Display window.

Many of these icons act the same as if you had used the **View Setup** dialog box to affect the view. For information on setting up your views, see the "Setting up the Screen View" topic in the "Editing the CAD Display" chapter.

If you configured the Graphic Display window to show more than one view, some icons affect only the blue pane in the **Views** area of the **View Setup** dialog box.

For example, if you divided your screen into three views by using this icon from the **View Setup** dialog box:

This toolbar contains the following icons:

Graphic Window in X+

Shows the X+ view of the part

Graphic Window in X-

Shows the X- view of the part

Graphic Window in Y+

Shows the Y+ view of the part

Graphic Window in Y-

Shows the Y- view of the part

Graphic Window in Z+

Shows the Z+ view of the part

Graphic Window in Z-

Shows the Z- view of the part

You can also click on a face of the Trihedron Rotation interface on the bottom left of the Graphic Display window to rotate the model and "square up" the view to that face. For more information, see "Using the Trihedron Rotation Interface".

Graphic Window in Isometric

Shows an isometric view of the part

Graphic Window Based on Probe Orientation

Pans, zooms, and rotates the view to look at the part along the probe axis of the current probe's tip angle

Show Graphic Window as Solid

Graphic Display window switches between a wireframe or solid view of the part

Surface Edges

Draws a bold line around edges of surfaces. These are called "surface edges". This icon becomes available when the **Solid** icon is selected. You can customize the color and size of surface edges through the **Wireframe** tab of the **CAD and Graphic Setup** dialog box. For information, see "Wireframe Tab" in the "Editing the CAD Display: Introduction" chapter.

Part Model Transparency

Turns transparency on or off for the part model. This marks or clears the **Enable transparency** check box on the **Lighting** tab of the **CAD and Graphic Setup** dialog box for the part model.

For information on the transparency settings on that tab, see "Lighting Tab" in the "Editing the CAD Display" chapter.

Toggle Graphic Display Window 3D

Displays or hides the 3D Grid in the Graphic Display window.

Graphic Items Toolbar

You can use the **Graphic Items** toolbar (**View | Toolbars | Graphic Items**) to show or hide graphic items in the Graphic Display window. These toolbar icons are available:

Show CAD - Shows the CAD model.

Show GD&T - Shows your CAD model's embedded CAD GD&T callouts or PMI (Product Manufacturing Information).

Show GD&T Notes - Shows your CAD model's embedded GD&T notes. You can only view these embedded CAD elements. You cannot import them as commands into the Edit window.

Characteristics IDs - Shows the characteristic IDs for any CAD GD&T callouts. You can only see a characteristic ID if its corresponding CAD GD&T is also visible (see **Show GD&T** above). For information on characteristic IDs, see the "Showing or Hiding CAD GD&T Characteristic IDs" subtopic under the "Working with CAD GD&T Callouts" topic in the "Editing the CAD Display" chapter.

Show Rotation Widget - Shows a small trihedron and cube in the bottom-left corner of each view of the Graphic Display window. You can click on the cube to rotate the CAD model to a face. You can double-click on the cube to both rotate and scale to fit the CAD model. For more information, see "Using the Rotation Widget" in the "Editing the CAD Display" chapter.

Show Feature Label IDs - Shows the IDs for feature commands that you added to the measurement routine.

Show Point Info Label IDs - Shows the label IDs for point information (Point Info) boxes.

Show DimInfo Label IDs - The label IDs for dimension info (DimInfo) boxes.

Show All Label IDs - Shows all the label IDs above.

Show All COPs - Shows all COPs and associated annotations.

Show All Meshes - Shows all Mesh features and associated annotations.

Activate Colormaps - Shows or hides the active colormap in the Graphic Display window.

- For details on Pointcloud Colormap operators, see the "Pointcloud Operators" section of the PC-DMIS Laser documentation.
- For details on the Mesh Colormap operator, see the "Creating a Mesh Operator" section of the PC-DMIS Laser documentation.

Show All Cross Sections - Shows all measured and nominal Cross Sections, and associated annotations.

Show Probe - Shows the animated probe's model.

Probe Component Draw Options - Shows the **Component Draw Options** dialog box with check boxes for each component. You can mark or clear a check box to show or hide that component respectively.

Show Machine - Shows the animated machine's model. This option is available if you use PC-DMIS in offline mode.

Show Probe Changer - Shows the animated probe changer's model.

Show Fixture - Shows any fixtures you've imported.

Show Clearance Plane - Shows any clearance planes you have defined.

Show ClearanceCube - Shows the ClearanceCube.

Edit Window Toolbar

Edit Window Toolbar

Use the **Edit Window** toolbar (**View | Toolbars | Edit Window**) to quickly access the many Edit window functions.

You can also use the toolbar to manipulate Edit window content in a way that is easy and intuitive to use. The toolbar provides ready access to these important Edit window commands. Some icons only appear if you have the proper configuration.

Execute - This icon runs (or executes) the measurement process for any currently marked features. For more information, see "Executing Measurement Routines" in the "Using Advanced File Options" chapter.

Auto Tune Execution - This icon allows you to teach illumination, magnification, and image processing parameters of your measurement routine commands for the target optical machine. This icon only appears if you have a vision configuration. For more information, see "Using AutoTune Execution" in the PC-DMIS Vision documentation.

Sequential Execution - This icon turns on a sequential execution mode, where execution does not proceed with the execution until it finishes calculating the current feature. For more information, see "Using Sequential Execution Mode" in the PC-DMIS Laser documentation. This icon only appears if you have a laser configuration.

Mark Current Feature - This icon highlights the feature under your cursor and prepares it for execution. For more information, see "Mark" in the "Editing a Measurement Routine" chapter.

Mark All - This icon highlights *all* features in the Edit window. PC-DMIS asks you if you want to highlight features used for the alignment as well. For more information, see "Mark All" in the "Editing a Measurement Routine" chapter.

Clear Mark - This icon removes all highlights from currently marked features within the Edit window. For more information, see "Mark All" in the "Editing a Measurement Routine" chapter.

Insert Breakpoint - This icon inserts a breakpoint at the current cursor location in the measurement routine. For more information, see "Using Breakpoints" in the "Editing a Measurement Routine" chapter.

Remove All Breakpoints - This icon removes any breakpoints from the measurement routine. For more information, see "Remove All" in the "Editing a Measurement Routine" chapter.

Set Start Point - This icon allows you to set the start point for the active learn arm. In the Edit window, click on the command that you want to use for the start point and then click on the **Start Point** toolbar icon. The start point for the current arm will be set to the

command located at the insertion point. You can also set a start point by right-clicking on a command and choosing **Set Start Point**. For more information on multiple arm mode, see the "Using Multiple Arm Mode" chapter. See "Setting Start Points" in the "Editing a Measurement Routine" chapter.

Insert Bookmark - This icon places a bookmark at the current cursor location in the measurement routine. For more information, see "Bookmark" in the "Editing a Measurement Routine" chapter.

Next Bookmark - This icon moves your cursor to the next Edit window location that contains a bookmark in the measurement routine. For more information, see "Next Bookmark" in the "Editing a Measurement Routine" chapter.

Clear All Bookmarks - This icon removes any bookmarks from the measurement routine. For more information, see "Clear All Bookmarks" in the "Editing a Measurement Routine" chapter.

Summary Mode - This icon places the Edit window in Summary Mode. This mode provides you with an expandable and collapsible graphical summary of groups and gage commands in your measurement routine. For more information, see "Working in Summary Mode" in the "Using the Edit Window" chapter.

Command Mode - This icon places the Edit window in Command Mode. This mode allows you to access the various commands that give you the capability to edit various aspects of the measurement routine. For more information, see "Working in Command Mode" in the "Using the Edit Window" chapter.

DMIS Mode - This icon displays the Edit window commands in DMIS format so you can work with the DMIS syntax. If you don't see this icon on your toolbar, select the **Use DMIS Button in Edit window** check box on the **General** tab in the **Setup Options** dialog box. For information on the **General** tab, see "Setup Options: General tab" in the "Setting your Preferences" chapter. For information on DMIS, see https://qifstandards.org/ and "Working in DMIS Mode" in the "Using the Edit Window" chapter.

Cut - This icon removes selected text and commands from the Edit window and stores that information temporarily in the Clipboard to be pasted later. For more information, see "Cut" in the "Editing a Measurement Routine" chapter.

Copy - This icon copies selected text and commands in the Edit window and stores that information temporarily in the Clipboard to be pasted later. For more information, see "Copy" in the "Editing a Measurement Routine" chapter.

Paste - This icon inserts information stored in the Clipboard into the Edit window at the cursor's current location. For more information, see "Paste" in the "Editing a Measurement Routine" chapter.

Define Pattern - This icon opens the **Pattern Setup** dialog box to define the pattern to use with **Paste with Pattern**. For more information, see "Editing Patterns of Features" in the "Editing a Measurement Routine" chapter.

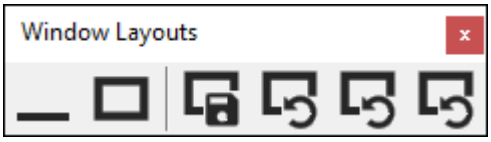
Paste with Pattern - This icon pastes patterned features with their offsets into the measurement routine at your cursor's current location. For more information, see "Editing Patterns of Features" in the "Editing a Measurement Routine" chapter.

Undo - This icon removes the last action that you took in the Edit window. For more information, see "Undo" in the "Editing a Measurement Routine" chapter.

Redo - This icon redoes an action that was removed with **Undo**. For more information, see "Redo" in the "Editing a Measurement Routine" chapter.

Print - This icon immediately sends a report of the current contents of the Edit window to your default printer without displaying any dialog boxes. This option is not available in Summary Mode.

Window Layouts Toolbar



Window Layouts Toolbar

You can use the **Window Layouts** toolbar (**View | Toolbars | Window Layouts**) to store the layout of any open windows, editors, and toolbars. You can click on a stored layout to restore it in your current measurement routine.

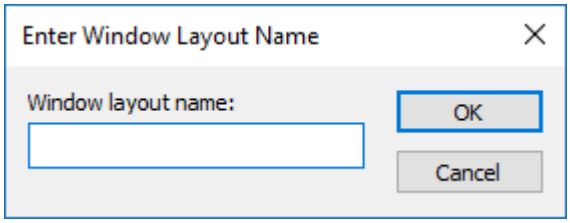
Layouts don't only store positions of toolbars and windows; they also store settings and options associated with the windows and toolbars. If a layout has a certain setting selected, and you then save that layout, the setting gets saved with the layout. If you later deselect that window setting, without overwriting the stored layout, the next time you select the layout, PC-DMIS restores the saved setting.

This toolbar contains the following icons:

Minimize - This icon minimizes the Edit window's size.

Maximize - This icon maximizes the Edit window's size.

Save - This icon displays the **Enter Window Layout Name** dialog box that allows you to save the current position and size of any open windows, editors, or toolbars under a desired layout name:



The **Window Layout Name** box specifies the name of the layout. PC-DMIS also displays this name as a tooltip for the icon associated with the layout. If you don't include a layout name, PC-DMIS uses a default name of "Recall Window Layout".

Once you click **OK**, the layout gets saved and an icon ( ) that represents that layout appears on the toolbar with your layout configuration.

You cannot overwrite an existing layout with a new layout by giving it the same name. If you want to overwrite an existing layout, first delete the existing layout, and then give it the same name.

Restore Saved Layout - This icon restores the stored layout. Any windows, editors, and toolbars are restored to their saved positions. Position your mouse pointer over the icon to show the name of the stored layout.

If you have multiple measurement routines open and have created a stored layout using one of the measurement routines, precedence is given to the measurement routine on which the layout was created. The measurement routine used to create the stored layout then becomes the active measurement routine and displays the stored window layout.

Deleting Saved User Layouts

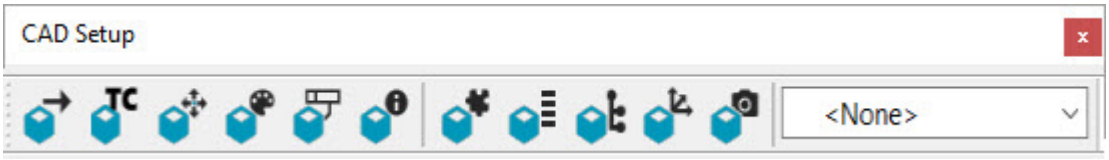
To delete a saved user layout, follow these steps:

1. Right-click on the toolbar area, and select **Customize**. The **Customize** dialog box appears.
2. Press and hold the Shift key on your keyboard.

3. Click and hold the mouse over the icon on the **Window Layouts** toolbar. The icon becomes movable.
4. Drag the icon outside of the toolbar's boundaries.
5. Release the mouse button.

The icon and the layout associated with the icon are deleted.

CAD Setup Toolbar



CAD Setup Toolbar

The **CAD Setup** toolbar contains these options to help you set up and work with a CAD model in the Graphic Display window:

Import from CAD File - Displays the **Import** dialog box so you can navigate to and import a supported part model from your library. Select the **Files of Type** list to view the file types. PC-DMIS remembers the file type that you selected the last time you used this button and defaults to that file type. For more information on importing files, see "Importing CAD Data or Feature Data" in the "Using Advanced File Options" chapter.

Import CAD from Teamcenter - Displays the **Teamcenter Open** dialog box so you can navigate to and import a CAD model from the Teamcenter database. For information on how to import a CAD model, see "Step 3 - Load Supported Models" in the "Using Teamcenter" chapter of the PC-DMIS DCI documentation.

CAD Transform - Shows the **CAD Transform** dialog box so you can transform (translate, scale, and rotate) your CAD model. For information on how to transform your CAD model, see "Transforming a CAD Model" in the "Editing the CAD Display" chapter.

CAD Colors - Shows the **Edit CAD Elements** dialog box so you can change the color of selected elements. For information on these CAD colors, see "Editing CAD Elements" in the "Editing the CAD Display" chapter.

CAD GD&T Options - Shows the **CAD and Graphic Setup** dialog box, **CAD GD&T** tab so you can configure how CAD GD&T callouts appear in the Graphic Display window.

For information on using this tab, see "CAD GD&T Tab" in the "Editing the CAD Display" chapter.

CAD Info - Shows the **CAD Information** dialog box so you can view additional information about the CAD model. For information on CAD information, see "Viewing CAD Information" in the "Editing the CAD Display" chapter.

Assembly - Shows the **CAD Assembly** dialog box so you can create a CAD assembly (a collection of part models), or display and hide parts inside an assembly. For information on CAD assemblies, see "Working with Assemblies of Parts" in the "Editing the CAD Display" chapter.

CAD Levels - Shows the **CAD Levels** dialog box so you can group CAD elements as CAD levels for the currently displayed CAD model. For information on CAD levels, see "Working with CAD Levels" in the "Editing the CAD Display" chapter.

CAD Groups - Shows the **CAD Groups** dialog box so you can group CAD elements as a CAD group. You can also use this to modify groups, delete groups, and view or hide groups in the Graphic Display window. For information on CAD groups, see "Working with CAD Groups" in the "Editing the CAD Display" documentation.

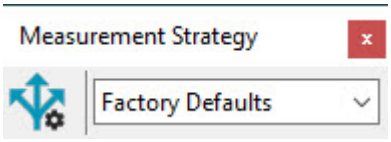
Coordinate System - Shows the **CAD Coordinate System** dialog box so you can create or select coordinate systems for your CAD model. For information on CAD Coordinate systems, see "Working with CAD Coordinate Systems" in the "Editing the CAD Display" chapter.

CAD Model Views - Shows the **CAD Model Views** dialog box so you can define and activate CAD model views. These views show and hide CAD elements similar to CAD groups, but they also include orientation, shading, scaling, and clipping plane information.

The list to the right of the icon contains the available CAD model views. If there are no CAD model views, the list is empty. You can select **<None>** from the list to deactivate a CAD model view. If a CAD model view contains embedded CAD GD&T callouts, this icon ( ) appears next to that item in the list. For information on CAD GD&T callouts, see the "Working with CAD GD&T Callouts" topic in the "Editing the CAD Display" documentation.

For information on CAD model views, see "Working with CAD Model Views" in the "Editing the CAD Display" documentation.

Measurement Strategy Toolbar



Measurement Strategy Editor - This button opens the **Measurement Strategy Editor** (MSE). For more information, see "Using the Measurement Strategy Editor" in the "Setting Your Preferences" chapter. This button appears if you enable the **Using Measurement Strategy Editor** check box in the list of check boxes in the **General** tab of the **Setup Options** dialog box (F5).

Measurement Strategy Group - The list to the right of the button lets you change the default measurement strategy group. A group is a saved set of modified settings for all the Auto features. The list shows all the available groups that you defined in the MSE. If you select a group from this list, that's the default group that PC-DMIS then uses. PC-DMIS uses that group in QuickFeatures and with the Measurement Strategy Widget. For more information, see "Creating QuickFeatures" and "Using the Measurement Strategy Widget" in the "Creating Auto Features" chapter. If the list only contains **Factory Defaults**, then you have not yet created any groups in the MSE.

Auto Features Toolbar

Auto Features Toolbar

With the **Auto Features** toolbar (**View | Toolbars | Auto Features**), you can access the same Auto feature available on the various tabs of the **Auto Features** dialog box.

PC-DMIS determines which Auto features are enabled based on your PC-DMIS configuration and your current probe type. For more information about what Auto features the different configurations support, see "Inserting Auto Features" in the "Creating Auto Features" chapter.

The following icons are available:

Vector Point - Creates an Auto Vector Point feature from the surface vector at the clicked location.

Surface Point - Creates an Auto Surface Point feature at the clicked location.

Edge Point - Creates an Auto Edge Point feature on a line between two surfaces.

Angle Point - Creates an Auto Angle Point feature where three perpendicular surfaces meet.

Corner Point - Creates an Auto Corner Point feature where two lines or surfaces meet.

High Point - Creates an Auto High Point feature at the highest location on the surface.

Line - Creates an Auto Line feature on a line.

Plane - Creates an Auto Plane feature on a surface.

Circle -Creates an Auto Circle feature on a circle.

Ellipse - Creates an Auto Ellipse feature on an ellipse.

Round Slot - Creates an Auto Round Slot feature on a round slot.

Square Slot - Creates an Auto Square Slot feature on a square slot.

Notch Slot -Creates an Auto Notch Slot feature on a three-sided square slot.

Flush and Gap - Creates an Auto Flush and Gap feature to measure the height difference between two mating sheet metal parts. *This becomes available with laser probes.*

Polygon - Creates an Auto Polygon feature from a feature of three or more sides of equal length.

Profile 2D - Creates an Auto Profile 2D feature. *This is available with vision probes.*

Blob - Creates an Auto Blob feature. *This is available with vision probes.*

Cylinder - Creates an Auto Cylinder feature.

Cone - Creates an Auto Cone feature.

Sphere - Creates an Auto Sphere feature.

When you click on an Auto feature icon, the **Auto Feature** dialog box opens and selects that feature type. To create the Auto feature, follow the instructions provided in the "Creating Auto Features" chapter. When ready, press the End key, or click the **Create** button on the **Auto Feature** dialog box to insert the feature into your measurement routine. Before creating your feature, you can remove hits from the hit buffer by pressing the Alt + '-' keyboard combination.

You can click the minimize button on the **Auto Feature** dialog box to *hide* it and still create the selected feature.

Measured Features Toolbar

Measured Features Toolbar

With the **Measured Features** toolbar (**View | Toolbars | Measured Features**), you can force PC-DMIS to accept a given feature type, or you can have it guess and display the feature type for which you are taking hits.

Point - Forces PC-DMIS to accept the Point feature type

Line - Forces PC-DMIS to accept the Line feature type

Plane - Forces PC-DMIS to accept the Plane feature type

Circle - Forces PC-DMIS to accept the Circle feature type

Round Slot - Forces PC-DMIS to accept the Round Slot feature type

Square Slot - Forces PC-DMIS to accept the Square Slot feature type

Cylinder - Forces PC-DMIS to accept the Cylinder feature type

Cone - Forces PC-DMIS to accept the Cone feature type

Sphere - Forces PC-DMIS to accept the Sphere feature type

Torus - Forces PC-DMIS to accept the Torus feature type

Set - Creates a Measured Set of features

Guess - Enables a Guess Mode where the software guesses the feature type based on the number of hits. PC-DMIS draws the guessed feature type in the Graphic Display window after you take each hit.

Guess Mode Preview

PC-DMIS provides a visual preview of the feature type before you actually accept it. For example, if you measure a cylinder, you need to take two sets of four hits, at two different levels along the height of the pin. When you take four hits at the first level, PC-DMIS draws a circle in the Graphic Display window. After you take the second level of hits, PC-DMIS displays two circles, one on each level, that represents the measured cylinder.

The feature is only added to the measurement routine when you press the End key. You can press the '-' or ALT + '-' keys to remove hits.

Forcing a Guessed Feature

If PC-DMIS displays the wrong feature type, you can click a specific feature icon to force PC-DMIS to accept that feature. For example, if you select the **Circle** icon, PC-DMIS is forced to accept the circle feature type as long as you take the minimum number of hits.

Constructed Features Toolbar

Constructed Features Toolbar

From the **Constructed Features** toolbar (**View | Toolbars | Constructed Features**), you can use icons to quickly access the feature construction options available from the **Insert | Feature | Constructed** submenu.

Point - Constructs a point from inputs. For more information, see Point.

Line - Constructs a line from inputs. For more information, see Line.

Plane - Constructs a plane from inputs. For more information, see Plane.

Circle - Constructs a circle from inputs. For more information, see Circle.

Ellipse - Constructs an ellipse from inputs. For more information, see Ellipse.

Round Slot - Constructs a round slot from inputs. For more information, see Round Slot.

Square Slot - Constructs a square slot from inputs. For more information, see Square Slot.

Width - Constructs a width from inputs. For more information, see Width.

Curve - Constructs a curve from inputs. For more information, see Curve.

Cylinder - Constructs a cylinder from inputs. For more information, see Cylinder.

Cone - Constructs a cone from inputs. For more information, see Cone.

Sphere - Constructs a sphere from inputs. For more information, see Sphere.

Surface - Constructs a surface from inputs. For more information, see Surface.

Feature Set - Constructs a feature set from inputs. For more information, see Feature Set.

Filter - Constructs a filter set from a scan, certain constructed features, or another filter set. For more information, see Filter Set.

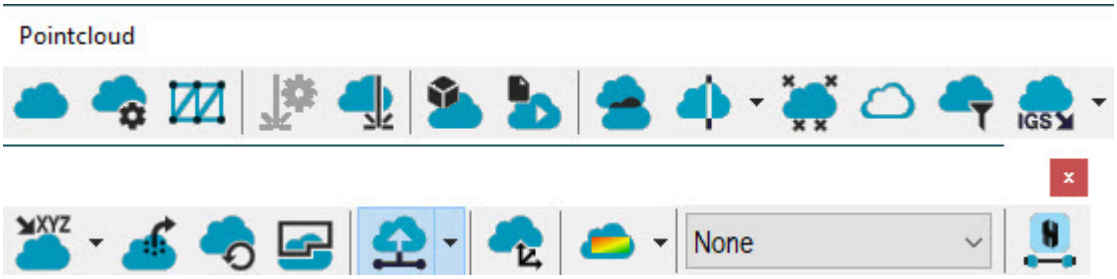
Adjusted Filter - Constructs an adjusted filter from adjusted scan data that was gathered while scanning around sphere, cone, or cylinder features. For more information, see Adjusted Filter.

Generic - Constructs a generic feature based on inputs but modified with expressions. For more information, see Generic.

For information on these items, see the "Constructing New Features from Existing Features" chapter and the "Creating Generic Features" chapter.

For information on inserting Read Points, view the documentation available in the "Creating Generic Features" chapter.

Pointcloud Toolbar

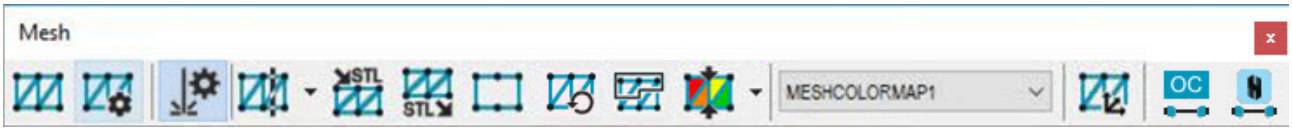


Pointcloud Toolbar

The **Pointcloud** toolbar provides all pointcloud operations, features, and functions. It is accessible from the **View | Toolbars | Pointcloud** menu, depending on your system's configuration.

For details on all Pointcloud toolbar functions, see the topic "Pointcloud Toolbar" in the PC-DMIS Laser documentation.

Mesh Toolbar



Mesh Toolbar

The **Mesh** toolbar provides all mesh operations, features, and functions. It is accessible from the **View | Toolbars | Mesh** menu depending on your system's configuration.

> You must be licensed for Mesh before it is available.

For details on all Mesh toolbar functions, see the topic "Mesh Toolbar" in the PC-DMIS Laser documentation.

Dimension Toolbar

PC-DMIS shows variations of the Dimension toolbar based on the GD&T standard you select.

Dimension toolbar - Shown with the "Use Legacy Dimensions" menu option enabled

Dimension toolbar - "Use Legacy Dimensions" menu option disabled, ISO 1101 2012/2017, ASME Y14.5 – 1994, or ASME Y14.5 2009 selected as the GD&T standard

Dimension toolbar – Use Legacy Dimensions disabled, ASME Y14.5 – 2018 selected as the GD&T standard

Toolbar options that are not compatible with the selected standard are disabled.

The **Dimension** toolbar (**View | Toolbars | Dimension**) is used to quickly access the options available from the **Dimension** menu. The icons for the dimensions below reference the "Using Legacy Dimensions" chapter. If you want to use Geometric Tolerances, see the "Using Geometric Tolerances" chapter.

Location - See Location

Distance - See Distance

Angle Between - See Angle Between

Straightness - See Straightness

Flatness - See Flatness

Circularity - See Circularity

Cylindricity - See Cylindricity

Perpendicularity - See Perpendicularity

Parallelism - See Parallelism

Angularity - See Angularity

Position - See Position

Concentricity - See Concentricity

Coaxiality - See Coaxiality

Symmetry - See Symmetry

Total Runout - See Total Runout

Circular Runout - See Circular Runout

Surface Profile - See Surface Profile

Line Profile - See Line Profile

Size - See Dimensioning Size

Keyin - See Keyin

Datum Definition - See Defining and Using Datums

Alignment Toolbar

Alignment Toolbar

With the **Alignment** toolbar (**View | Toolbars | Alignment**), you can access the same alignment options that are on the **Insert | Alignment** submenu.

The following alignment functions are available:

QuickAlign - Creates alignments from the Edit window with a single click of a toolbar icon. For more information on the QuickAlign functionality, see the "About QuickAlign" topic.

New Alignment - Displays the **Alignment Utilities** dialog box for creating a new alignment in your measurement routine. For information on using the **Alignment Utilities** dialog box, see the "Description of the Alignment Utilities Dialog Box" topic.

Save Alignment - Displays the **Save Alignment As** dialog box to save the current alignment in an external file. External alignments can be recalled by a different measurement routine. For more information on saving alignments, see the "Saving an Alignment" topic.

Recall Alignment - Recalls an alignment that was previously created in the current measurement routine (Internal Alignment) or saved from another measurement routine (External Alignment). For more information on recalling alignments, see the "Recalling an Existing Alignment" topic.

Equate Alignment - Changes the position or orientation of a part, or recovers the alignment when a part has been accidentally moved. For more information on equating alignments, see the "Equating an Alignment" topic.

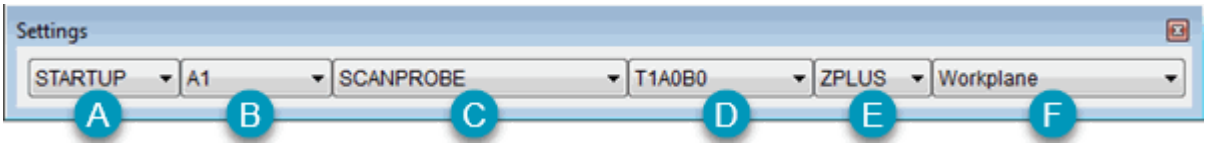
> The **Leapfrog** and **Bundle Alignment** functions below are only available when connected to a Portable system.

Leapfrog - Displays the **Leapfrog / Relocation** dialog box. This menu option is disabled in Offline mode. You must be connected to a physical supported portable

machine for the icon to be active. For more information on leapfrog operations, see the "Performing a Leapfrog Operation" topic.

Bundle Alignment - Creates a bundle alignment. Bundle alignments are used mainly in Portable systems for large or complex measurements to create a number of stations in a common network by moving the same sensor to different positions around the object. For more information on bundle alignments, see the "Using Bundle Alignments" topic in the PC-DMIS Portable documentation.

Settings Toolbar



Settings Toolbar

The **Settings** toolbar (**View | Toolbars | Settings**) contains lists that provide a more convenient way to switch between the some settings. From left to right on the toolbar, these settings are:

A. Views - Saved views of the Graphic Display window

With the **Views** list, you can recall a stored viewset that was created and saved with the active measurement routine.

PC-DMIS inserts a `RECALL/VIEWSET` command into the Edit window.

Command line in the Edit window:

```
RECALL/VIEWSET, view_set_name
```

View_set_name = the name of the view set to recall.

While PC-DMIS is recalling a view, all other functions are temporarily inaccessible.

For information on viewsets, see "Setting Up the Screen View" in the "Editing the CAD Display" chapter, and "Working with Viewsets" in the "Inserting Report Commands" chapter.

B. Alignments - Saved alignments

With the **Alignments** list, you can select a previously saved alignments. These alignments can then be inserted into the Edit window when selected.

See "Saving an Alignment" from the "Creating and Using Alignments" chapter for additional information on how to create and save alignments.

You can control which alignments are displayed in this list by going to the Alignment in the Edit window and changing the value for the `LIST=` portion of the `ALIGNMENT/START` command (for more information, see the "Creating and Using Alignments" chapter).

C. Probes - Defined probe types

With the **Probes** list, you can select probes already defined in your probe file. PC-DMIS inserts a `LOADPROBE` command into the Edit window.

PC-DMIS marks the currently active TProbe in the **Probes List** of the **Settings Toolbar** in a **bold** font weight. If you select a different probe from the list that isn't the physically active TProbe and then take a hit, the software displays a warning message. We recommended that you always use the probe settings of the physically active probe, otherwise your hit data might not be properly corrected for the ball diameter and offset.

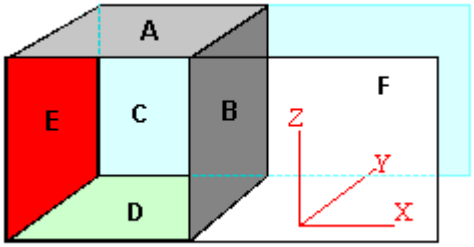
D. Probe Tips - Defined probe tips and probe tip angles

The **Probe Tips** list stores previously defined probe tip angles. See "Add Angles" from the "Defining Hardware" chapter for information on how to create tip angles.

E. Workplane - Available workplanes

The **Workplane** list allows you to change the plane in which you are working. This option does not change the view in the Graphic Display window; it changes only the workplane.

The available workplanes are:



Face	Color	Workplane
A=TOP	Light Gray	ZPLUS
B=RIGHT	Dark Gray	XPLUS
C=BACK	Light Blue	YPLUS
D=BOTTOM	Light Green	ZMINUS
E=LEFT	Red	XMINUS
F=FRONT	Clear	YMINUS

Measured Lines and Workplanes

When creating a measured line, PC-DMIS expects the hits for the line to be taken at a vector perpendicular to the current workplane.

For example, if your current workplane is ZPLUS (with a vector 0,0,1), and you have a block-like part, the hits for the measured line must be on a vertical wall of that part, such as the front or side.

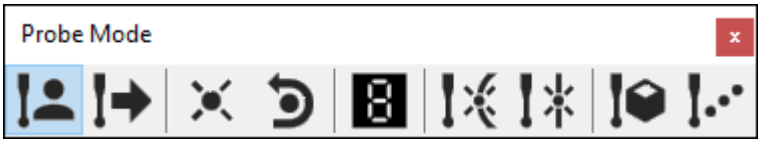
If you then wanted to measure a line feature on the top surface of the part, you would need to switch the workplane to XPLUS, XMINUS, YPLUS, or YMINUS, depending on the direction of the line.

F. Projection Planes - Planes (faces) to which to project certain features

The **Projection Planes** list contains a list of any plane features you have created in your measurement routine. By default, PC-DMIS projects each measured line and circle into your current workplane, so the default item on this list shows **Workplane**. However, you can select a different plane from this list.

PC-DMIS will then project each measured line and circle into the newly selected plane.

Probe Mode Toolbar



Probe Mode Toolbar

The **Probe Mode** toolbar (**View | Toolbars | Probe Mode**) contains icons that you can use to enter the different modes used by the current probe or CMM.

The following describes the available icons:

Manual Mode - This icon places PC-DMIS into Manual Mode. Manual Mode enables you to manually control your machine's movements and measurements. Manual Mode is used on a manual CMM or during the manual alignment portion of a measurement routine that is run on an automatic CMM. When you click this icon, PC-DMIS inserts a `MODE/MANUAL` command into the Edit window at the cursor's location. Edit window commands following this command are executed in Manual Mode.

DCC Mode - This icon places PC-DMIS into DCC Mode. DCC Mode allows supported DCC machines to automatically take over the measurement of your measurement routine. When you click this icon, PC-DMIS inserts a `MODE/DCC` command into the Edit window at the cursor's location. Edit window commands following this command are executed in DCC Mode.

Readouts Mode - This icon becomes available if you use an LK machine and if PC-DMIS is in Online mode. The icon places the probe into Readouts Mode. For more information, see "Placing the Probe into Readouts and Hits Mode" in the PC-DMIS CMM help documentation.

Hits Mode - This icon becomes available if you use an LK machine and if PC-DMIS is in Online mode. The icon places the probe into Hits Mode. For more information, see "Placing the Probe into Readouts and Hits Mode" in the PC-DMIS CMM help documentation.

Take a Hit - For supported probes in Manual Mode, this icon takes a hit at the probe ball location.

Erase Hit - This icon deletes the most recent hit from the hit buffer.

Probe Readouts - This icon the opens the Probe Readouts window. For information on probe readouts, see "Using the Probe Readouts Window" in the "Using Other Windows, Editors, and Tools" chapter.

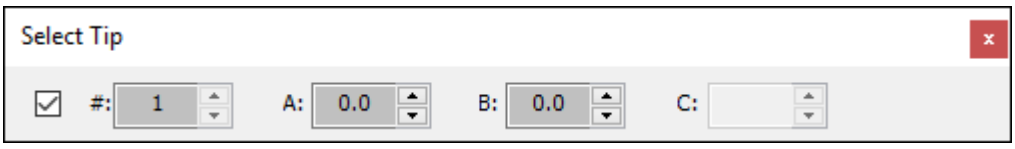
Point AutoTrigger Mode - For supported portable arm machines, this icon inserts a `POINT AUTOTRIGGER` command into the measurement routine. For more information on Point Auto Trigger Mode, see the "Point AutoTrigger area" topic in the "Setting Your Preferences" chapter.

Plane AutoTrigger Mode - For supported portable arm machines, this icon inserts a `PLANE AUTOTRIGGER` command into the measurement routine. For more information on Plane Auto Trigger Mode, see the "Plane AutoTrigger area" topic in the "Setting Your Preferences" chapter.

Find Nominals from CAD Mode - This icon selects the **Find Nominals** check box in the **General** tab of the **Setup Options** dialog box. For more information on that check box, see "Find Nominals" topic in the "Setting Your Preferences" chapter.

Point Only Mode - This icon selects the **Point Only Mode** check box in the **General** tab of the **Setup Options** dialog box. For more information on that check box, see "Point Only Mode" topic in the "Setting Your Preferences" chapter.

Select Tip Toolbar



Select Tip Toolbar

You can use the **Select Tip** toolbar (**View | Toolbar | Tip Selection**) to insert tips (for star probes) and tip angles (for configurations supporting probe head angles and wrist rotation) into your measurement routine on the fly. These can be constrained to allow only existing calibrated tips; or, you can insert any new angle that your probe hardware supports without having to go through the traditional route of accessing the **Probe Utilities** dialog box and then clicking on the **Add Angles** button.

For example, suppose you are learning a feature in Offline Mode, but a fixture holding the part in place won't allow the probe to measure the feature unless you use a different angle. Use this toolbar to quickly visualize and define the desired angle, which allows a much simpler workflow than the traditional procedure. To do this, type the desired angles (or use the up and down increment arrows). With each change in any of the edit boxes, PC-DMIS dynamically inserts the tip or tip angle into your measurement routine and appropriately draws the probe in the Graphic Display window.

The available items are as follows:

Check box - This check box constrains the toolbar, which allows you to select only calibrated tip angles. With this item selected, if you increment one of the A, B, or C angles, it chooses the closest calibrated tip angle and inserts it into the measurement routine.

If you clear this check box, you can define any tip angle increment that is valid for your probe type.

 # box - This box represents the tip number. It applies to probe types that don't have a rotating head or wrist but have multiple physical tips instead. For example, a star probe has five probe tips, each represented by a tip number. Type or select that number in this box to insert the associated TIP command into the measurement routine.

A, B, C boxes - These angle boxes define the A, B, and C angles for the probe head and wrist. Click the arrows to the right of each box to increment or decrement the respective angle to the next valid value.

Active Arms Toolbar

Active Arms Toolbar

The Active Arms toolbar (**Operation | Enter Multiple Arm Mode**) is used to manipulate multiple arms.

Arm1 Active - This corresponds to the primary arm. When selected, Arm1 is considered active.

Arm2 Active - This corresponds to the secondary arm. Only one arm can be active at a time. When selected, Arm2 is considered active.

Execute on Arm1 - This executes the measurement routine on Arm1.

Execute on Arm2 - This executes the measurement routine on Arm2.

Move to Arm1 Start Point - This moves the cursor in the Edit window to the top of the Arm1 measurement routine starting point.

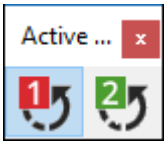
Move to Arm2 Start Point - This moves the cursor in the Edit window to the top of the Arm2 measurement routine starting point.

Execute from Start Point - This starts measurement routine execution for both arms at the currently defined start point. You can also execute commands tied to a specific arm by clicking the colored check mark icon that corresponds to the active arm.

The **Active Arms** toolbar is available if the Multiple Arm Mode add-on package has been purchased for your system.

See the "Using Multiple Arm Mode" chapter for more information on setting up and using multiple arm CMMs. Also, see the "Assigning a Command to an Arm" topic from that chapter for additional information on using this toolbar.

Active Rotary Table Toolbar



Active Rotary Table Toolbar

The **Active Rotary Table** toolbar (**View | Toolbars | Active Rotary Table**) selects which of the two rotary tables is the active table. This toolbar becomes available when you set up the rotary tables from the **Edit | Preferences | Rotary Table Setup** menu option. For more information, see "Defining the Rotary Table" in the "Setting Your Preferences" chapter.

ClearanceCube Toolbar

ClearanceCube Toolbar

Use the **ClearanceCube** toolbar (**View | Toolbars | ClearanceCube**) to enable or disable options related to ClearanceCube functionality. See "Using the ClearanceCube" in the "Editing the CAD Display" chapter.

The following icons are available:

ClearanceCube Definition - This icon displays the **ClearanceCube Definition** dialog box. For more details on the dialog box, see the "(Simple) ClearanceCube Definition" topic. You can also press Alt + C to open the dialog box.

Activate ClearanceCube Motion - This icon enables or disables ClearanceCube motion.

- If enabled, your measurement routine uses the ClearanceCube to control its motion and *disables clearance planes*.
- If disabled, your measurement routine uses existing clearance plane commands to control its motion.

> You cannot deactivate ClearanceCube if any feature in a measurement routine uses ClearanceCube motion.

Show ClearanceCube - This icon shows or hides ClearanceCube in the Graphic Display window.

To increase transparency through ClearanceCube in the Graphic Display window, select the **High quality transparency** check box in the **OpenGL** tab of the **CAD and Graphic Setup** (**Edit | Graphic Display Window | OpenGL**) dialog box.

ClearanceCube Settings window - This icon shows or hides ClearanceCube Settings window.

Wizards Toolbar

Wizards Toolbar

The icons on this toolbar are bound to external wizards that come installed with PC-DMIS. These wizards serve to increase the usability for new users who are still learning how to use the application.

These wizards are available:

3-2-1 Alignment Wizard - This wizard walks you through two basic types of alignments: a 3-2-1 alignment and a two circle alignment.

CAD Import Wizard - This wizard allows you to import single or multiple CAD models into the Graphic Display window. Click the icon and a dialog box appears. Select one or more files to import and then click **Open**. This replaces any CAD data that already exists in the window.

Data Field Edit Wizard - This wizard lets you replace certain fields in the Edit window that you cannot replace using the **Replace** dialog box. See "Editing Data Fields" in the "Editing a Measurement Routine" chapter.

DMIS Out Wizard - This wizard helps you export a DMIS output file of your inspection report.

Iterative Alignment Wizard - This wizard walks you through the steps to create an iterative alignment and explains the rules behind the inputs to the alignment.

Multiple Execution Wizard - This wizard lets you loop your measurement routine execution indefinitely or a specified number of times. You can also choose the specific message to display between iterations.

Probe Wizard - This wizard demonstrates how to define a probe and enables you to calibrate the newly defined probe.

Database Wizard - This wizard helps you create and register ODBC databases. This only appears if your license was configured with the **Integrated Stats** option.

Decremented Functionality

The PCD2Excel Wizard (PCD2excel.exe) is no longer supported and is no longer available from the user interface. It does not support Geometric Tolerance commands. Please use the Excel Form Report instead (**Insert | Report Command | Excel Form Report**).

For details on how to use the Excel Form Report, see the "Using the Excel Form Report Command" section of the PC-DMIS Toolkit Modules documentation.

While we recommend that you use the Excel Form Report above, the PCD2Excel Wizard is still available in the "*C:\Program Files\Hexagon\PC-DMIS 2026.2 64-bit\en*" folder of the PC-DMIS installation location.

The PCD2Excel Wizard will be removed in a future version.

PCD2Excel Wizard - This wizard exports your PC-DMIS measurement routine data into a Microsoft Excel file of your choice. To configure the wizard's options, click **Configure**. For information on the configuration options, see the "Configuring Excel Output" topic.

Excel 2003 or later is required for the PCD2Excel Wizard to function properly.

Configuring Excel Options

This topic covers the configuration options for the PCD2Excel Wizard.

The PCD2Excel Wizard (PCD2excel.exe) is no longer supported and is no longer available from the user interface. It does not support Geometric Tolerance commands. Please use the Excel Form Report instead (**Insert | Report Command | Excel Form Report**).

For details on how to use the Excel Form Report, see the "Using the Excel Form Report Command" section of the PC-DMIS Toolkit Modules documentation.

While we recommend that you use the Excel Form Report above, the PCD2Excel Wizard is still available in the "*C:\Program Files\Hexagon\PC-DMIS 2026.2 64-bit\en*" folder of the PC-DMIS installation location.

The PCD2Excel Wizard will be removed in a future version.

The available configuration options are:

Active measurement routine - Displays the name of your current measurement routine.

Output file - Defines the file name and location. The **File name** box defines the name of the Microsoft Excel file to which the output is sent. You can type the full pathway, or you can click **Browse** to locate and select an existing file name.

Type - Defines the output type. It contains these options:

> **Excel** - Sends the output to an Excel formatted file.
>
> **CSV** - Sends the output to a plain-text Comma Separated Value (CSV) file. You can choose a different delimiting character in **Separator**.
>
> **Show Excel** - If marked, Excel appears during the export process. If cleared, Excel runs hidden in the background.
>
> **Separator** - Determines the separator, or delimiting character, to use when this exports to a CSV file. The default is a comma character (,).

Overwrite mode - Defines what happens when data is sent to an existing file.

> **No** - Existing dimensional data is not overwritten. New values for existing dimensions are ignored. Only entirely new dimensions are added to the output file.
>
> **Yes** - Existing dimensional data is overwritten with the latest information.
>
> **Add** - Existing data is not overwritten with new values. The new values for existing dimensions are appended.

Export multiple instances - Determines the order in which multiple instances of a feature being executed appear in the exported file. Multiple instances result when a feature is executed multiple times due to a loop or some other execution branching.

> **By position** - Multiple instances are exported using their top-to-bottom position in the measurement routine.
>
> **By execution** - Multiple instances are exported based on the execution order of the last time the measurement routine was executed. For example, a branching conditional statement that skips over some dimensions during execution causes intervening dimensions to be ignored in this export as well.

PC-DMIS statistics - Determines if `PC-DMIS STATS/ON` or `TRACEFIELD` commands are required to export the data.

> **Use STATS/ON and STATS/OFF** - If marked, you *must* include dimension (or the Geometric Tolerance dimension) commands between the `STATS/ON` and `STATS/OFF` commands in your measurement routine; otherwise, the wizard does not export data to the Excel file. If you clear this check box, all dimension commands get exported into the Excel file. See the "Tracking Statistical Data" chapter.
>
> **Use TRACEFIELD** - This becomes available for selection if you use `STATS/ON` and `STATS/OFF` .

Headers - Determines how the header data for the measurement routine is written to the output file.

Measurement routine name - If marked, the measurement routine names are included in the output file.

Short - The file name is a shortened format and shows just the file name, not the full pathway.

Full - The file name includes the full pathway.

Part name - If marked, the part name is included in the output file.

Revision number - If marked, the revision number is included in the output file.

Serial number - If marked, the serial number is included in the output file.

Statistic count - If marked, the number of transactions or runs is included in the output file.

Data and time - If marked, the current date and time are included in the output file.

Dimension - Determines how the dimensional data appears in the output.

By row - Dimensions are laid out horizontally

By column - Dimensions are laid out vertically.

Header start row and column - Defines the starting location where you want the header information to appear. The values of 2 and 2 mean it begins the second row down and the second column to the right.

Data start row and column - Defines the starting location where you want the dimensional data to appear. The values of 9 and 2 means it begins on the ninth row in the second column.

Ignore FCFs - If marked, PC-DMIS does not pass Geometric Tolerance commands to the Excel file. See the "Using Geometric Tolerances" chapter.

Units - If marked, the measurement units used by each dimension are included in the export. For example, angle measurements show DEG for degrees, size measurements show MM for millimeters or IN for inches.

Fields label - Determines how labels appear next to data fields in the output.

Once - Field labels appear only once after the heading information and before all the dimensions are listed. These are either column headings or row headings based on the **By row** or **By column** setting.

Always - Field labels appear before each dimension.

All fields - If you mark **Always**, this check box becomes available for selection. If you mark **All Fields**, all the labels appear. These are: DIMENSION, DESCRIPTION, FEATURE, AXIS, SEGMENT, NOMINALS, MEAS, +TOL, -TOL, BONUS, DEV, OUTTOL, DEVANG, Datum Shift Effect, Unused Zone, Shift X, Shift Y, Shift Z, Rotation X, Rotation Y, and Rotation Z.

If cleared, only these labels appear: DIMENSION, DESCRIPTION, AXIS, NOMINALS, MEAS, +TOL, -TOL, DEV, and OUTTOL.

Blank row/column between dimensions - If marked, this inserts a blank row after each dimension.

Defaults - This button creates a defaults file (Defaults.p2x) that stores this dialog box's settings. When you click this button, PC-DMIS automatically updates the contents of the defaults file without showing a message.

PC-DMIS uses this defaults file in these cases:

- For new measurement routines
- For the current measurement routine if PC-DMIS cannot locate the routine-specific settings file (Note that PC-DMIS creates a routine-specific .p2x file in the same directory as the measurement routine whenever you first access this dialog box. It then modifies that routine-specific file whenever you click **Continue**.)

PC-DMIS attempts to save the Defaults.p2x file in a Wizards folder under where you installed PC-DMIS. You need to first create the Wizards folder inside the installation folder. If that folder doesn't exist, nothing happens when you click **Defaults**. You can view this defaults file in any text editor.

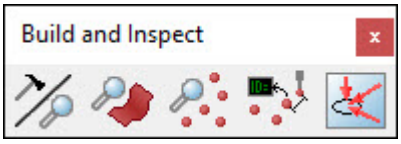
Running the Wizard Unattended

You can tell PC-DMIS to run the PCD2Excel wizard without any human intervention. After configuring the wizard, insert `EXTERNALCOMMAND` into your measurement routine and add an 'A' parameter to the command so that it looks something like this:

```
EXTERNALCOMMAND/DISPLAY ; C:\Program Files\Hexagon\PC-DMIS 2026.2 64-bit\en\Wizards\PCD2EXCEL.EXE A
```

This tells the wizard to run in unattended mode. If you're using a non-English version, instead of "en" in the pathway, use the two-letter subdirectory for your language.

Build and Inspect Toolbar



Build and Inspect Toolbar

This toolbar is used when measuring a part with a portable measurement device. For information on this toolbar, see the "Build and Inspect Toolbar" topic in the PC-DMIS Portable documentation.

QuickMeasure Toolbar

The **QuickMeasure** toolbar models the typical flow of operation for the software's various supported configurations. You can access it from the **View | Toolbars | QuickMeasure** menu.

The toolbar uses drop-down menus for many of the buttons. PC-DMIS stores the last-selected option for these buttons and displays it the next time you access the **QuickMeasure** toolbar.

If needed, you can add the drop-down menus to other toolbars from the **View | Toolbars | Customize** menu option. For information on how to customize toolbars, see "Customizing Toolbars" in the "Navigating the User Interface" chapter.

For CMM and Vision

From left to right, the icons on the **QuickMeasure** toolbar for PC-DMIS CMM or vision model the typical flow of operation on a system configured for CMM or vision.

QuickMeasure toolbar for CMM and vision users

For information on this toolbar, see "CMM QuickMeasure Toolbar" in the "PC-DMIS CMM" documentation.

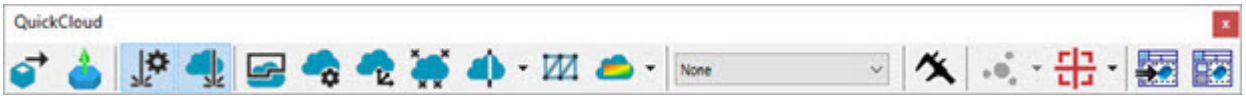
For Portable

From left to right, the icons on the **QuickMeasure** toolbar for PC-DMIS Portable model the typical flow of operation on a portable machine. This toolbar is only available for selection if your version of PC-DMIS is configured to work with a portable measuring device.

QuickMeasure toolbar for Portable users

For information on this toolbar, see the "Portable QuickMeasure Toolbar" topic in the "PC-DMIS Portable" documentation.

QuickCloud Toolbar



QuickCloud Toolbar

The **QuickCloud** toolbar is only available when PC-DMIS is licensed and configured for Portable devices. It provides the buttons to complete all the steps for working with pointclouds (COP).

For more information on this toolbar, see the "QuickCloud Toolbar" topic in the "PC-DMIS Portable" documentation.

For details on all Pointcloud toolbar functions, see the "Pointcloud Toolbar" topic in PC-DMIS Laser documentation.

Quick Fixture Toolbar

To show or hide the **Quick Fixture** toolbar, do one of the following:

- On the Graphic Modes toolbar, select the **Quick Fixture Mode** icon ( ).
- Select **Operation | Graphic Display Window | Change Screen Mode | Quick Fixture Mode**.

When the toolbar appears, PC-DMIS permanently docks the **Quick Fixture** toolbar vertically to the right of the Graphic Display window.

The toolbar provides drop-down functionality for some of the buttons. PC-DMIS stores the last selected option for each of these buttons and displays it the next time the software shows the **Quick Fixture** toolbar.

The **Drop**, **Level**, **Rotate**, and **Square** buttons are mode buttons. The active mode has a blue highlight. While a mode is active, you can apply it to any component by clicking that component in the Graphic Display window. The Quick Fixture gizmo, by contrast, acts on a single selected component. For gizmo-based moving, rotating, aligning, and dropping, see "Using Quick Fixture Mode".

Drop - Drops the selected quick fixture onto the object directly below it in the Graphic Display window. If there is nothing in the Z minus direction, then nothing happens. If there is, then the fixture drops down until it touches the object.

Level - Takes the normal surface vector of where you just clicked and squares that up with the machine coordinate system.

Rotate - Takes the point where you just clicked and drops that invisible point on to the closest edge. It then squares up the vector that is tangent to the curve at the drop point. The rotate is a 2D squaring, looking down the current view normal.

Square - Squares up the selected quick fixture so that its axes are parallel to the CAD axes shown by the trihedron.

Link - Enters a grouping mode that groups all on-screen fixture components together so that dragging or rotating one fixture drags or rotates them all together. When it is cleared, you can manipulate the fixture components individually.

Load QuickFixture file - Loads a previously saved QuickFixture (.qfxt) file into the Graphic Display window.

Save QuickFixture file - Saves the on-screen fixture components as a QuickFixture (.qfxt) file that you can reload in other measurement routines.

Export Draw file - Exports the on-screen fixture components as a .draw file. If there is more than one fixture on the screen, then PC-DMIS exports them all as a single set.

Delete - Deletes the selected quick fixture component from the Graphic Display window.

The per-axis Move and Rotate buttons (X, Y, Z, XY, YZ, ZX) that appeared on this toolbar in earlier versions have been removed. To move and rotate components interactively, use the Quick Fixture gizmo. For more information, see "Using Quick Fixture Mode".

Defining Hardware

Defining Hardware: Introduction

This chapter describes all of the options associated with defining the hardware used to measure parts. These include the definition and calibration of your probe with its probe tips; the creation and use of a virtual machine; the creation, positioning, and use of quick fixtures; the calibration of probe changers; the creation of animated probe changers; information on working with rotary tables; and configuring advanced mouse devices.

These are discussed in the following topics:

- Defining Probes
- Defining Machines
- Inserting Quick Fixtures
- Defining Probe Changers
- Showing an Animated Probe Changer
- Protecting Probes from Probe Changer Collisions
- Working with Rotary Tables
- Configuring Advanced Mouse Devices
- Working with Environment Builder

Defining Probes

One of the first steps in programming is to define which probe to use during the inspection process. Because probe definition and calibration processes are often different depending on your specific configuration of PC-DMIS, this information is instead covered in the documentation for your configuration. Consult the appropriate documentation below for information on setting up, calibrating, and using an appropriate probe for your specific situation:

- PC-DMIS CMM
- PC-DMIS Vision
- PC-DMIS Laser

- PC-DMIS Portable

You use the **Probe Utilities** dialog box to define probes. You can also manually define probes by editing the usrprobe.dat file.

For Maestro CMM users, see the "Working with Environment Builder" section of the "Defining Hardware" chapter on how to use the Environment Builder to define your probe assemblies.

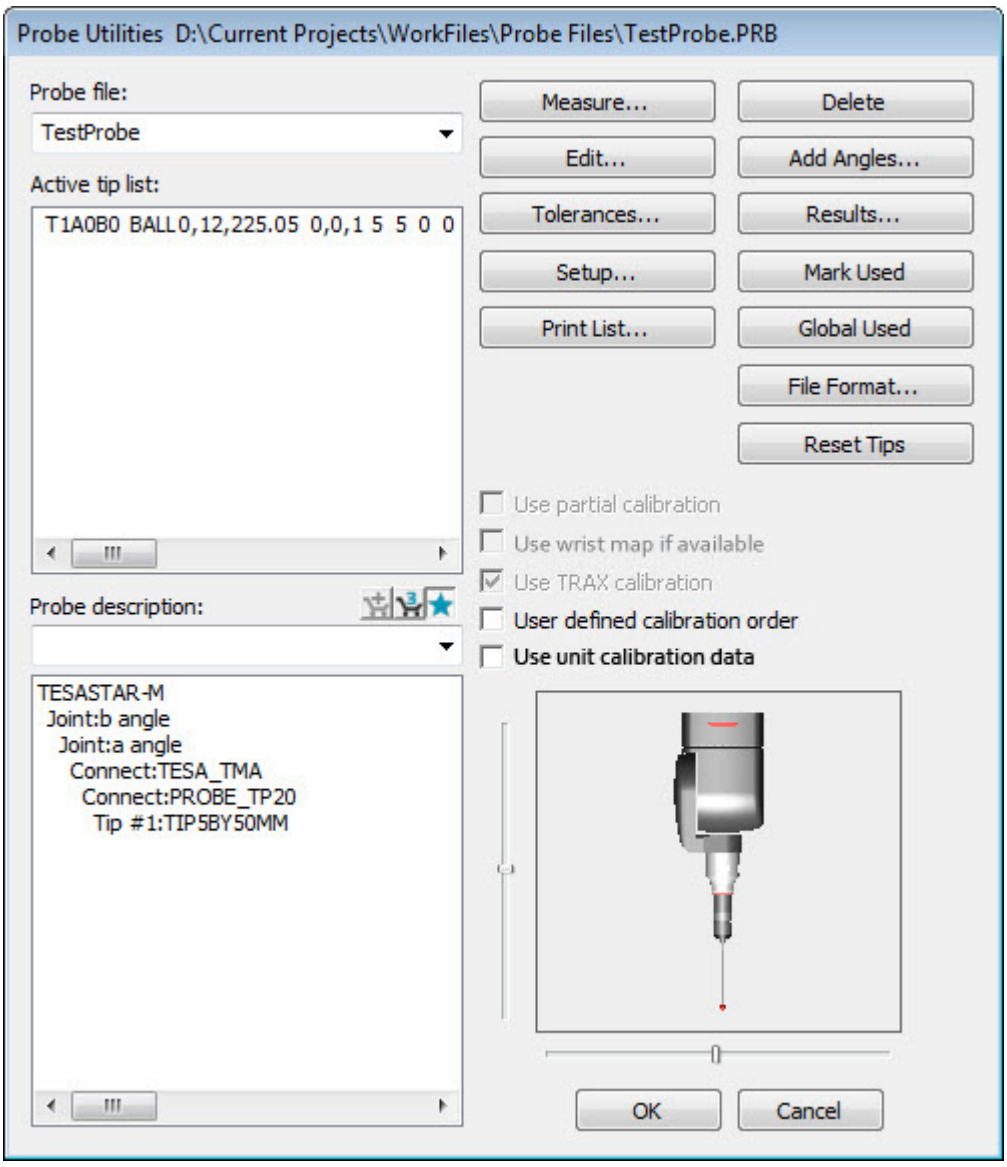
Click this icon from the **Wizards** toolbar to access the PC-DMIS Probe Wizard.

You can use the Check Calibration Limits command to check for errors and raise alerts during execution. For more information on this command, see "About Check Calibration Limits" in the PC-DMIS Toolkit Modules documentation.

You can enable the Leg Collision check. For more information, see "CMM Limits area".

Understanding the Probe Utilities dialog box

The **Probe Utilities** dialog box displays probe data for the active tip. You can use this dialog box to create new probe files, access previously-defined files, and edit a probe file. You can also calibrate probes. You can also select the Load Probe command of your measurement routine in the Edit window and then press F9 to display the **Probe Utilities** dialog box. To display the dialog box, select **Insert | Hardware Definition | Probe**.



Probe Utilities Dialog Box

The **Probe Utilities** dialog box image is intended to show all possible options and is for reference only. The options that appear depend on your PC-DMIS license settings, the PC-DMIS product being run, and how the measurement routine is written. Not all options are available at the same time.

Probe File Name

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Probe file** list displays the probe file loaded for your current measurement routine. To load a different probe file, click the drop-down arrow to display all the probe files saved. The probes are listed in alphabetic order, making it easy to locate a specific probe.

PC-DMIS stores probe files in a default directory when you install PC-DMIS. When PC-DMIS searches for a probe file to load, it searches in this directory unless you've changed the search path. For information, see "Specifying Search Paths" in the "Setting your Preferences" chapter.

PC-DMIS stores the probe files by default in the "C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2" folder. This is detailed in the "Understanding File Locations" topic unless you've changed the search path.

To create a new probe file:

1. Highlight the current name in the **Probe file** drop-down list.
2. Type in a new name.

Use only letters, numbers, and the underscore character (_) in a probe file name. PC-DMIS does not allow a period or any of these characters: < > : " / \ |

Because PC-DMIS does not allow a period, do not type the .prb extension. PC-DMIS adds it for you.

A probe file name is limited to 180 characters. If you type a longer name, PC-DMIS displays a message and shortens the name to 180 characters.

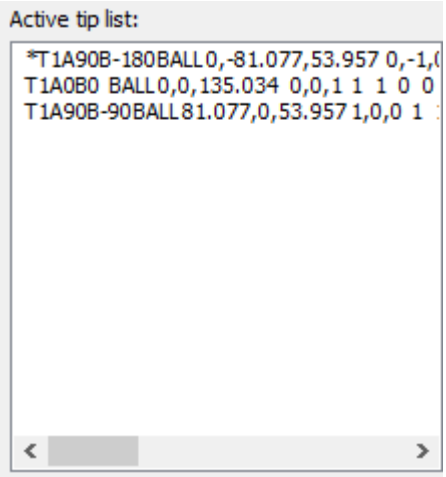
For the full list of naming rules, see "Setup Options: ID Setup tab" in the "Setting Your Preferences" chapter.

If a probe file has already been saved under the keyed-in name, PC-DMIS loads the previously saved file into the current measurement routine.

Active Tip List

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

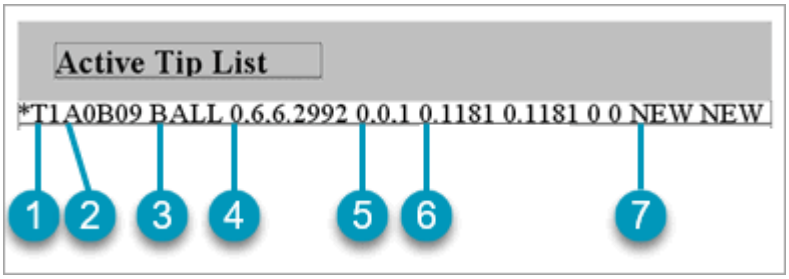
PC-DMIS is capable of storing data that describes a large number of probe tips. This data includes the probe tip's ID, rotation, type, location, direction, diameter and thickness, date and time of calibration, and any non-calibrated tips. These are all stored inside the **Active tip list**.



Active tip list Area

The list can display up to 32767 probe tips. This number may be limited by the amount of disk space available on your system.

PC-DMIS describes a probe according to the following criteria:



Tip List Description

1. ***TIP ID#*** *- This ID is the permanent number that PC-DMIS assigns to a tip when it is loaded into memory.*
2. ***Tip Rotation*** *- This field displays the rotation of the tip in the vertical (A) and horizontal (B) direction*
3. ***Tip Type*** *- This field displays the probe type (BALL, DISK, TAPER, SHANK, OPTICAL).*
4. ***X,Y,Z Location*** *- These values describe the location of the tip. This location is in relation to the bottom of the Z rail.*
5. ***I,J,K Direction*** *- These values describe the direction of the probe tip. This vector goes from the center of the probe tip towards the Z rail.*
6. ***Diameter and Thickness*** *- These values describe the diameter of the tip and the thickness of SHANK and DISK probes. PC-DMIS defines these values when the probe is loaded. (To change a field, see "Edit Tool".)*
7. ***Date and Time*** *- These boxes indicate the most recent date and time the probe tip was calibrated. If a new tip is created without being calibrated, PC-DMIS will display "NEW" for the date and time values. If an old probe tip is loaded and the date and time information is unavailable, PC-DMIS will display "UNKNOWN" for the values. Only probe tips that are actually calibrated have their time and date values updated.*

***(asterisk) -** An asterisk (*) to the left of the tip identifies any noncalibrated tips.*

Adding Tips to the List

You can use the **Add Angles** button to define new tips and add them to the list. To do this, see the "Add Angles" topic.

Editing Tip Data

1. Highlight the desired active tip in the **Active tip list**.
2. Click the **Edit** button to show the **Edit Probe Data** dialog box.
3. Modify the displayed values.
4. Click **OK** to save your changes.

Non-calibrated tips have an asterisk before their Tip ID # in the **Active tip list**.

Setting the Calibration Order

The calibration order is determined by the order in which the tips are chosen from the list box.

To set the calibration order:

1. From the **Probe Utilities** dialog box, click **Measure** to show the **Measure Probe** dialog box.
2. From the **Measure Probe** dialog box, **Calibration Mode** area, select the **User Defined** option. (See the "Measure" topic.)
3. Hold down the Ctrl key on the keyboard.
4. With your left mouse button, select the tips from the **Active tip list** for calibration. A number representing the index of the tip measurement order appears next to each Tip ID as you select it.

If you do not select any tips, PC-DMIS prompts if you want to measure all tips.

Selecting a Tip to Use

You can define a specific probe tip to use in your measurement routine in these ways:

- Type TIP in the Edit window's Command Mode and press the Tab key on your keyboard.
- Select the probe tip from the list on the Settings toolbar.

The Edit window command line for a sample tip would read:

```
TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
```

Until PC-DMIS encounters another TIP command in the measurement routine flow, it uses that tip.

Deleting a Tip or Wrist Angle

To delete one or more tips from the **Active tip list**:

1. Select the item or items to be removed.
2. Click the **Delete** button.

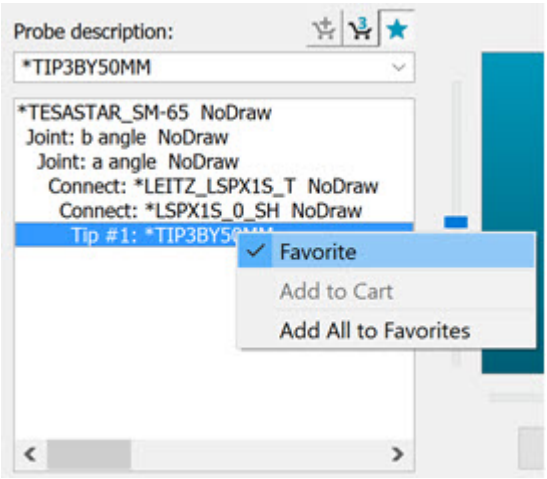
PC-DMIS also allows specific wrist angles to be removed from the **Active tip list**. If you delete a tip or probe head, PC-DMIS also deletes all related wrist angles from the file.

This option deletes the tip from the **Active tip list** and removes it from the file of tips known to the system.

Probe Description

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Probe description** area consists of the list of available probe options in alphabetical order and a box that displays the probe components.



Probe description Area

From this area, you can do the following:

- Define the probe, extensions, and one or more tips that you can use in the measurement routine.
- Create a list of your favorite probe components.
- Add the probe and its components to the PC-DMIS shopping cart. You can then purchase the items from the Hexagon eStore or generate a purchase request.

You can right-click on a probe component to display these menu options:

Favorite - This item adds the component to your list of favorites. To add a component to the list, select the component, right-click, and then select **Favorite**. The check mark beside the menu item indicates that it is already in the list. To remove a component from the list, right-click on the added component and select **Favorite**.

The FavoriteProbeHardwareComponents.xml file saves the list of your favorite probe components. The default folder location is:

C:\ProgramData\Hexagon\PC-DMIS\<version>

Where <version> is the PC-DMIS version.

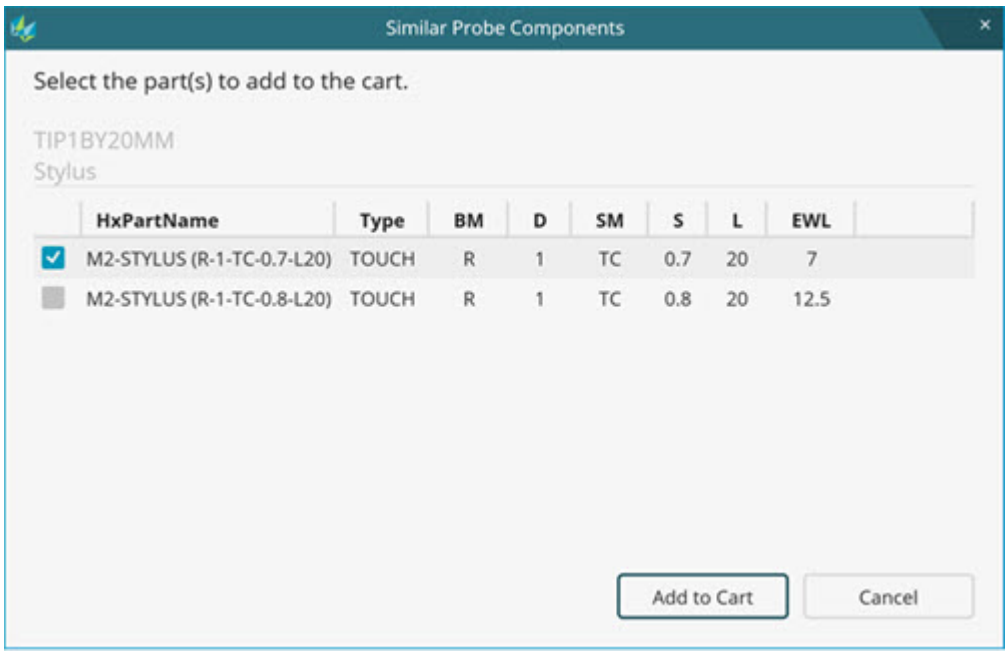
If you want to use the same list of favorites on your other computers, you need to copy the FavoriteProbeHardwareComponents.xml file from this computer and overwrite the file in all other computers.

Add to Cart - This item adds the component to the PC-DMIS shopping cart. This item is available only if the probe component has a Hexagon part number. This item is not available if the component has already been added to the cart.

Add All to Favorites - This item adds all the components of the selected probe file to your list of favorites.

Once you add the component to your list of favorites, the asterisk character (*) appears before the name of the component.

Add to Cart - To add a component to the cart, select the component and then click this icon. This icon is available only if the probe component has a Hexagon part number. For a selected stylus or extension of specified length, multiple part numbers might be available in the Hexagon catalog. The part numbers can be stylus or extensions made from different material or stem diameters. PC-DMIS checks for similar parts. If there are any available, PC-DMIS displays a menu to select the required parts.



Similar Probe Components Dialog Box

Mark the check box to select the required parts. To add the selected parts to the cart, click **Add to Cart**. To increase the quantity of the same part, click **Add to Cart** again. With each click, the quantity increases by one. Click **Cancel** to close the dialog box without adding anything to the cart.

In the top-right of the dialog box, you can click the icon to display the legend table which you can use to view material and technical terminology.

Material

AL - Aluminum

CE - Ceramic

CF - Carbon Fiber

DA - Dispal

DC - Diamond Coated

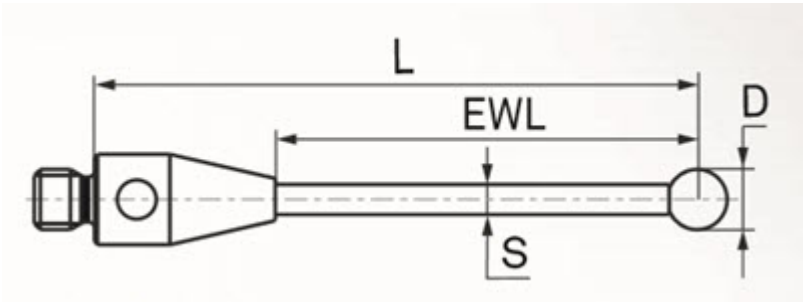
DS - Diamond Sphere

R - Ruby

SN - Silicon Nitride

SS - Stainless Steel

Technical



L - Length (mm)

D - Diameter (mm)

D1 - Extension End Pieces Diameter (mm)

S - Stem (mm)

SP - Span (mm)

EWL - Effective Working Length (mm)

View/Edit Cart - This icon displays the number of components in the cart. Click the icon to open the **PC-DMIS Shopping Cart** dialog box. For information on this dialog box, see "Shopping Cart".

Favorites - This toggle icon displays either your list of favorites or the complete list of probe options. If you select this icon, the list displays only the favorites that are available at the connection point. If you do not select the icon, the list displays all of the probe options.

Edit Probe Components

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

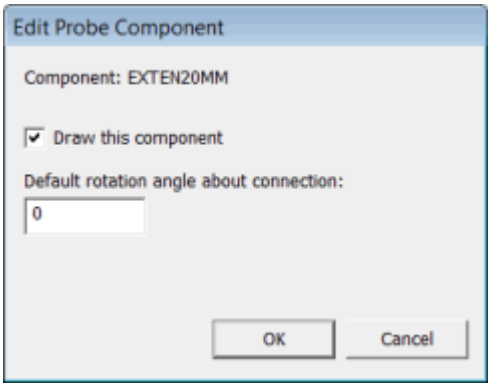
You can double-click on a component in the **Probe description** area (in the Probe Utilities dialog box) to open the **Edit Probe Component** dialog box. You can use this dialog box to hide some probe graphics to provide a better view of the features. This is useful in cases where the part geometry is particularly dense.

You can also do this from the **Graphic Items** toolbar with the **Probe Component Draw Options** button . For more information see "Showing and Hiding Probe Components".

The options that appear in this dialog box depend on the component you selected.

Default Options

The following two options apply to all components:



Edit Probe Component dialog box

Draw this component - If you select this check box, PC-DMIS draws the component in the Graphic Display window.

PC-DMIS organizes the different components in a hierarchical list. This means:

- When you choose to hide a component, PC-DMIS also hides any components above that component.
- When you choose to show a component, PC-DMIS also shows any components below that component.

Default rotation angle about connection - This value defines a rotation angle about the connection. The rotation angle is primarily used to manually define the angle of knuckle attachments. You can type any angle from +180° to -180°. 0 is the default angle.

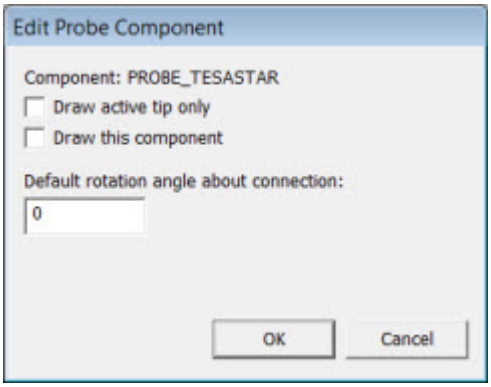
To hide probe graphics:

1. In the **Probe description** area, locate the part of the probe that you want to hide from view.
2. Double-click on that probe component to show the **Edit Probe Component** dialog box.
3. Clear the **Draw this component** check box.
4. Click the **OK** button to close the **Edit Probe Component** dialog box. Notice that PC-DMIS redraws the probe graphics without the indicated component and all other components above it in the **Probe Utilities** dialog box.

Additional Option for Star Probe Components

If you have a star probe, you can make it so that only the current tip is visible in the Graphic Display window. You can do this with the **Draw Active Tip only** check box in the **Edit Probe Component** dialog box.

To see the check box, from the **Probe Description** area, double-click on the probe head component.

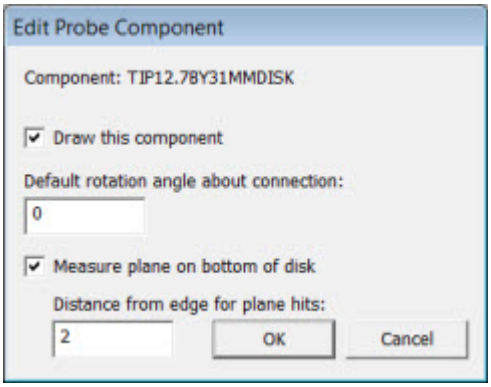


Edit Probe Component dialog box - Star Probe

Draw active tip only - If you select this check box, PC-DMIS hides any non-active probe tips instead of merely highlighting the active probe tip in the Graphic Display window. If you clear this check box, PC-DMIS highlights the active probe tip as usual.

Additional Options for Disk Stylus Components

For calibration of a disk stylus when used with an analog probe, two additional options are included.



Edit Probe Component dialog box - Disk Stylus

Measure plane on bottom of disk

- If you select this check box, four hits are taken on the bottom of the disk to calculate a plane and therefore determine a measured vector associated with the plane of the disk.
- If you clear this check box, PC-DMIS does not take any plane hits, and there is no measured vector. The vector for the plane of the disk is the theoretical value from the model of the probe.

  For a new probe, the default value is determined by the `ProbeQualAnalogDiskUsePlaneOnBottom` entry in the **ProbeCal** section of the PC-DMIS Settings Editor. If this entry is set to 1, PC-DMIS measures the plane. If this entry is set to 0, PC-DMIS does not measure the plane.

Distance from edge for plane hits

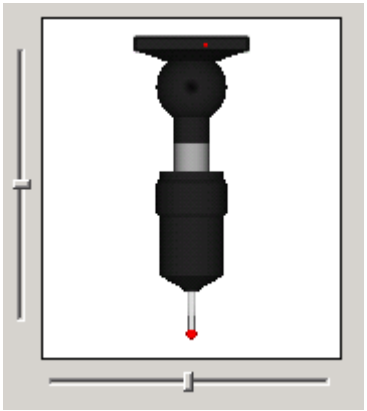
When you measure the plane, PC-DMIS distributes the hits in a circular pattern from this distance back from the outer edge of the disk. The radius for the pattern of the plane hits is the disk radius minus this distance.

For a new probe, the default value is determined by the `ProbeQualAnalogDiskBottomHitsDistanceFromEdge` entry in the **ProbeCal** section of the PC-DMIS Settings Editor. The units for this entry are always in millimeters. However, the value that you enter in the **Distance from edge for plane hits** box is in the units of the measurement routine that you are using (that is, either inches or millimeters).

> In 2012 MR1 and higher, the `ProbeQualAnalogDiskBottomHitsDistanceFromEdge` and `ProbeQualAnalogDiskUsePlaneOnBottom` entries provide the initial defaults for new probes. You can then define specific settings for individual probes in the **Edit Probe Component** dialog box.

Previewing your Probe Configuration

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)



Graphical view of a probe and sliders

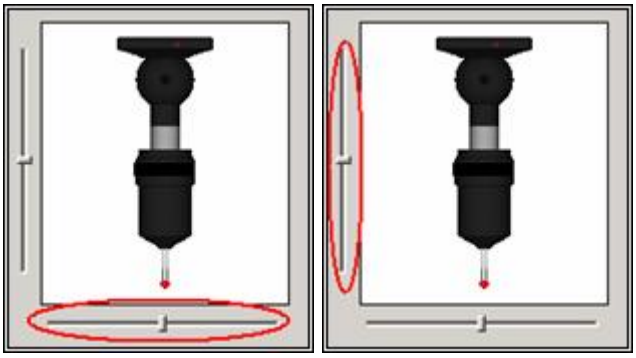
The graphical view of the **Probe Utilities** dialog box provides you with the ability to preview graphically:

- The components of your probe.
- The various AB angle positions in the **Active tip list** box.
- A complete 3D rotation of the probe.

Viewing Components: Once selected as a part of the probe, a component of the probe's configuration automatically appears in the graphical view of the **Probe Utilities** dialog box.

Viewing AB Angle Positions: Select an AB angle position from the **Active tip list**, and the graphical view of the probe dynamically changes to match what the current probe configuration looks like with the selected AB angle.

Rotating the Probe in 3D: Move the sliders located below and to the left of the probe's graphical view to rotate the view of the probe. The slider below rotates the probe horizontally. The slider to the left rotates the probe vertically.



Horizontal Slider, rotates display horizontally (left), and Vertical Slider, rotates display vertically (right)

Available Tip Types

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

PC-DMIS provides the following types of tips in the **Probe description** list:

If a tip needs to be added to the list of available tips, contact Hexagon Technical Support. Only authorized personnel are allowed to create additional tip files.

Ball - This defines a spherical probe. The user is able to edit the nominal diameter and thickness of the probe using the **Edit** button. The direction of the probe must also be defined.

Disk - This defines a disk probe. The user is able to edit the nominal diameter and thickness of the probe using the **Edit** button. The direction of the probe must also be defined.

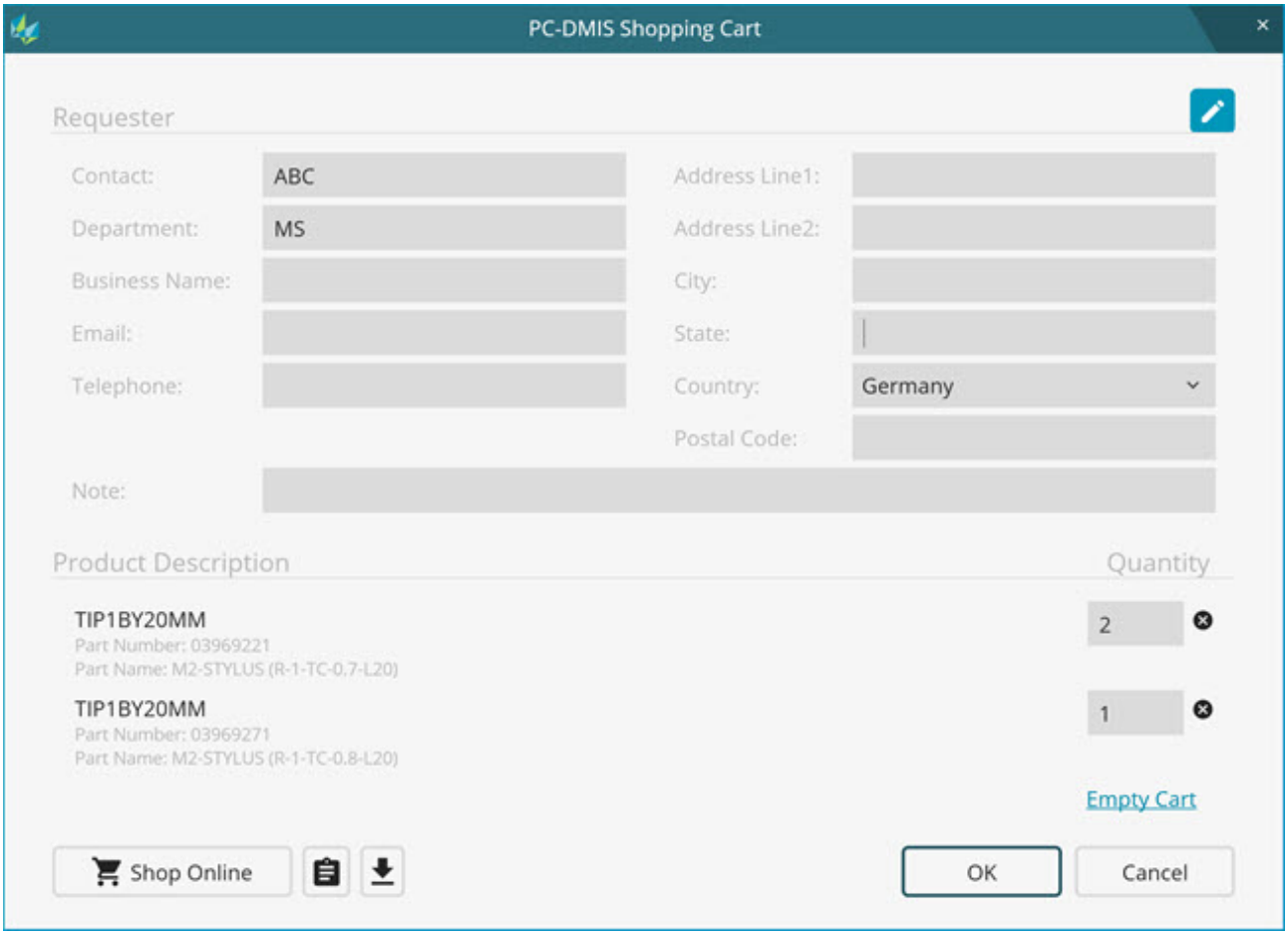
Optical - This option is only available if the optical tip is defined as a hard probe. This option defines an optical probe. The user is able to edit the nominal diameter of the probe using the **Edit** button. The direction of the probe must also be defined.

Shank - This defines a shank or barrel probe. The user is able to edit the nominal diameter and thickness of the probe using the **Edit** button. The direction of the probe must also be defined.

Shopping Cart

PC-DMIS lets you add the probe components to the shopping cart from the **Probe Utilities** dialog box.

To view and edit the shopping cart, click the **View/Edit Cart** icon to open the **PC-DMIS Shopping Cart** dialog box.



PC-DMIS Shopping Cart Dialog Box

Requester - This area displays the requester details, such as the contact person's name, department, address, and so forth. To edit the details, click the **Edit** icon .

Product Description - This column displays the probe's component name, part number, and part name (if available).

Quantity - This box sets the number of components that you want to include in your purchase request or order. To remove the component from the cart, click the **Remove** icon .

Empty Cart - This link deletes all the items from the shopping cart.

Shop Online - This button completes the online shopping if an online store is available for the country where you want to ship your part. Once you click this button, PC-DMIS creates

and opens the link in your default browser. You can then see your selected items in the cart and complete your purchase.

Copy web address into clipboard - This button lets you copy the web address on the clipboard. If your computer is not connected to the internet, you can copy the link and then send it to the computer where internet is available to shop online.

Download Purchase Request - This button opens the **Save As** dialog box to let you save a purchase request in a Portable Document Format (.pdf). You can use the purchase request for further approval process in your organization.

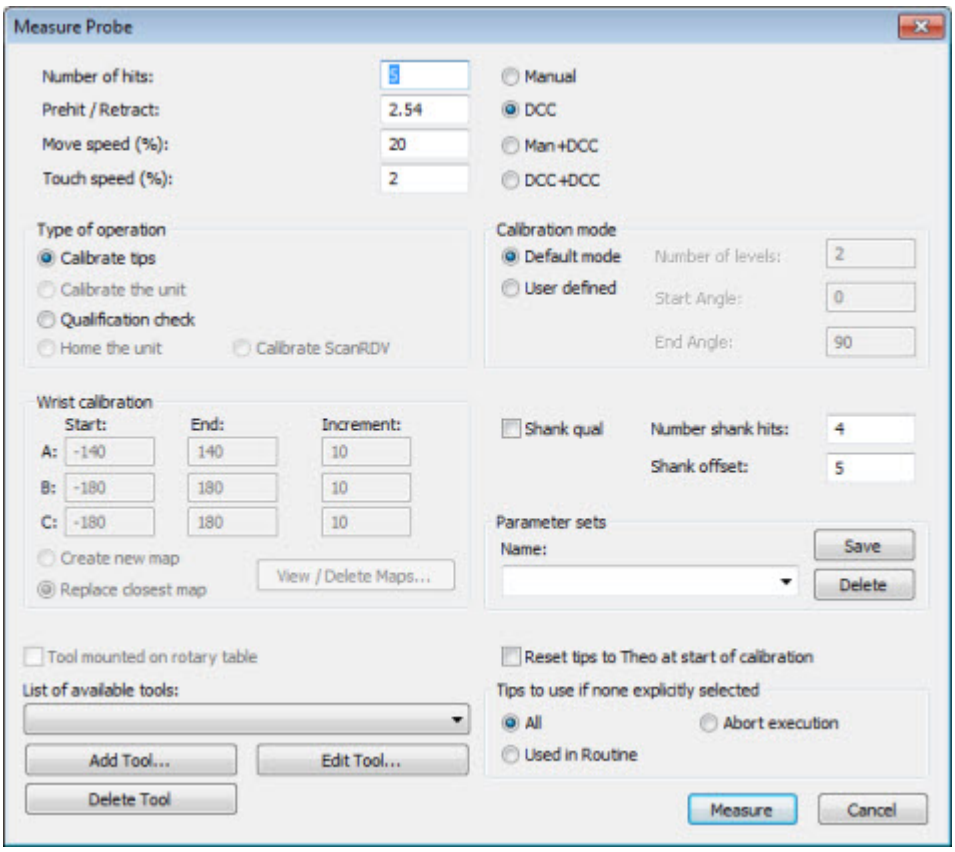
OK - This button saves the changes made to the shopping cart.

Cancel - This button closes the dialog box without saving any changes.

Measure

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Measure** command button allows you to calibrate probe tips selected from the **Active tip list** of the **Probe Utilities** dialog box. When you click this button, the **Measure Probe** opens.



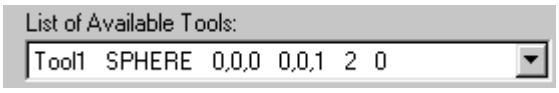
Measure Probe Dialog Box

For information on the options in this dialog box and on how to calibrate probe tips, see "Calibrating Probe Tips" in the PC-DMIS CMM documentation.

For information on adding, editing, and deleting calibration tools, see the topics below.

List of Available Tools

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)



The **List of Available Tools** drop-down list displays the available calibration tools. PC-DMIS uses probed hits on the calibration tool to calibrate the selected probe. PC-DMIS only uses spherical tools.

The type of tip you calibrate may affect where and how you take hits on the calibration tool. Consider what happens during calibration with these different probe tip types:

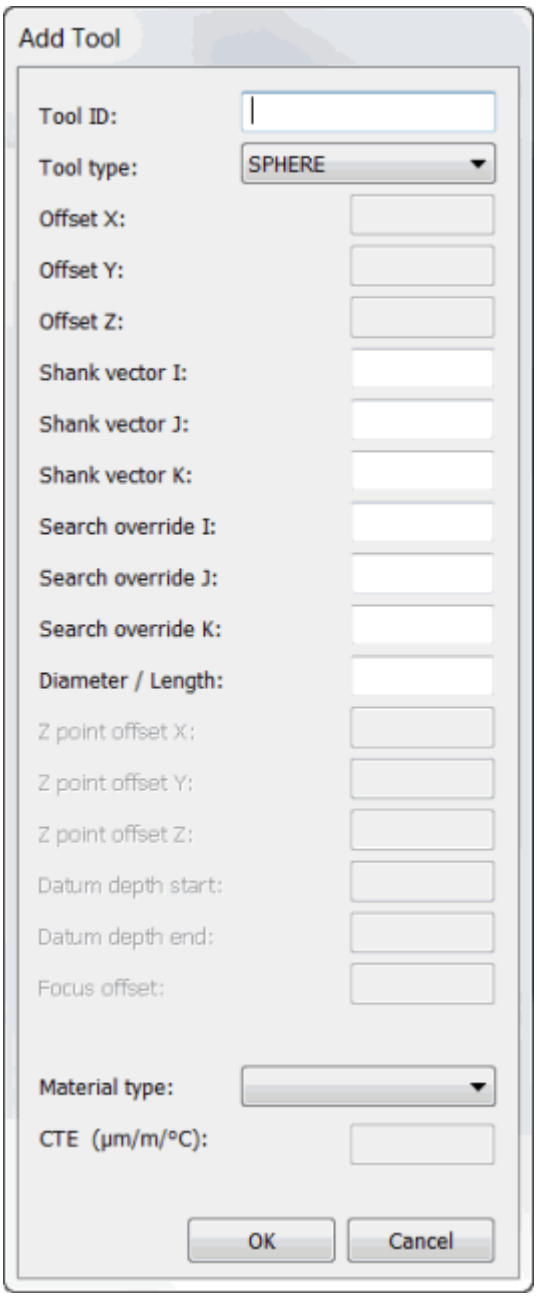
- **Ball Tip** - PC-DMIS prompts you to enter the number of hits to be taken on the tool and then asks you to take the hits.

- **Disk Tip** - If Mode = MANUAL, PC-DMIS prompts you to take six hits on the calibration tool. Take three hits above the equator of the sphere and three below the equator. If Mode = DCC, the location of the calibration tool must previously be known.
- **Taper Tip** - PC-DMIS prompt you to take six hits on the sphere with the tapered part of the probe. The spherical tool is, in effect, being used as the probe to measure the taper tip as a cone. The first three hits need to form a plane that is approximately perpendicular to the cone centerline. If a small taper probe is being used, it is best to use a small spherical tool to perform this operation.
- **Barrel/Shank Tip** - PC-DMIS prompts you to take four hits in a planar cross section of the sphere to form a circle. The hits must be made with the same part of the probe that will be used for inspection. After you take the four hits, PC-DMIS prompts you to touch the sphere once with the bottom surface of the probe.
- **Optical Tip** - This option is only available if the optical tip is defined as a hard probe.

Add Tool

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Add Tool** button displays the **Add Tool** dialog box. To access this button, select the **Measure** button in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).



Add Tool Dialog Box

The **Add Tool** dialog box allows you to store data that describes the qualification tools. Each tool is assigned a sequential ID number.

Once you define a new tool, PC-DMIS inserts it in the **List of available tools** list in the **Measure Probe** dialog box.

You must define at least one probe tip before you can measure a calibration tool.

To add a tool to the **List of available tools** list, select the **Add Tool** button to open the **Add Tool** dialog box.

You can define these items:

Tool ID - In this box, type the name of the tool you are defining.

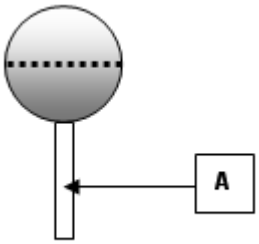
Tool type - Select the type of calibration tool. Some items are available for selection only if you use a specific probing system.

- SPHERE
- SPHERE(ARM 2)
- POLYHEDRAL
- POLYHEDRAL(ARM 2)
- RING
- RING(ARM 2)

  Ring tools are only used with vision probing systems. See the PC-DMIS Vision documentation for information on how to use this tool to calibrate.

Offset X, Y, Z - These X, Y, and Z values define the location of the calibration tool in machine coordinates. To recalibrate a tip, highlight the desired tip in the **Active tip list** option. Proceed to use the **Measure** button to calibrate the probe tip. (See "Previewing your Probe Configuration".)

Shank Vector I, J, K - These values define the vector of the shaft on the tool. PC-DMIS uses these values during calibration to avoid the shaft.



Example showing a shaft (A) on the spherical tool.

Search Override I, J, K - These boxes define a vector that PC-DMIS uses to determine the most efficient order to measure all the tips. This is enabled when you select the **User defined calibration order** check box in the **Probe Utilities** dialog box. For more information, see "User Defined Calibration Order check box".

Diameter / Length - This box displays the diameter or length of a spherical calibration tool.

The **Z point offset X**, **Z point offset Y**, **Z point offset Z**, **Datum depth start**, **Datum depth end**, and **Focus offset** options become enabled only if you are running PC-DMIS Vision. See the PC-DMIS Vision documentation for descriptions of those items.

Material type - This list contains standard material types. When you select a material, its coefficient of thermal expansion (CTE) appears in the **CTE** box.

Delete Tool

The **Delete Tool** command button allows you to remove a tool that is no longer needed in the **List of Available Tools** list.

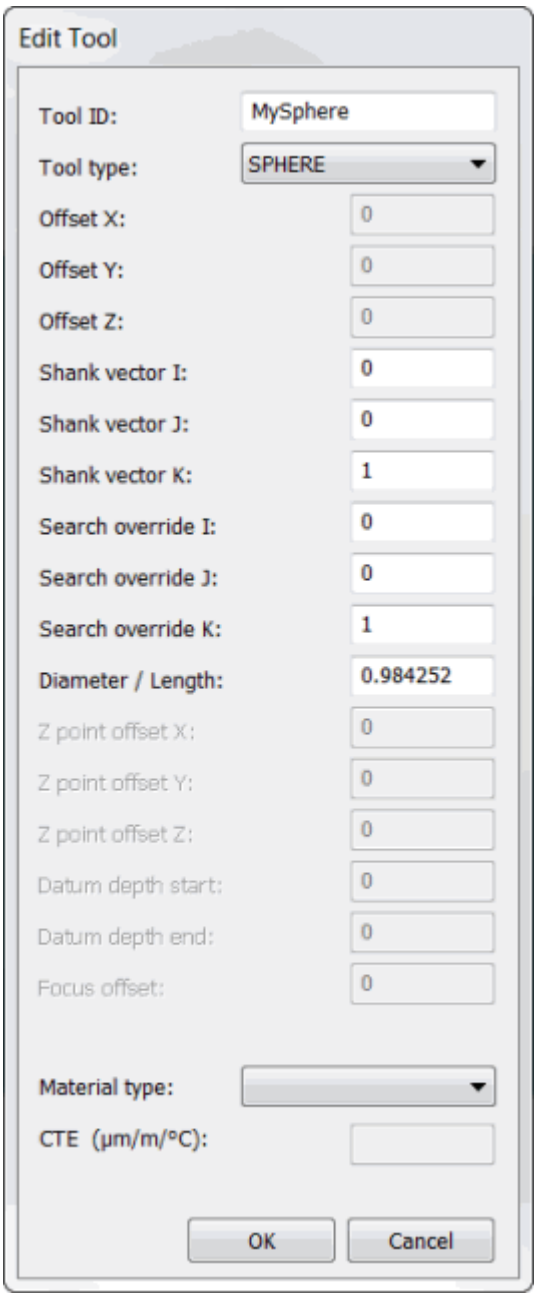
To remove a tool:

1. From the **List of Available Tools**, select the desired tool ID.
2. Click the **Delete Tool** button.

Edit Tool

You can use the **Edit Tool** command button to edit a calibration tool that already exists in the **List of available tools** list in the **Measure Probe** dialog box.

The **Edit Tool** button displays the **Edit Tool** dialog box. To access this button, select the **Measure** button in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).



Edit Tool Dialog Box

The **Edit Tool** dialog box has the same options as the **Add Tool** dialog box. For information about the options, see "Add Tool".

Notes on the Add Angle option

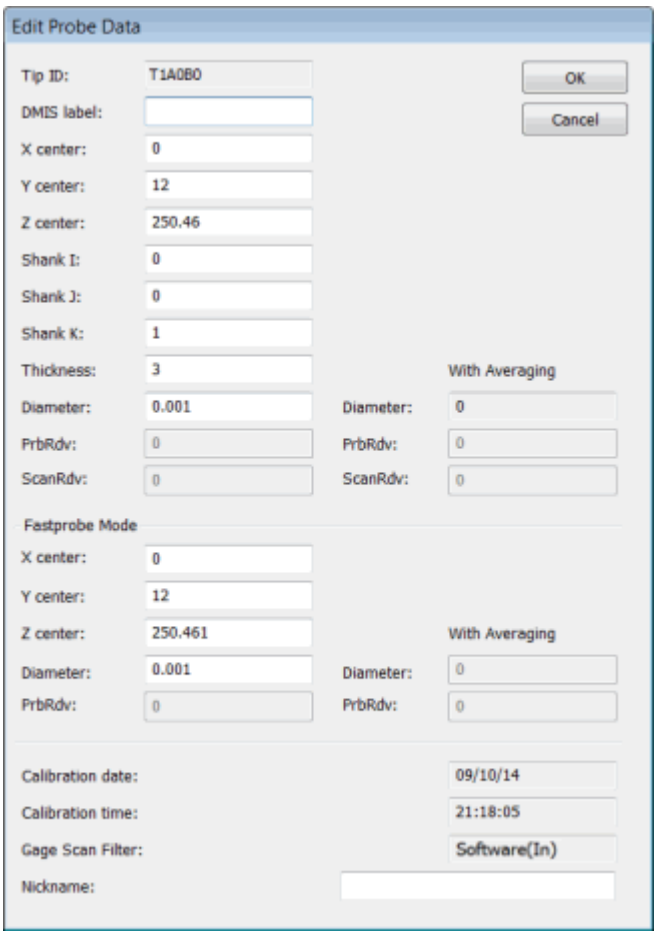
- Automatic probe calibration can only be done using a spherical probe and spherical qualification tool.
- The X, Y, and Z coordinate of the center of the tool that is being used must reflect the current position of the tool on the table. If this is not the case, answer YES to the question, "Has probe position moved?"

- When measuring AB positions under operator control, PC-DMIS automatically rotates the probe head to the next manual position when the current position is completed. Be sure the probe clears the tool before you press the END key.
- If the calibration tool position has changed or you need to add new angles to the list of tips, you can mark the needed angles in the **New Angles List** box. You must also mark the default position of the probe tip (normal to the shaft of the tool). PC-DMIS first executes the angle that is normal to the tool, and then executes the other marked angles.
- If the automatic AB calibration is stopped before it has finished measuring all the appended positions, PC-DMIS asks you if you want to keep the partial calibrated data before quitting.

Edit Probe Data

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

You can use the **Edit** button to modify the highlighted tip's details. Highlight the desired tip in the **Active Tip List** and click the **Edit** button in the **Probe Utilities** dialog box to open the **Edit Probe Data** dialog box.



Edit Probe Data Dialog Box

The following items are available in this dialog box:

TIP ID

This box contains the permanent number that PC-DMIS assigns to a tip when it is loaded into memory. You cannot edit this value. It is shown for display purposes only; however, you can define a more descriptive identification by using the Nickname box.

DMIS Label

This box displays the DMIS label. When importing DMIS files, PC-DMIS uses this value to identify any SNSDEF statement inside the imported DMIS file.

X, Y, and Z Center

These values describe the location of the tip. This location is in relation to the bottom of the Z rail.

Shank / Optical Vector I, J, K

These values describe the direction of the probe tip shank if you're using a probe tip. This vector goes from the center of the probe tip towards the Z rail. If you're using an optical probe, these values describe the direction of the optical device.

Diameter of the ball tip

This box contains the diameter of the tip.

The diameter for probes on non-portable machines may vary with the current part temperature if you mark the **Temperature compensation enabled** check box inside of the **Temperature Compensation** dialog box and choose a compensation method where PC-DMIS does the part compensation rather than the machine controller. The **Temperature Compensation** dialog box is accessible by pressing F9 on a TEMPCOMP command. See "Temperature Compensation Enabled" under "Compensating for Temperature" in the "Setting Your Preferences" chapter.

Thickness of the ball tip

This box contains the thickness of the tip. It defines the usable and graphical height/ thickness of the ruby tip. PC-DMIS applies this value during the disk probe calibration procedure to move the probe north or south of the equator. In the case of a disk probe, for example, you might need to adjust this value down to make the probe qualify better.

PrbRdv

The **PrbRdv** box defines *a radial deviation for the calibrated size of the tip.*

When you execute the probe calibration, PC-DMIS does one of two things:

1. If you have a machine configuration where PrbRdv applies, the calibration process automatically sets the tip size to the theoretical value, calculates, and then saves a **PrbRdv** value.

2. If you have a machine where PrbRdv does not apply, the calibration automatically sets the **PrbRdv** value to zero, calculates, and then saves a tip size slightly different from the theoretical value.

The **Edit Probe Data** dialog box is merely used to edit the tip size and/or PrbRdv if needed for some reason after calibration. If you re-calibrate, the resulting values come from the calibration, not from whatever may have been manually entered into this dialog box prior to calibrating.

This box is only available for selection if you use an analog probe on certain machines.

ScanRdv

Some machines that support **PrbRdv** allow for a different **ScanRdv** value. The **ScanRdv** box on the **Edit Probe Data** dialog box has the same meaning as PrbRdv in that it is a radial deviation for the calibrated size of the tip. The difference is that this applies to scan operations, whereas **PrbRdv** applies to taking individual hits.

With Averaging

If you're using tip averaging, additional Diameter, PrbRdv, ScanRdv values appear to show what the values would be if size averaging is utilized. For information on tip averaging, see the "Probe Setup" topic.

Because Fast Probe Mode is something that only applies to individual hits and not scans, ScanRdv does not exist for Fast Probe Mode.

Fast Probe Mode

The **Fast Probe Mode** area contains several read-only fields that show what the same tip values would be if Fast Probe Mode is used (or Fast Probe Mode with Tip Size Averaging).

- X center
- Y center
- Z center
- Diameter
- PrbRdv
- With Averaging - Diameter and PrbRdv

For information on Fast Probe Mode, see "Using Fast Probe Mode" under "Motion Commands" in the "Using the Edit Window" chapter.

For information on tip averaging, see the "Probe Setup" topic in this chapter.

Calibration Date and Time

These values indicate the most recent date and time the probe tip was calibrated. These values are available for viewing in the **Edit Probe Data** dialog box by clicking the **Edit** button on the **Probe Utilities** dialog box.

If a new tip is created without being calibrated, PC-DMIS will display NEW for the date and time values. If an old probe tip is loaded and the date and time information is unavailable, PC-DMIS will display UNKNOWN for the values. Only probe tips that are actually calibrated have their time and data values updated.

Gage Scan Filter

The **Gage Scan Filter** box in the Edit Probe Data dialog box displays the type of gage scan filter compensation for the selected probe tip: None, Software, or Hardware.

It also indicates how the gage scan filter was calibrated:

- (In) - with an inside circle
- (Out) - with an outside circle
- (Both) - with both an inside circle and an outside circle

For more information about the gage scan filter, see the "Enabling the Gage Scan Filter" topic in the "PC-DMIS CMM" documentation.

Nickname

The **Nickname** box on the Edit Probe Data dialog box is used to provide a more descriptive name to the selected probe Tip ID.

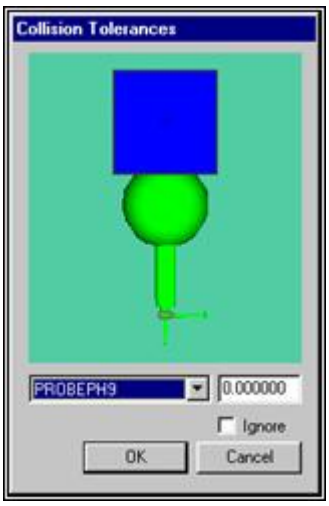
For example, if you named your probe tip, "MyTip" in the **Nickname** box, PC-DMIS uses "MyTip" in the user interface for that probe tip in dialog boxes, messages, on reports, etc. If you don't define an ID, PC-DMIS uses the default generated Tip ID instead.

Collision Tolerance

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

Collision Detection (CD) in PC-DMIS is designed to detect collisions between the probe and CAD surfaces.

Collision tolerances are specified in the **Collision Tolerances** dialog box. Click on the **Tolerances** button in the **Probe Utilities** dialog box to access this option.



Collision Tolerances Dialog Box

For additional information on Collision Detection, see "Detecting Collisions" in the "Editing the CAD Display" chapter.

Specifying Collision Tolerances

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

Collision tolerances are specified in the **Collision Tolerance** dialog box. This dialog box can be activated by clicking the **Tolerances** button in the **Probe Utilities** dialog box.

In the edit box, you can specify a positive or negative value for each probe component selected from the drop-down list. This effectively changes the size of that component.

- A positive number will increase the size of the component, so a collision is detected if that component comes within the specified distance of the part.
- A negative distance decreases the size of that portion of the probe, having the opposite effect of a positive distance.

You can also choose to ignore a probe component by selecting the **Ignore** check box. PC-DMIS ignores that probe component when checking for collisions. This may be useful with a tip, where collisions are expected to occur when taking hits.

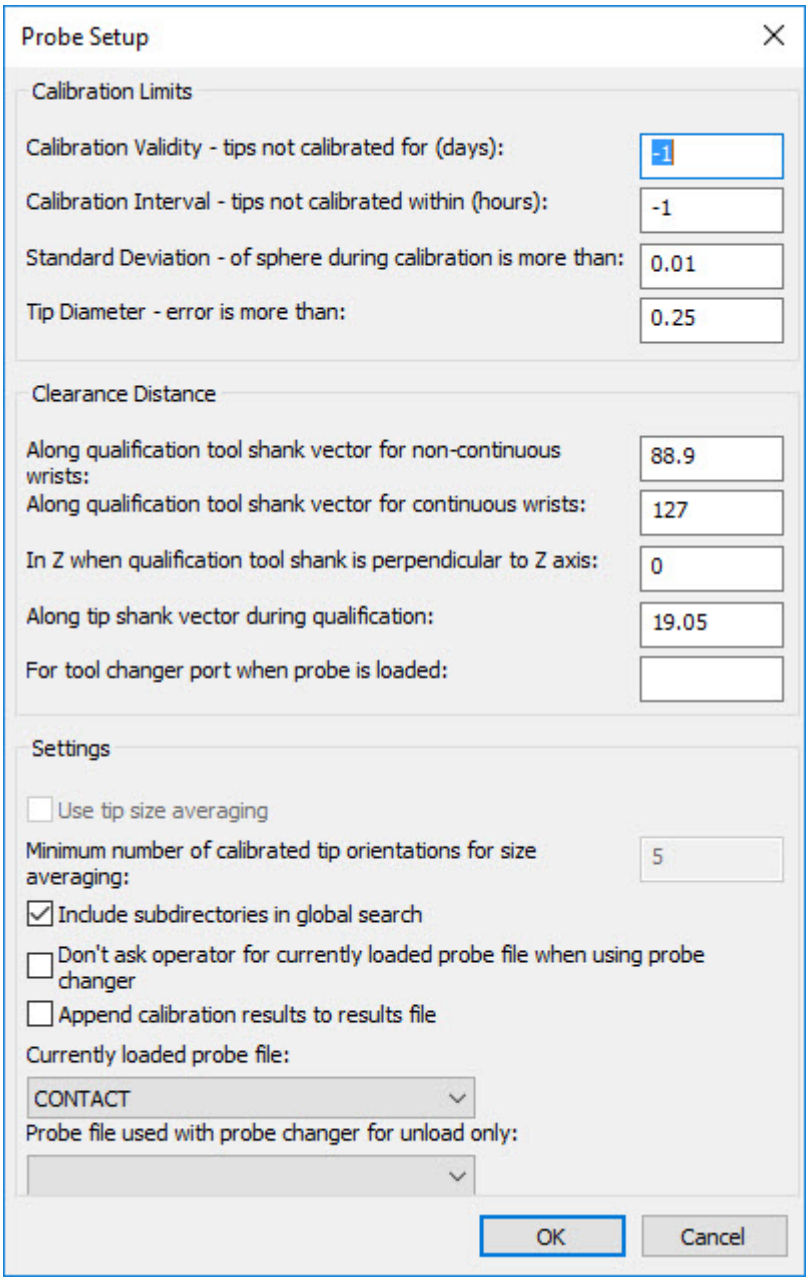
Once you start collision detection by selecting the **Collision Detection** menu option, PC-DMIS's internal CAD engine performs all the calculations required to detect a collision. Any collisions detected are displayed in the **Collision Detection** dialog box. This dialog box reports the results in the Probe Display window of the dialog box and saves them for later use in drawing the Edit Path lines.

For additional documentation regarding the Collision Detection option, see "Detecting Collisions" in the "Editing the CAD Display" chapter.

Probe Setup

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Setup** button in the **Probe Utilities** dialog box displays the **Probe Setup** dialog box:



Probe Setup Dialog Box

This dialog box further customizes your probe settings. PC-DMIS uses most of these settings globally for all probe files. Note, however, that the **Use tip size averaging** check box applies only to the current probe file.

You can use the options in this dialog box to change or select the following information:

Calibration Limits

The Check Calibration Limits command checks the calibration limits for all the tips that are used in the active measurement routine. The command also checks the limits when you open the measurement routine or edit the probe file. PC-DMIS warns you if any value goes above a limit.

For more information on this command, see "About Check Calibration Limits" in the PC-DMIS Toolkit Modules documentation.

You can set four types of calibration limits:

Calibration Validity - tips not calibrated for (days): - This item sets the limit based on the number of days that you can use a tip after calibration. If you don't want to receive any warning, set this item to a negative number. The default value is -1.

Calibration Interval - tips not calibrated within (hours): - This item sets the limit based on the time interval in hours within which all the tips in the active measurement routine should be calibrated. If you don't want to receive any warning, set this item to a negative number. The default value is -1.

Standard Deviation - of sphere during calibration is more than: - This item sets the limit to the standard deviation to accept tip calibration. During tip calibration, PC-DMIS calculates the standard deviation for the sphere measurement. The measurement units are the same as those used by your measurement routine.

Tip Diameter - error is more than: This item sets the limit to the deviation of the tip diameter from its nominal value. During tip calibration, PC-DMIS calculates the tip diameter for the measurement. The measurement units are the same as those used by your measurement routine.

Clearance Distance

Along qualification tool shank vector for non-continuous wrists: - This item is the distance the machine moves away from the qualification tool in the direction of the shank vector. During calibration, this item is useful to avoid collisions with the tool when PC-DMIS changes to a new AB tip angle. The measurement units are the same as those used by your measurement routine.

Along qualification tool shank vector for continuous wrists: - This item is the distance the machine moves away from the qualification tool in the direction of the shank vector. This is the same as the above setting except it is used with continuous wrists. Since continuous wrists are somewhat larger, they need more clearance. The measurement units are the same as those used by your measurement routine.

In Z when qualification tool shank is perpendicular to Z axis: - This item defines the clearance distance in the Z axis when the qualification tool's shank is perpendicular to the Z axis. When the qualification tool is mounted on a horizontal shank, it sometimes needs extra Z clearance movement, especially on horizontal

arm machines, to avoid collisions. The measurement units are the same as those used by your measurement routine.

Along tip shank vector during qualification: - This item defines a clearance distance that is added to the retract move during qualification that is based on the tool diameter. If you do not want to retract by even that tool diameter base value, you can set this value to a negative number to offset part of this retract distance.

This item sets the value for the `AutoQualClearanceTipDirection` entry in the **User_Option** section of the PC-DMIS Settings Editor. The default value is 0.75 inches (19.05 mm). The measurement units are the same as those used by your measurement routine.

For tool changer port when probe is loaded: - This item defines a probe-specific port clearance distance that overrides the port clearance distances defined on the probe changer. If you use this item, you won't have to continually update port clearances in your probe changer setup when you switch between measurement routines with different probe configurations for a given port.

See "Using Probe-Specific Port Clearance Distances" in the "Defining Hardware" chapter.

Settings

Use tip size averaging - This check box determines whether tip-size averaging is required.

Minimum number of calibrated tip orientations for size averaging: - This item defines the minimum number of validly calibrated tip orientations that must be present for averaging to take place. The default value is 5 and can never go below 2. This item sets the value for the `MinimumTipOrientationsForSizeAveraging` entry in the **USER_Probecal** section of the PC-DMIS Settings Editor.

The **Use tip size averaging** option averages the calibrated tip size for all AB tip angles used for a physical tip.

Normally, when a physical tip is calibrated at a specific AB tip angle, the calibration results include the measured tip size when calibrated at that angle. These stored tip sizes may vary slightly even though they all use the same physical tip.

In some very specific cases, select this check box to improve the overall measurement accuracy.

This option is not intended for general use. Generally, you should only select this check box if recommended to do so by the machine's manufacturer for your specific configuration. Misuse may cause loss of accuracy.

If you select this check box, the following occurs:

- After PC-DMIS completes a given calibration, it averages the tip size for all *calibrated* AB tip angles for that same physical tip, regardless of whether you included them in the same calibration or not.
- PC-DMIS excludes *non-calibrated* AB tip angles which use that same tip from the averaging calculation.
- PC-DMIS excludes any AB tip angles in the current calibration that exceed the warning limits for standard deviation or for size deviation from the nominal from the averaging calculation. However, their sizes are still set to the calculated average value for that physical tip.

By default, you must have at least five validly calibrated tip locations before size averaging takes place. You can use the `MinimumTipOrientationsForSizeAveraging` entry in the **USER_Probecal** section of the PC-DMIS Settings Editor to specify a different minimum number, but you must have at least two.

Include subdirectories in global search - If you select this check box, the **Global Used** search includes subdirectories. See "Global Used".

Don't ask operator for currently loaded probe file when using probe changer - When you first start PC-DMIS, it doesn’t know what may have taken place on the machine while PC-DMIS wasn’t running. By default, it prompts you to confirm or select the currently-loaded probe before you attempt the first change after startup. If you select this check box, PC-DMIS assumes that the last-known loaded probe is still correct, and it doesn't prompt you for confirmation.

Append calibration results to results file - This item determines whether existing calibration results report data is overwritten or appended during the next calibration.

Currently loaded probe file: - If you're using a probe changer, PC-DMIS needs to know what (if anything) needs to be dropped off first. To provide that information, this list displays what is currently loaded on your machine and allows you to manually specify it.

Probe file used with probe changer for unload only: - This list lets you choose a dummy probe file used to force an unload of the current probe without loading a new probe from the probe changer. See "To Drop off a Probe without Picking Up a New Probe" in the "Defining Hardware" chapter.

Print List

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Print List** command button is found on the **Probe Utilities** dialog box and displays the **Print** dialog box.

When you click the **OK** button on this dialog box, PC-DMIS prints a list of calibrated and non-calibrated tip angles in a table. These are the same tip angles listed in the Active Tip List. Each row of the table contains a tip angle while the columns display the XYZ, IJK, diameter, and thickness values for each tip angle. It also displays the date and time of the calibrated tip angles.

If a tip angle hasn't been calibrated or if there's a Diameter, Date, or Time error, the list shows the tip in red text.

Example Printout

Below, you can see example printouts of the active tip list for a contact probe and a laser probe:

Probe File = D:\ProbeFiles\PH9_V41.PRB

ID	Type	X	Y	Z	I	J	K	Diam	Thick	Date	Time
T1A15B-60	BALL	25.328	14.623	186.15	0.224	0.129	0.966	4	4	NEW	NEW
T1A22.5B-105	BALL	41.77	-11.192	181.398	0.37	-0.099	0.924	4	4	NEW	NEW
T1A22.5B120	BALL	-37.45	-21.622	181.398	-0.331	-0.191	0.924	4	4	NEW	NEW
T1A22.5B37.5	BALL	-26.325	34.307	181.398	-0.233	0.304	0.924	4	4	NEW	NEW
T1A30B120	BALL	-48.93	-28.25	174.861	-0.433	-0.25	0.866	4	4	NEW	NEW
T1A30B-22.5	BALL	21.622	52.199	174.861	0.191	0.462	0.866	4	4	NEW	NEW
T1A30B-60	BALL	48.93	28.25	174.861	0.433	0.25	0.866	4	4	NEW	NEW
T1A30B82.5	BALL	-56.017	7.375	174.861	-0.496	0.065	0.866	4	4	NEW	NEW
T1A30B-97.5	BALL	56.017	-7.375	174.861	0.496	-0.065	0.866	4	4	NEW	NEW
T1A37.5B105	BALL	-66.446	-17.804	166.649	-0.588	-0.158	0.793	4	4	NEW	NEW
T1A37.5B-37.5	BALL	41.877	54.575	166.649	0.371	0.483	0.793	4	4	NEW	NEW
T1A37.5B45	BALL	-48.642	48.642	166.649	-0.43	0.43	0.793	4	4	NEW	NEW
T1A37.5B-67.5	BALL	63.554	26.325	166.649	0.562	0.233	0.793	4	4	NEW	NEW
T1A45B-112.5	BALL	73.821	-30.578	156.903	0.653	-0.271	0.707	4	4	NEW	NEW
T1A45B15	BALL	-20.68	77.18	156.903	-0.183	0.683	0.707	4	4	NEW	NEW
T1A52.5B120	BALL	-77.638	-44.824	145.79	-0.687	-0.397	0.609	4	4	NEW	NEW
T1A52.5B-22.5	BALL	34.307	82.825	145.79	0.304	0.733	0.609	4	4	NEW	NEW
T1A52.5B52.5	BALL	-71.123	54.575	145.79	-0.629	0.483	0.609	4	4	NEW	NEW
T1A52.5B-60	BALL	77.638	44.824	145.79	0.687	0.397	0.609	4	4	NEW	NEW
T1A60B7.5	BALL	-12.773	97.024	133.5	-0.113	0.859	0.5	4	4	NEW	NEW
T1A60B-75	BALL	94.526	25.328	133.5	0.837	0.224	0.5	4	4	NEW	NEW
T1A60B-97.5	BALL	97.024	-12.773	133.5	0.859	-0.113	0.5	4	4	NEW	NEW
T1A67.5B67.5	BALL	-96.452	39.952	120.243	-0.854	0.354	0.383	4	4	NEW	NEW
T1A7.5B22.5	BALL	-5.644	13.627	189.033	-0.05	0.121	0.991	4	4	NEW	NEW
T1A0B0	BALL	0	0	190	0	0	1	4	4	11/22/06	16:56:39
T1A7.5B-37.5	BALL	8.979	11.702	189.033	0.079	0.104	0.991	4	4	11/22/06	16:56:59
T1A7.5B60	BALL	-12.773	7.375	189.033	-0.113	0.065	0.991	4	4	11/22/06	16:56:59
T1A7.5B-7.5	BALL	1.925	14.623	189.033	0.017	0.129	0.991	4	4	11/22/06	16:56:59
T1A7.5B82.5	BALL	-14.623	1.925	189.033	-0.129	0.017	0.991	4	4	11/22/06	16:56:59
T1A7.5B-90	BALL	14.749	0	189.033	0.131	0	0.991	4	4	11/22/06	16:56:59

Example Printout of Active Tips for a Contact probe

Probe File = D:\PROBE LIST\NEW208.PRB

ID	Type	X	Y	Z	I	J	K	Avg Err	Std Dev	Date	Time
T1A0B0	LASER	112.476	-2.01	407.666	0	0	1	0	0	07/30/19	09:02:38
T1A0B180	LASER	-112.968	1.989	407.476	0	0	1	0	0	07/30/19	09:02:38
T1A0B-180	LASER	-112.97	1.989	407.471	0	0	1	0	0	07/30/19	09:02:38
T1A0B90	LASER	1.745	112.717	407.594	0	0	1	0	0	07/30/19	09:02:38
T1A0B-90	LASER	-2.237	-112.738	407.541	0	0	1	0	0	07/30/19	09:02:38
T1A15B0	LASER	112.513	85.918	396.636	0	0.259	0.966	0	0	07/30/19	09:02:38
T1A15B-180	LASER	-112.986	-85.932	396.404	0	-0.259	0.966	0	0	07/30/19	09:02:38
T1A15B180	LASER	-112.986	-85.937	396.41	0	-0.259	0.966	0	0	07/30/19	09:02:38
T1A15B-90	LASER	85.698	-112.763	396.567	0.259	0	0.966	0	0	07/30/19	09:02:38
T1A15B90	LASER	-86.174	112.748	396.47	-0.259	0	0.966	0	0	07/30/19	09:02:38

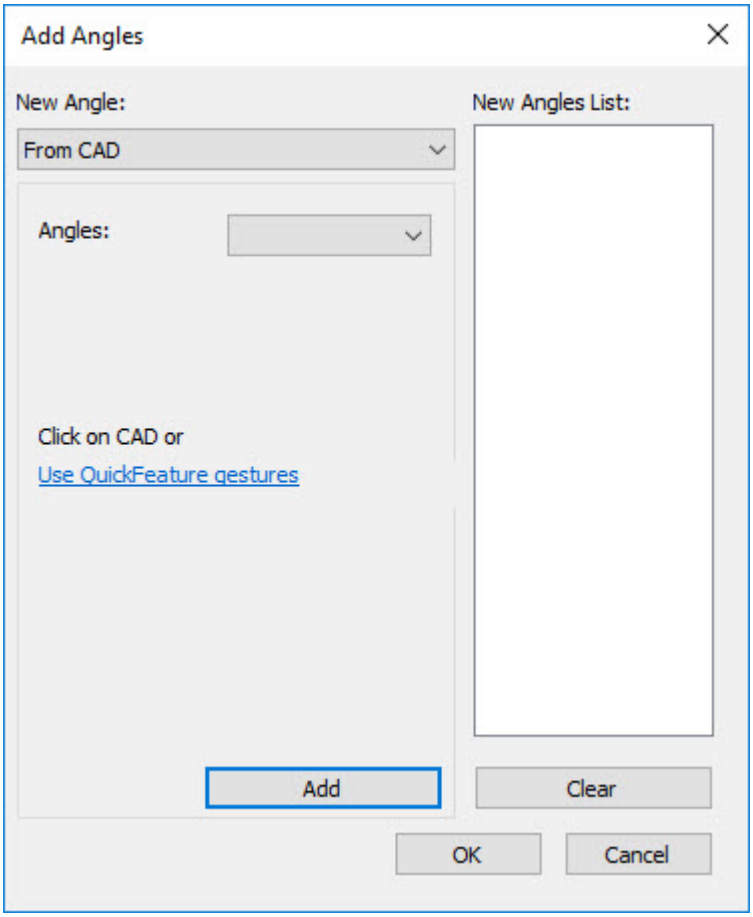
Example Printout of Active Tips for a laser probe

Add Angles

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Add Angles** command button is on the **Probe Utilities** dialog box. This button is available only if the probe head has the capability to rotate in your probe configuration.

Click the **Add Angles** button to open the **Add Angles** dialog box.



Add Angles Dialog Box

The **Add Angles** dialog box has the following parameters:

New Angle - The list displays four options to add angles:

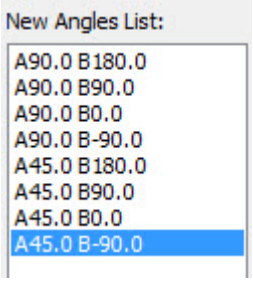
- **Single** - This option lets you add AB positions one at a time into the **New Angles List** box.
- **Multiple** - This option lets you add equally-spaced angles.
- **From CAD** - This option is available only if CAD is available in the active measurement routine. This option lets you select the CAD surface and adds recommended angle into the **New Angles List** box.

- **From Grid** - This option lets you view and select angles with the use of an angle grid.

Add - This button adds the angle into the **New Angles List** box.

Clear - This button removes the list of angles from the **New Angles List** box.

New Angles List Box



New Angles List Box

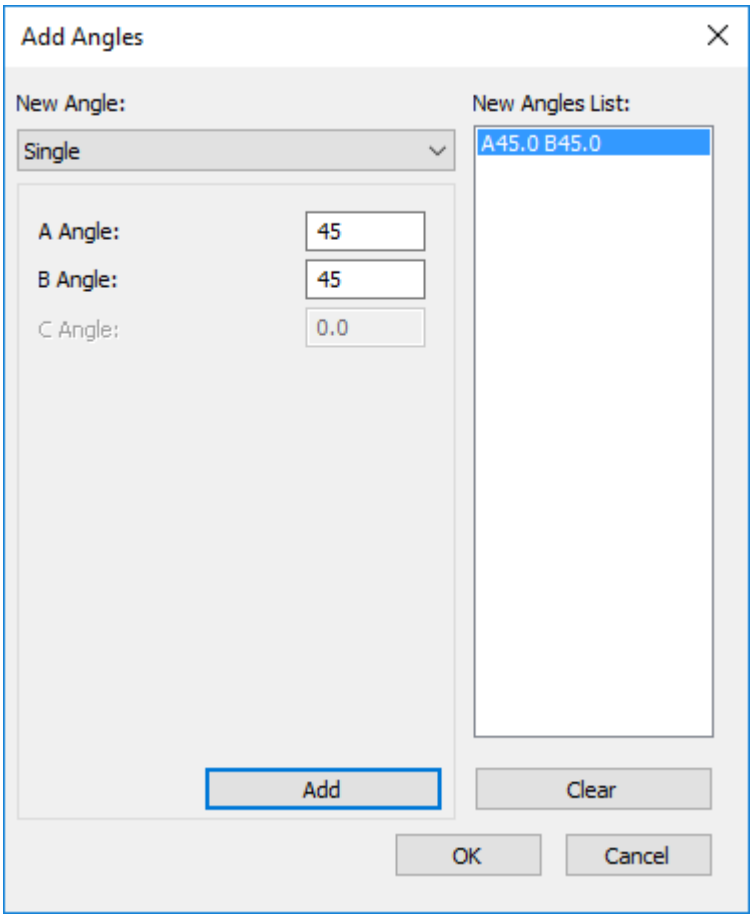
The **New Angles List** box contains the list of AB angles that you added from the options available in the **New Angle** list.

Select an angle to see the angle's position on the CAD in the **Probe Utilities** dialog box.

To remove an angle from the **New Angles List** box, right-click and select **Remove**. To remove all the angles from the **New Angles List** box, click the **Clear** button.

Add Single Angle

The **Single** option lets you add AB positions one at a time into the **New Angles List** box.



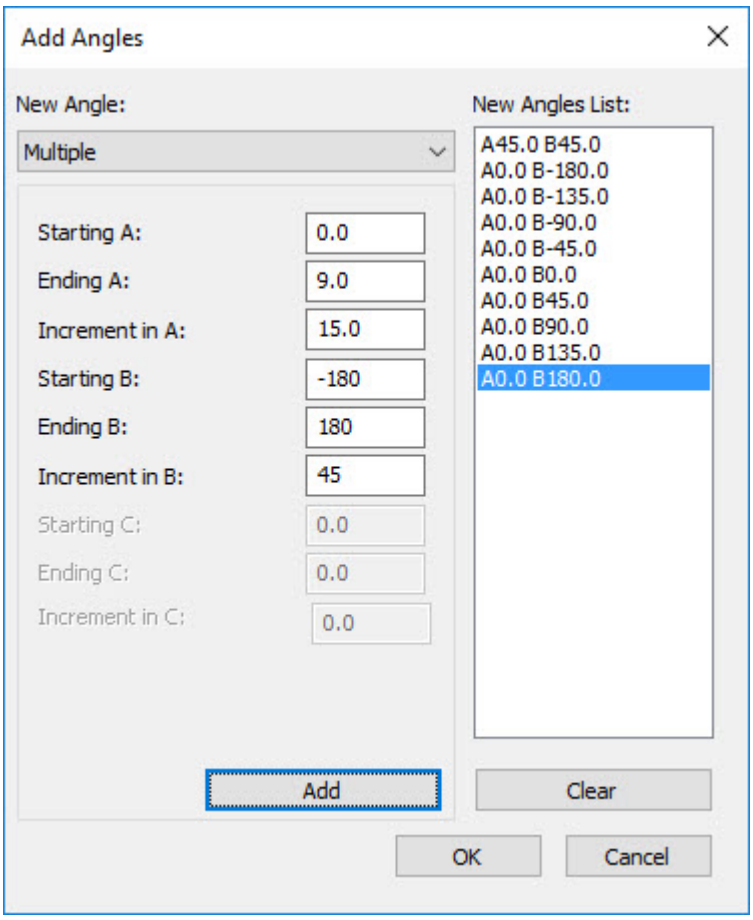
Add Angles Dialog Box - Single Option

In the **A Angle**, **B Angle**, and **C Angle** boxes, type angles and click the **Add** button.

The individual angle you specified appears in the **New Angles List** box.

Add Multiple Angles

The **Multiple** option lets you add equally-spaced angles into the **New Angles List** box.



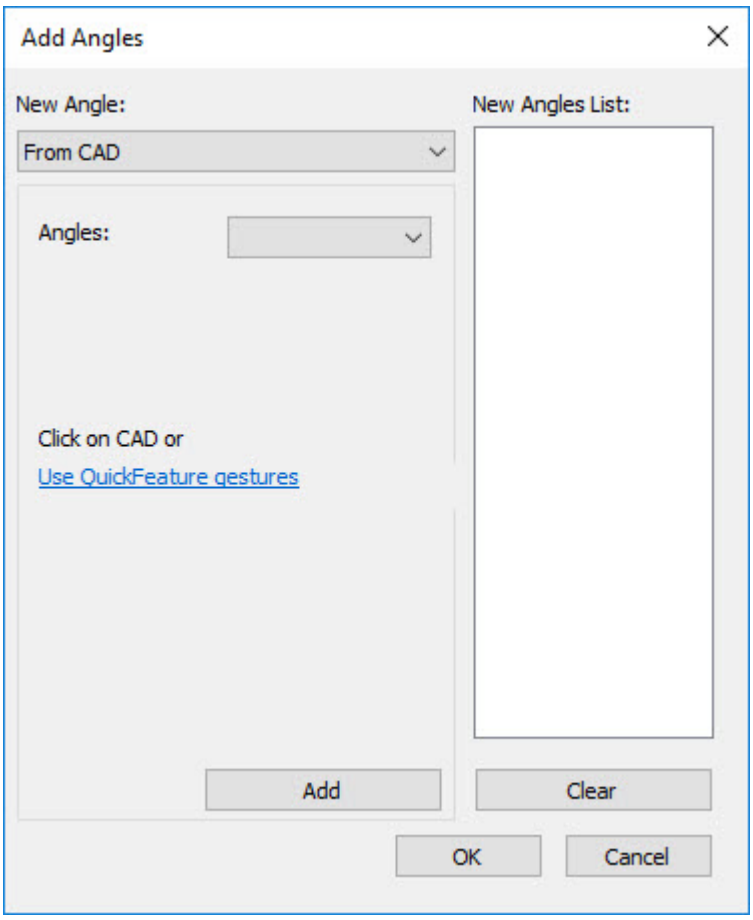
Add Angles Dialog Box - Multiple Option

To add the new angles, perform the following steps:

1. Enter the necessary information in each available angle box (**Starting A**, **Ending A**, **Increment in A**, **Starting B**, **Ending B**, and **Increment in B**).
2. Click the **Add** button. PC-DMIS displays the equally-spaced angles in the **New Angles List** box.

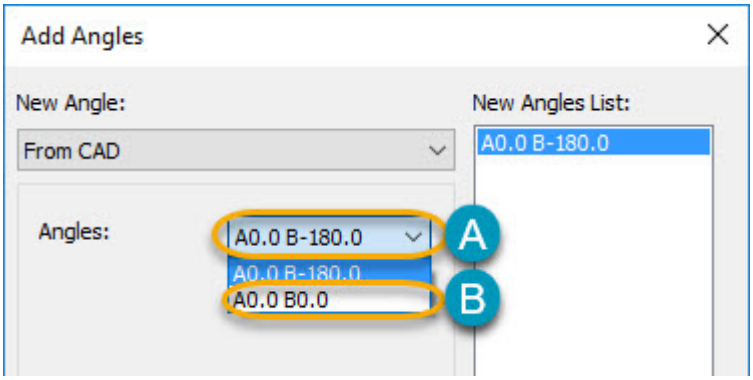
Add Angles from CAD

The **From CAD** option is available only if CAD is available in the active measurement routine. This option lets you select the CAD surface and add the recommended angle into the **New Angles List** box.



Add Angles Dialog Box - From CAD Option

If you select this option, you can either click on the CAD surface or use the QuickFeature gesture. PC-DMIS calculates the recommended and alternate angles wherever possible. Both angles are available to select from the **New Angles** list.

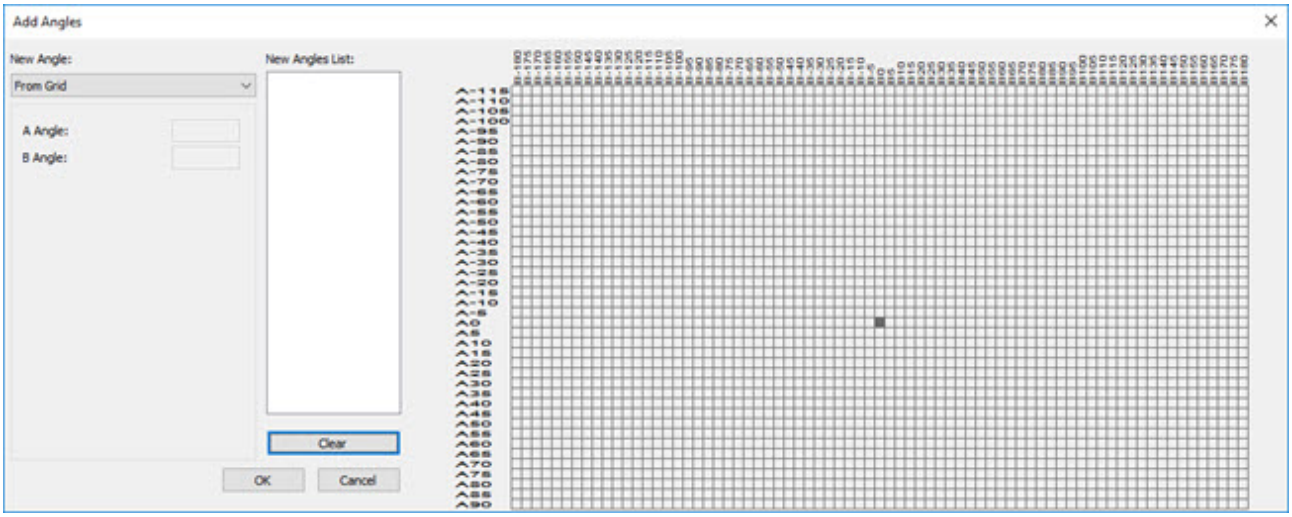


A. For the recommended angle, click on the CAD surface or use the QuickFeature gesture. PC-DMIS automatically adds the angle into the **New Angles List** box.

B. For the alternate angle, from the **Angles** list box, select the alternate angle and click the **Add** button to add the angle into the **New Angles List** box.

Add Angles from Grid

The **From Grid** option lets you view and add angles from the angle grid. The angle grid displays all of the available A-angle positions down the side of the dialog box. It displays the B-angle positions along the top of the dialog box.



Add Angles Dialog Box - From Grid Option

To add a new angle from the grid, perform the following steps:

1. Position your mouse pointer over the grid. The **A Angle** and **B Angle** boxes display the angle dynamically as you move your mouse pointer on the grid to determine the angle.

2. Find the row of the A angle you want to add. Then find the column with the appropriate B angle.

3. Click the box in the grid where the desired A and B angles intersect. The selected box turns red, and PC-DMIS inserts the selected AB angle positions into the **New Angles List** box.

To clear a selected AB angle position, click one of the red boxes again. PC-DMIS then removes the AB angle positions associated with that box from the **New Angles List** box.

Supported QuickFeature Gestures to Add Angle

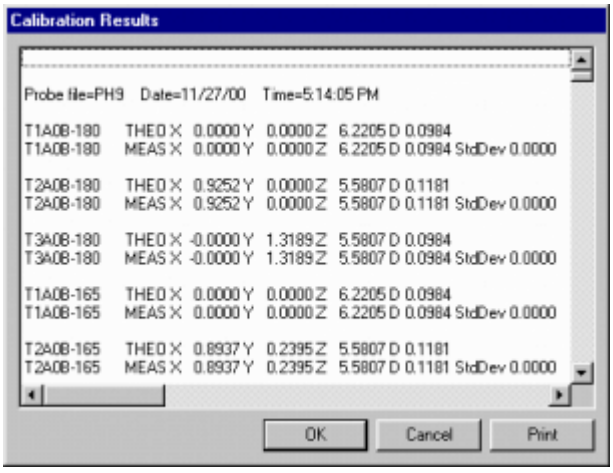
PC-DMIS supports the list of QuickFeature gestures in the following table:

QuickFeature	Gesture
Vector Point, Edge Point, Angle Point	Press Ctrl + Shift and click on the CAD element.
Plane, Circle, Ellipse, Cylinder, Sphere	Press Ctrl + Shift and click on the CAD element.
Square Slot	Press and hold Shift, (1) hover the pointer over one edge of the slot, click and drag the pointer for a short distance along the edge to highlight it; then (2) move the pointer to an adjacent edge. Once the entire slot is highlighted, (3) release the mouse button to create the feature.
Round Slot	Press and hold Shift, (1) hover the pointer over one of the circular ends of the slot, and click and drag the pointer a short distance along the curve to highlight it; then (2) move the pointer to a straight side. Once the entire slot is highlighted, (3) release the mouse button to create the feature.

Results

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Results** command button is found on the **Probe Utilities** dialog box. It is used to display the results of the most recent probe calibration in the **Calibration Results** dialog box.



Results Dialog Box

In addition to displaying the probe's diameter and thickness, the dialog box also provides the standard deviation of the sphere. These measured results are used to verify the accuracy of the calibration.

PC-DMIS shows results up to six decimal places.

Mark Used

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Mark Used** command button is found on the **Probe Utilities** dialog box.

This option automatically searches the current measurement routine for AB angles. PC-DMIS then adds any AB angles that are found to the current probe configuration.

If you select the **Automatically Adjust Probe Head Wrist** check box (located on the **General** tab in the **Setup Options** dialog box) and click the **Mark Used** button, PC-DMIS may not select all the probe tips for calibration. For more information about the **Automatically Adjust Probe Head Wrist** check box, see the "Automatically Adjust Probe Head Wrist" topic in the "Setting your Preferences" chapter.

Global Used

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Global Used** command button is in the **Probe Utilities** dialog box.

The **Global Used** button searches for tips used in other measurement routines by the currently-active probe file. It then adds them to the **Active tip list** and marks them for calibration.

By default, this button searches through any measurement routine subdirectories. You can control whether or not this button searches these subdirectories by using the **Include subdirectories in global used search** check box in the **Probe Setup** dialog box.

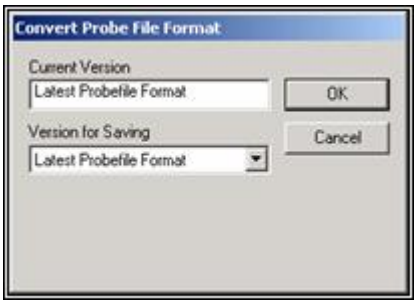
File Format

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

File Format

The **File Format** command button is found on the **Probe Utilities** dialog box.

The **File Format** button saves the existing probe file into a format compatible with previous versions of PC-DMIS. When you click the **File Format** button, the **Convert Probe File Format** dialog box appears.



Convert Probe File Format Dialog Box

The dialog box contains these two items:

- **Current Version** box - This lists the current version of PC-DMIS.
- **Version for Saving** list - This contains the probe file format types you can save the probe file as.

Reset Tips

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

Reset Tips

The **Reset Tips** button resets any selected tips in the Active Tip List so that they are no longer calibrated. This provides you with a quick way to reset tip data back to their theoretical values. Previously, you had to delete and then add the tip back in to accomplish this same thing.

If you do not have any tips selected when clicking this button, you will be prompted to confirm whether or not you want to reset all the tips. If you choose **Yes**, then all the tips will be reset, otherwise no action will be taken.

Use Partial Calibration check box

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

When using the Renishaw scan-based method to calibrate a Renishaw analog probe (such as the SP25, SP600, or SP80), the first time it's calibrated using the Scan method you must perform the full calibration that runs the entire series of scans. After this full calibration, you can choose a more simplified calibration.

- Full calibration calculates all the analog probing coefficients in addition to the tip offset and tip size.
- Partial (simplified) calibration works just like a non-analog probe calibration: it consists of discrete hits (no scans) and only calculates the tip offset and tip size; the analog probing coefficients remain unchanged.

To perform the simplified calibration, follow this procedure:

1. Select **Insert | Hardware Definition | Probe** to display the **Probe Utilities** dialog box.
2. Load a Renishaw analog probe from the **Probe File** list.
3. Select the **Use Partial Calibration** check box. This remains disabled for probes that do not apply.
4. Select one or more already calibrated probe tips from **Active Tips List**.
5. Click the **Measure** button. The **Measure** dialog box appears.
6. Make changes as needed on the **Measure** dialog box. If you define any named parameter sets, PC-DMIS stores the state of the **Use Partial Calibration** check box in the set for future use.
7. Click **Measure**. Follow any on-screen prompts. PC-DMIS performs the simplified calibration.

The `ProbeUsePartialCalibration` entry in the **ProbeCal** section in the PC-DMIS Settings Editor stores the default for whether or not this check box is selected when you define a new probe file.

Use Wrist Map If Available check box

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Use Wrist Map If Available** check box determines whether or not PC-DMIS uses a wrist map (also known as an "error map") file when you calibrate AB angle tips on probe configurations that use an indexable wrist. If you select this check box, PC-DMIS searches for the wrist map file (a file named *abcomp.dat*) on your computer. If it finds the file, PC-DMIS compensates for the wrist's error data when it calibrates the AB angle tips.

For information on creating the wrist map file, see the "Calculate Error Map" topic in the "Using a Wrist Device" chapter of the PC-DMIS Core documentation.

Use TRAX Calibration check box

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

When calibrating an analog probe on some machine types, particularly when using the Leitz interface, there is typically a choice of whether or not to use the TRAX calibration algorithm for calculating the calibration coefficients. You can choose which calibration algorithm is used by selecting or clearing the **Use TRAX Calibration** check box. Previously, this setting was controlled in the PC-DMIS Settings Editor by modifying the `DISABLETRAXCAL` option.

- If you select this check box, PC-DMIS uses the TRAX calibration algorithm. The TRAX algorithm is optimized for use with scanning, so you get the best scanning results using this method. It also usually produces good results for discrete point measurements as well.
- If you clear this check box, the algorithm used depends on whether or not your machine supports the VFT (Vibration Free Transmission) calibration methodology. If it is supported, VFT is used; otherwise, the PMM algorithm is used.

The selected algorithm only applies to the currently selected tips at the time of calibration. You can change this option later for different groupings of tips as needed. The type of algorithm you chose can also be stored and recalled using named parameter sets. See the "Parameter Sets area" topic in the PC-DMIS CMM documentation.

Note that this check box does not change the distribution of discrete hits that are collected. It defines the math algorithm used to process the data after the calibration finishes. In the VFT case, it also results in a set of scans being performed after the discrete hits so that the calibration automatically includes both discrete hit data and scan data.

When to select the TRAX check box or not

The normal expected usage would be as follows:

- Do NOT select the TRAX check box on machines that support VFT so that it uses the VFT specific calibration.
- DO select the TRAX check box on non-VFT machines so that it uses the TRAX algorithm instead of the PMM algorithm.

TRAX is still available for VFT capable machines as an alternative to VFT in case of unexpected issues but is expected to be less accurate on these machines than the VFT calibration.

PMM is still available as an alternative to TRAX on non-VFT machines but is rarely used. TRAX is superior to PMM for scanning and usually just as good or better for discrete hits. There may be a few circumstances, usually limited to actual PMM machines, where the PMM algorithm might be slightly more accurate than TRAX if the inspection is only using discrete hits.

If you're unsure about when to switch calibration algorithms, consult with the creators of your particular machine. They can recommend the best practice for that machine and for particular situations.

User Defined Calibration Order check box

(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **User Defined Calibration Order** check box is used to determine the order PC-DMIS takes to measure selected tips.

If you select this check box, PC-DMIS measures the tips in the order defined by the user when marking the tips for calibration in the **Active Tip List**. (See "Active Tip List") If no tips are chosen, PC-DMIS uses the I, J, K vectors defined in the **Search I, J, K** boxes in the **Edit Tool** or **Add Tool** dialog boxes to determine the most efficient order to measure all the tips.

If you do not select this check box, PC-DMIS measures the tips in an order that it determines to be the most efficient. When this happens, PC-DMIS ignores the order that was given to selected tips in the **Active Tip List**. PC-DMIS also uses the calibration tool vector to determine the most efficient order to measure.

Use Unit Calibration Data

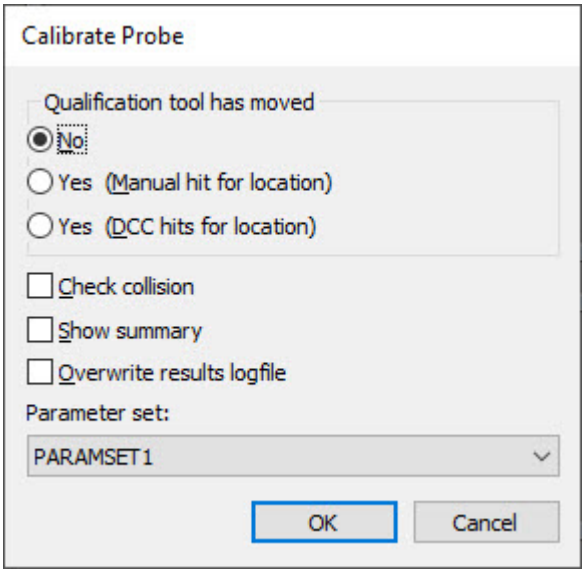
(This item pertains to the **Probe Utilities** dialog box, accessible through **Insert | Hardware Definition | Probe**.)

The **Use Unit Calib Data** check box only appears on the dialog box if you have already performed a unit calibration. If this check box is not selected, PC-DMIS uses the standard calibration. Selecting this check box allows you to use the unit calibration data. See "Calibrate the Unit" in the "Calibrating Probe Tips" topic for additional information.

AutoCalibrate Probe

The Autocalibrate command automatically calibrates the current probe during measurement routine execution. PC-DMIS begins the calibration routine once it executes the command.

To insert this command, select **Insert | Calibrate | AutoCalibrate Probe** from the menu to open the **Calibrate Probe** dialog box.



Calibrate Probe Dialog Box

An example for the Edit window code for this command reads:

```
AUTOCALIBRATE/PROBE, PARAMETER_SET=, QUALTOOL_MOVED=Y/Y/N,
CHECK COLLISION=Y/N, SHOW_SUMMARY=Y/N,
OVERWRITE_RESULTSFILE=Y/N
```

`PARAMETER_SET=` This field specifies the name of a defined set of probe qualification parameters. A default set named `ALL-TIPS-WITH-DEFAULTS` is always available and calibrates all tips defined for the current probe. It uses the default set of qualification parameters, which are the last ones that were used interactively. To create your own parameter sets, see "Parameter Sets".

`QUALTOOL_MOVED=` This field sets your response to the computer's query of whether or not the calibration tool has moved. It can be set to one of these three options:

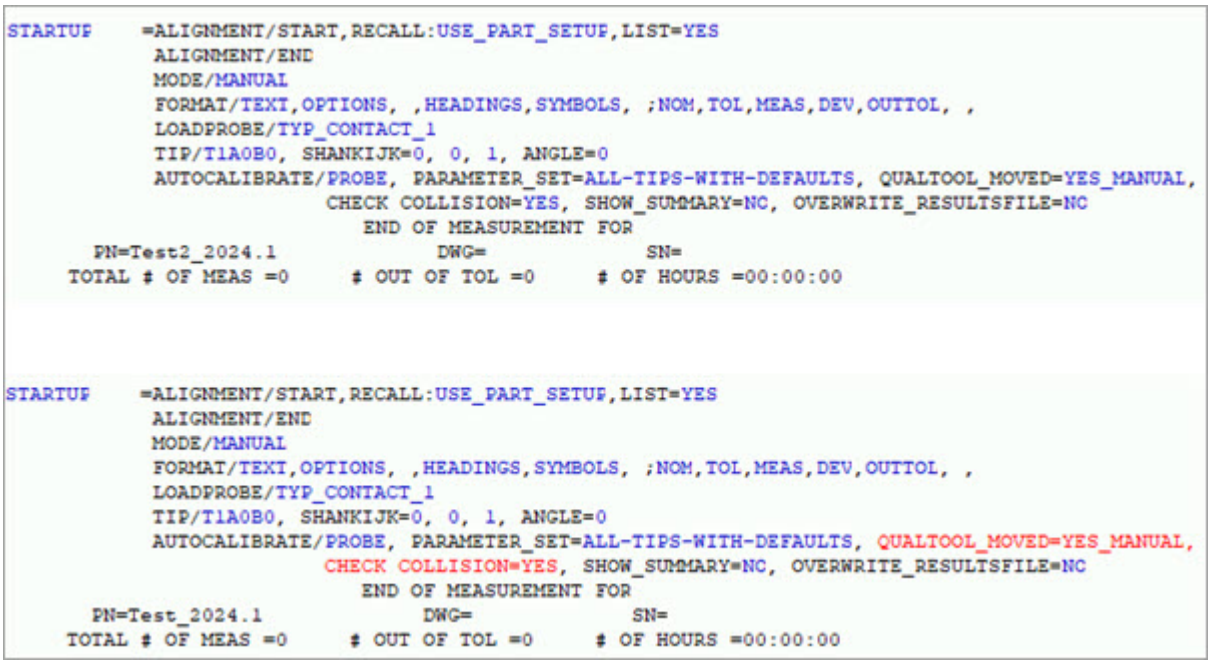
- NO - This means the calibration tool has not moved.
- YES_MANUAL - This means the calibration tool has moved, but PC-DMIS requires you to take a manual hit to locate the sphere.

- YES_DCC - This means the calibration tool has moved but PC-DMIS should use DCC hits to attempt to locate the calibration tool. You may use this option when you have repositioned the qualification tool to nearly the same previous location.

`CHECK COLLISION=` Starting with PC-DMIS version 2024.2, this field is only enabled when `QUALTOOL_MOVED=NO` . If `QUALTOOL_MOVED=NO` and `CHECK COLLISION=YES` , PC-DMIS enables the Leg Collision check on both Bridge and horizontal/dual arm machines. If you have a vertical machine, this option is only available on the vertical calibration tools. PC-DMIS does not support this functionality on star probes or laser sensors.

If you open a measurement routine earlier than version 2024.2 and it has the `QUALTOOL_MOVED=YES` and `CHECK COLLISION=YES` defined, the `AUTOCALIBRATE` command in the Edit window displays these fields in red.

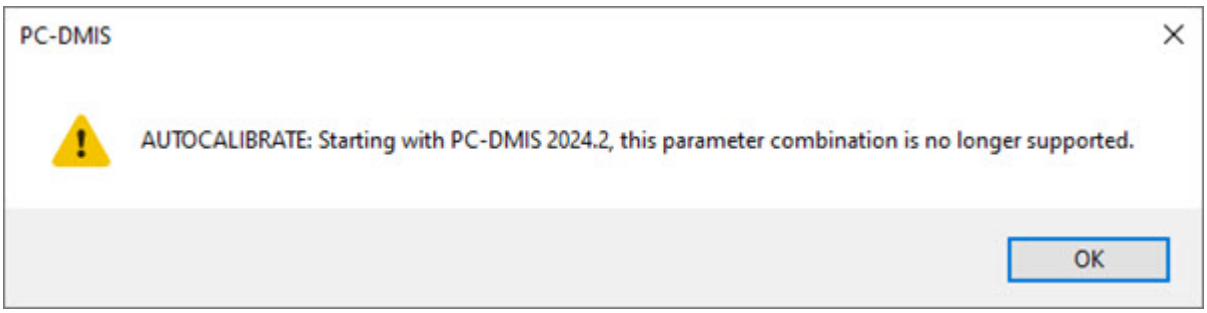
For example, the top code snippet in the image below shows a measurement routine created in PC-DMIS 2024.1 with `QUALTOOL_MOVED=YES_MANUAL` and `CHECK COLLISION=YES` . When you open this same measurement routine in PC-DMIS 2024.2, the software displays those fields in red in the Edit window as shown at the bottom of the image below.



Example of an AUTOCALIBRATE command in version 2024.1 (top) and version 2024.2 (bottom).

If you execute the PC-DMIS 2024.1 measurement routine in PC-DMIS 2024.2, it will perform the collision detection.

When you save the 2024.2 version of that measurement routine, PC-DMIS shows this message:



The next time you open the same measurement routine in PC-DMIS 2024.2, the software updates the AUTOCALIBRATE command as shown below:

```
STARTUP    =ALIGNMENT/START,RECALL:USE_PART_SETUP,LIST=YES
            ALIGNMENT/END
            MODE/MANUAL
            FORMAT/TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;NOM,TOL,MEAS,DEV,OUTTOL, ,
            LOADPROBE/TYP_CONTACT_1
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
            AUTOCALIBRATE/PROBE, PARAMETER_SET=ALL-TIPS-WITH-DEFAULTS, QUALTOOL_MOVED=YES_MANUAL,
                         CHECK COLLISION=NO, SHOW_SUMMARY=NO, OVERWRITE_RESULTSFILE=NO
                            END OF MEASUREMENT FOR
      PN=Test_2024.1              DWG=              SN=
    TOTAL # OF MEAS =0      # OUT OF TOL =0      # OF HOURS =00:00:00
```

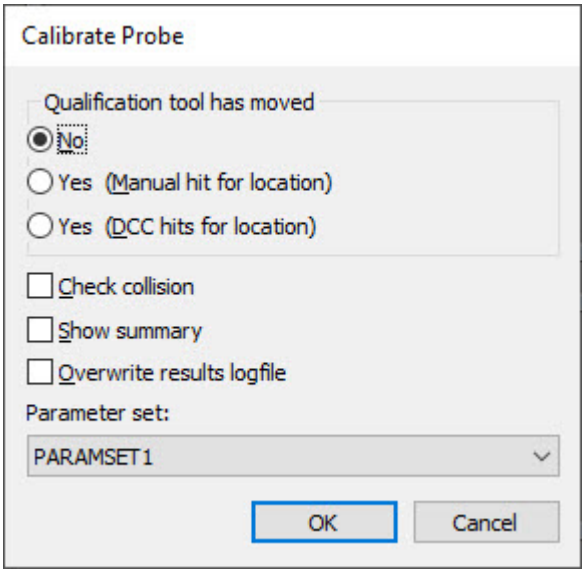
SHOW_SUMMARY= This YES / NO field determines whether or not PC-DMIS displays a summary of the calibration.

OVERWRITE_RESULTSFILE= This YES / NO field determines whether or not PC-DMIS overwrites or appends information sent to the results file. This is the same results file that's referenced when calibrating interactively.

In this example, the AUTOCALIBRATE command performs the calibration using a custom parameter set named PARAMSET1. The ASSIGN commands get the parameter set name from the command. The COMMENT command then displays the parameter set name in an Operator comment:

```
AUTOCALIBRATE/PROBE, PARAMETER_SET=PARAMSET1,
QUALTOOL_MOVED=NO,

CHECK COLLISION=Y, SHOW_SUMMARY=NO,
OVERWRITE_RESULTSFILE=NO

ASSIGN/AC=GETCOMMAND("Autocalibrate Active Probe",
"UP", 1)

ASSIGN/PSET=GETTEXT("Parameter Set",0,AC)

COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,

"The parameter set is " + PSET
```

You can press F9 on the command block to edit the command block through the **Calibrate Tip** or **Calibrate Probe** dialog box:



Calibrate Probe Dialog Box

If the Leg Collision check is disabled, the **Check collision** check box is unavailable and unmarked.

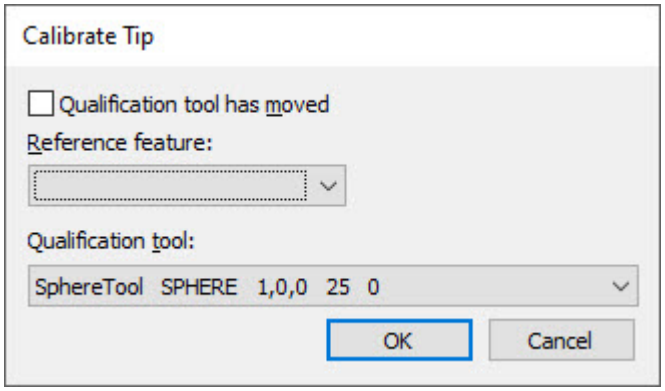
You can also insert a command to automatically calibrate multiple arms. For more information, see "Performing an Automatic Calibration" in the "Using Multiple Arm Mode" chapter.

To Automatically Calibrate a Single Tip

You can also perform an automatic calibration of the active tip by following this procedure:

1. Click the **Command Mode** button from the Edit Window toolbar to place the Edit window in Command Mode.
2. From the **Settings** toolbar, select the tip you want to calibrate. A new `TIP` command appears in the Edit window.
3. PC-DMIS requires a measurement of your calibration sphere in order to do this type of calibration. Create or insert a measured or auto sphere feature of the calibration tool to use in the calibration process. This sphere feature must appear sometime *after* the `TIP` command in the Edit window.
4. On any line following the sphere feature you'll be using, access the **Insert | Calibrate | Single Tip** menu option.
5. PC-DMIS inserts a `CALIBRATE ACTIVE TIP WITH FEAT_ID` command block into the Edit window. Note that this command does not communicate at all with the actual machine controller but instead sends data to the probe file.

6. Click anywhere on this command block, and press F9. The **Calibrate Tip** dialog box appears.



Calibrate Tip Dialog Box

7. Select the sphere feature to be measured by the selected tip from the **Sphere Feature** list.

8. Select the desired calibration tool from the **Qualification Tool** list.

9. If the calibration tool has moved since your last calibration, mark the **Qualification tools has moved** check box.

10. Click **OK** to update the calibration command block with your changes. For example, this example command block shows that the calibration tool is named *MyTool*, and that the sphere feature to use for this calibration is *SPH1*.

```
CALIBRATE ACTIVE TIP WITH FEAT_ID=SPH1,
QUALTOOL_ID=MyTool, MOVED=NO
Axis     THEO         MEAS         DEV          STD DEV
X        0.0000       8.0080       8.0080       -
Y        0.0000       1.0000       1.0000       -
Z        0.0000       0.9500       0.9500       -
DIA      2.0000       1.0000       1.0000       0.0000
```

11. Mark this command block and execute the measurement routine. PC-DMIS calibrates the active tip when it reaches this command block.

Custom Probe

You can define a custom probe by selecting **Edit | Preferences | Custom Probe Builder Utility** from the main menu. You can use the **Custom Probe** option to define custom probes that you can then insert into measurement routines.

When you select the **Custom Probe** option from the **Preferences** submenu, PC-DMIS displays a prompt, asking if you want to open an existing Probe Builder data file (*.dat) or create a new one.

Create a New Custom Probe

1. At the prompt, click **New** to display the **New** dialog box.
2. Navigate to the folder where you want to store the Custom Probe Builder file.
3. Type the name of the new custom probe data file (*.dat) and then click **OK**. The probe file is created and the Custom Probe Builder Utility dialog box appears.

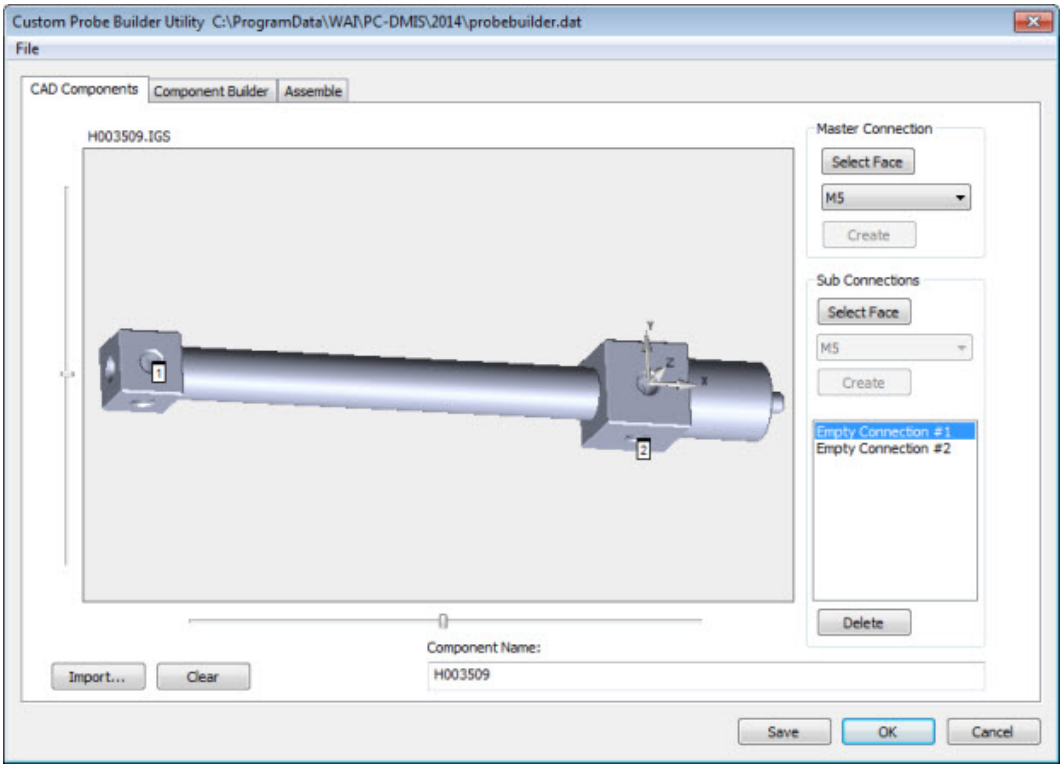
Open an Existing Custom Probe

1. At the prompt, click **Open** to display the **Open** dialog box.
2. Navigate to the folder where the Custom Probe Builder (*.dat) file is located.
3. Select the file and click **Open**. The **Custom Probe Builder Utility** dialog box appears.

You can manually define a custom probe by editing the usrprobe.dat file. For details, see the "Editing the usrprobe Data File" topic.

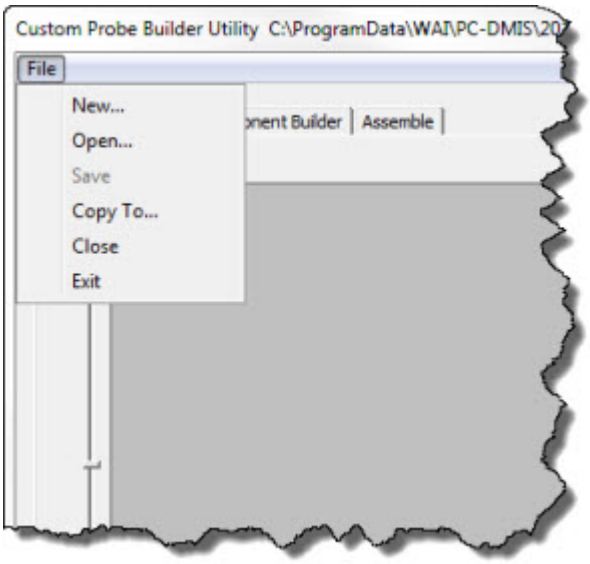
Custom Probe Builder Utility

Use the **Custom Probe Builder Utility** to create a custom probe. You can then use this probe in new or existing measurement routines.



Custom Probe Builder Utility Dialog Box - CAD Components Tab

Custom Probe Builder Utility File Menu

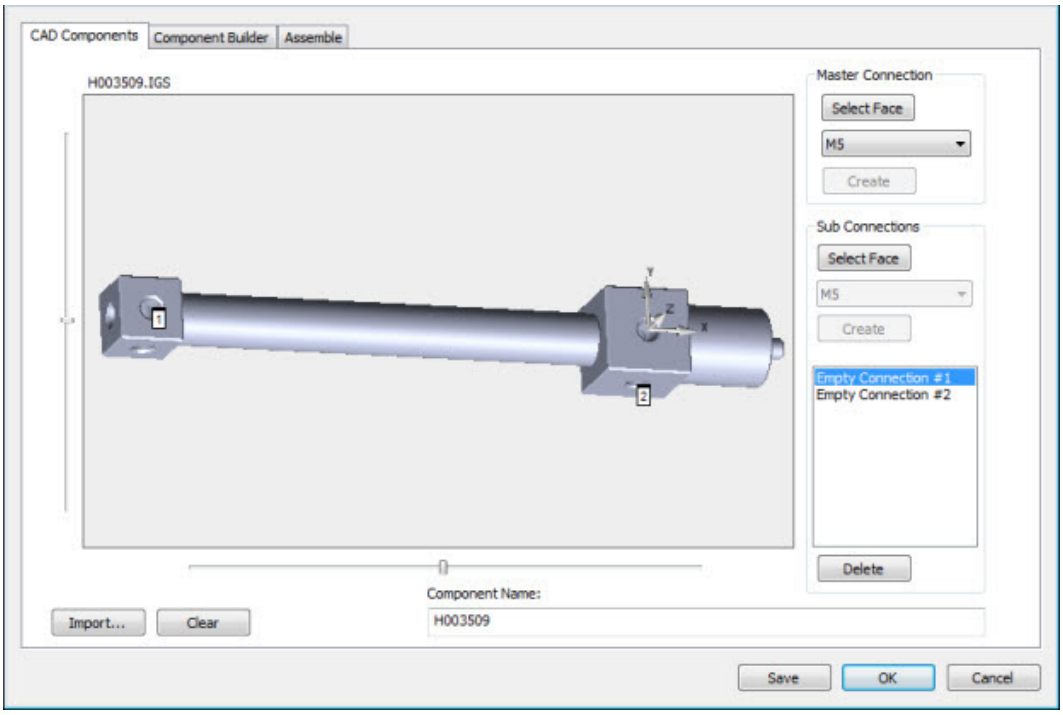


Custom Probe Builder Utility Dialog Box - File menu Options

The **File** menu options for the **Custom Probe Builder Utility** are:

- **New** - Displays the **New** dialog box. You can use this to enter the filename of the new .dat custom probe file. You can then navigate to a location on a local or network drive to store the file.
- **Open** - Displays the **Open** dialog box. You can navigate to a location on a local or network drive to open an existing custom probe .dat file.
- **Save** - Saves any changes made to the current .dat custom probe file.
- **Copy To** - Opens a **Copy part file to** dialog box. You can then browse to a location to save the current .dat part file.
- **Close** - Closes the current .dat file.
- **Exit** - Exits the utility.

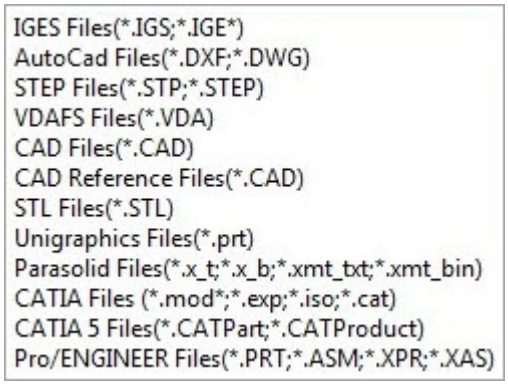
Custom Probe Builder Utility - CAD Components Tab



Custom Probe Builder Utility Dialog Box - CAD Components Tab

The options available on the dialog box are:

- Click **Import** to import most of the formats PC-DMIS allows from the local or network drive location into the measurement routine. These are the allowed file types.



- Click **Clear** to clear any unsaved work and start work on another item.
- Click **Save** to save any changes to the defined custom probe .dat file. The dialog box remains open if you need to make additional edits.
- Click **OK** or **Cancel** to close the dialog box. The application does not automatically save your changes. Make sure you click **Save** prior to clicking **OK** or **Cancel**.

Defining the Master Connection

The Master Connection defines the main connection point of a component once you add it to an assembly. It also serves to define the alignment of the assembly. Left-click a surface or feature on the CAD model to create the master connection points.

To configure a master connection type:

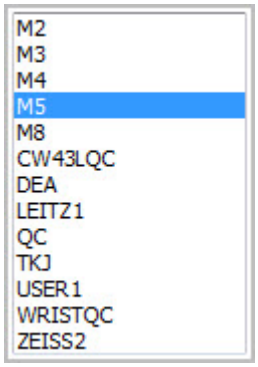
1. Click **Import** to open a CAD file. Once it imports, you can interact with the model and define connection points for other probe components.

   > If you opened a file with all your connection points defined, click the **Component Builder** tab to build custom probe components.

   You can zoom and rotate the image in the dialog box's graphic area similar to the PC-DMIS Graphic Display window. For example, you can press Ctrl + Z to re-draw the image to fit entirely within the graphics area. Use the sliders along the bottom and left of the graphic area to manipulate the CAD model.

2. Click the **Select Face** button from the **Master Connection** section to place the dialog box into a selection mode. The dialog box remains in this mode until you click the **Select Face** button again or you define a master connection.

3. From the list in the **Master Connection** area, select a connection type.

   Custom Probe Builder Utility Dialog Box - CAD Components Tab Master Connection Types menu

4. Click the surface or feature for the master connection. The software highlights the selected surface.

5. Click **Create** to finalize the master connection. The alignment updates to the selected feature with the Z axis perpendicular to it.

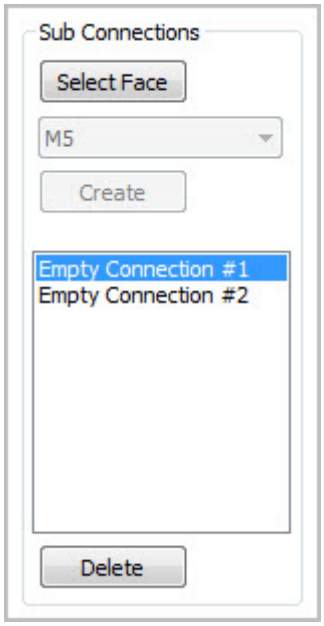
Defining Sub Connections

You can connect other components to sub connections to create an assembly. The Master Connection of one component connects to the selected sub connection of another component. For details on creating assemblies, see the topic "Custom Probe Builder Utility - Assemble Tab".

To define sub connections:

1. Click the **Select Face** button from the **Sub Connections** section to place the dialog box into a selection mode. The dialog box remains in this mode until you click the **Select Face** button again.

2. Select a connection type from the list in the **Sub Connection** area.
3. Click the surface or feature for the sub connection. The software highlights the selected surface.
4. Click **Create** to finalize the sub connection. Once you create the sub connection, the **Create** button is disabled until you select another feature. Create as many sub connections as needed. As you create sub connections, they appear in the lower list box as "Empty Connection #1", "Empty Connection #2", and so forth.



Custom Probe Builder Utility Dialog Box - Sub Connections Area

When you click **Create**, the software disables the lower connection type list box.

5. You can delete connections. Select one or more connections, then click **Delete**.
6. Once you have defined all master and sub connections, in the Component Name box, enter a name for the new component, then click **Save**. The dialog box remains open so you can build components from the **Components Build** tab or probe assemblies from the **Assemble** tab.
7. Click **OK** to exit the utility or click **Cancel** to exit the utility.

Custom Probe File Format

The software saves your custom probe configuration as a .dat file. This uses the same format as the probe.dat file.

```
ITEM:5H003512 M5

cadgeom 0.000 0.000 0.000 1.000 0.000 0.000 0.000 1.000
0.000 0.000 0.000 1.000 -1 5 H003512.draw

connect -1.108 -27.715 -113.565 0.508 0.759 0.408 M5

connect 2.859 33.883 -112.237 0.479 -0.789 -0.384 M5
```

Custom Probe Builder Utility - Component Builder Tab

You can use the **Custom Probe Builder Utility Component Builder** tab to create custom probe components from element types.

To build a probe component:

1. In the **Component Name** box, enter a unique name.
2. From the **Element Type** area, select the type of element to create. You can create tips, extensions, or polygons. You can combine multiple elements of the same type to create more advanced components. However, you cannot create an extension and add a tip onto it from this tab.

   For example, to create a tip named "Tip1" from two tip element types (a Cylinder and a Sphere) you would:

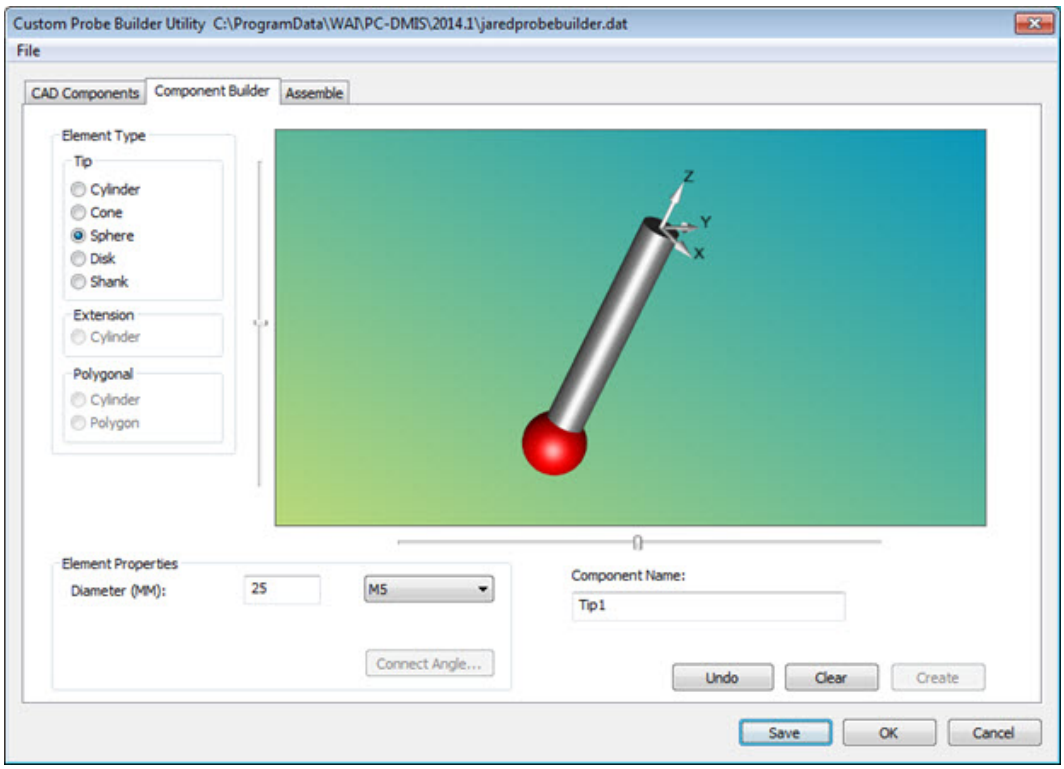
   a. Enter "Tip1" in the **Component Name** box.

   b. Select **Cylinder** from the **Tip** area of the **Element Type** section.

   c. Enter the diameter and height values for the cylindrical tip in the **Diameter** and **Height** boxes that appear, and then click **Create**.

   d. Select **Sphere** in the **Tip** area.

   e. Enter the diameter of the sphere in the **Diameter** box and click **Create**. The software draws each element in the dialog box's graphic area.



Custom Probe Builder Utility - Component Builder tab with a custom probe defined

f. Click **Save** to save your work.

The software indicates terminating elements in red. Terminating elements are the **Sphere**, **Disk**, and **Shank** tips. You cannot append other elements onto them. The example above shows a terminating **Sphere** tip connected to a non-terminating **Cylinder** tip.

The software indicates non-terminating elements in gray. Non-terminating elements are the **Cylinder** and **Cone**.

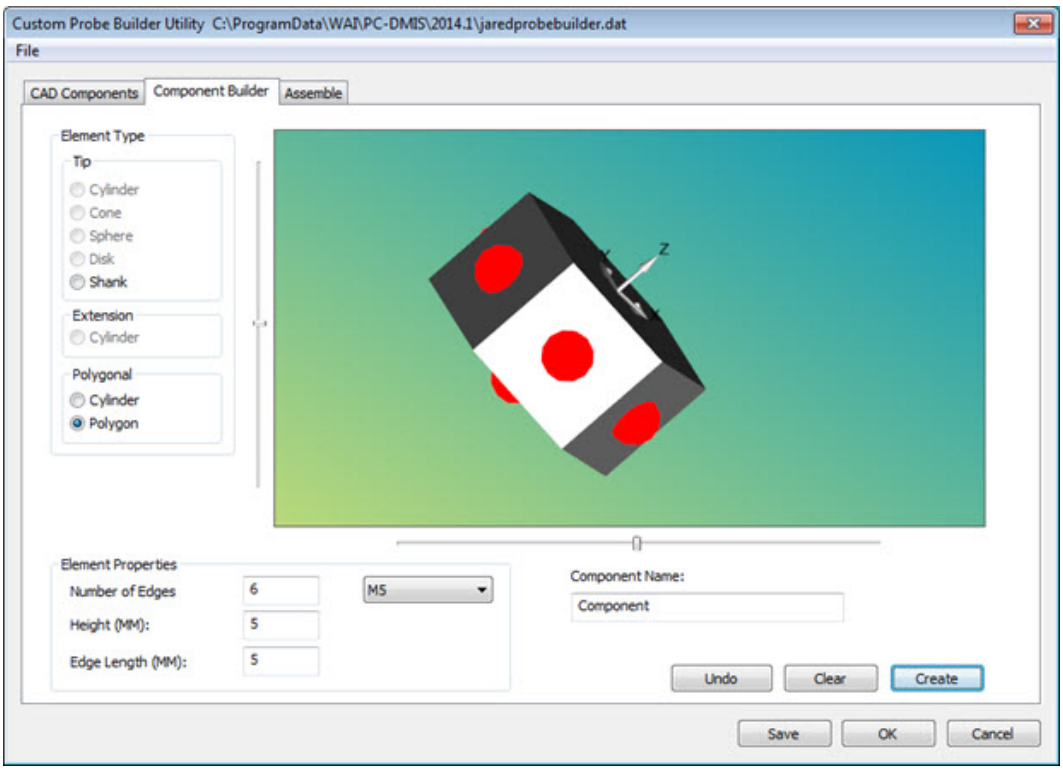
The software requires a unique name for each custom probe saved. If you want to continue to build a new component, click **Clear** to remove the current component.

About Element Properties

The **Element Properties** area changes based on the **Element Type** selected. All elements allow you to specify the connection type available from the drop-down list.

- Cylinders, Disks, and Shanks have **Diameter** and **Height** properties.
- Spheres have a **Diameter** property.
- Cones have **Diameter**, **Height**, and **Minor Diameter** properties.
- Polygons have **Number of Edges**, **Height**, and **Edge Length** properties.

For example, a Polygon component has these element properties:



Custom Probe Builder Utility - Component Builder tab with a Polygon created

Using Create, Undo, Save, and Clear

The **Create** button temporarily stores the built component. This lets you combine multiple element types of the same category together.

The **Undo** button removes the most recently created element from the current component. You can click **Undo** multiple times to remove each successive element until the graphic area empties.

The **Clear** button discards any created items or selected options and lets you start over with the creation process. If you have not saved your work, this permanently removes the current component.

The **Save** button saves your work to the .dat file you opened when you started the **Custom Probe Builder Utility**. Once you save a component, you can then use the **Clear** button to safely remove your work and clear the dialog box in order to create new components. This makes it so you don't have to close and reopen the dialog box to create new element types.

Zooming and Rotating

You can pan, zoom, rotate, and size the image in the graphic area:

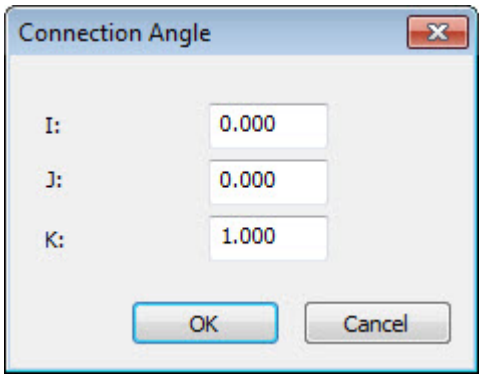
- To re-draw the image to fit entirely within the graphic area, press Ctrl + Z.
- To rotate the image three-dimensionally, press Ctrl, right-click, and then drag the cursor.

Creating a Cut Cylinder

From the **Extension** section, use the **Cylinder** option to cut a cylindrical extension at a defined angle. You can then attach the next extension at that angle to create a bend in the extension.

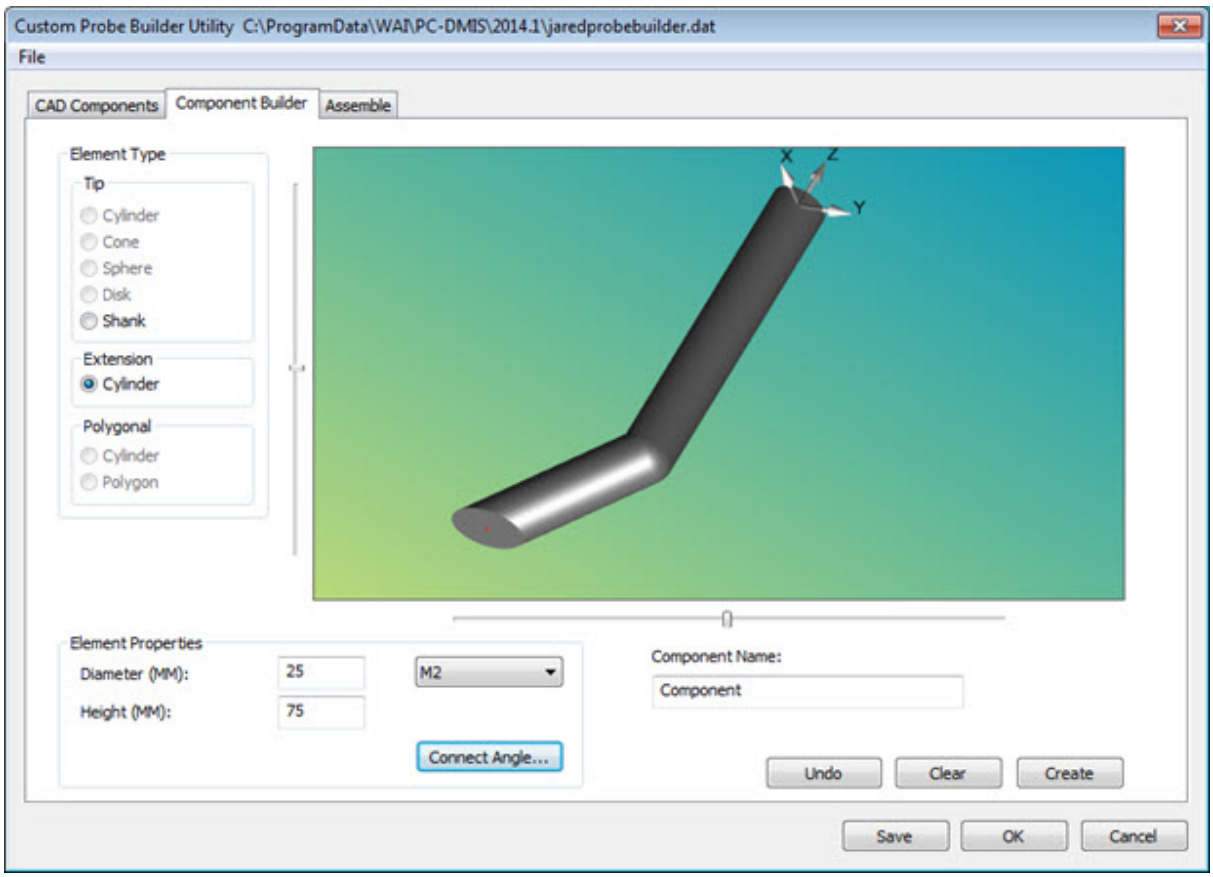
To create a cut cylinder:

1. In the **Component Name** box, enter the name of the cut cylinder.
2. In the **Extension** area of the **Component Builder** tab, click the **Cylinder** option to enable the **Connect Angle** button in the **Element Properties** area.
3. Click the **Connect Angle** button to display the **Connection Angle** dialog box.



Connection Angle dialog box

4. Enter the desired vector values and then click **OK**. These determine the angle that the software uses to cut the cylinder.
5. From the **Element Properties** area, in the **Diameter** and **Height** boxes, enter the diameter and height values for the cylinder.
6. Click the **Create** button to display the cut cylinder.

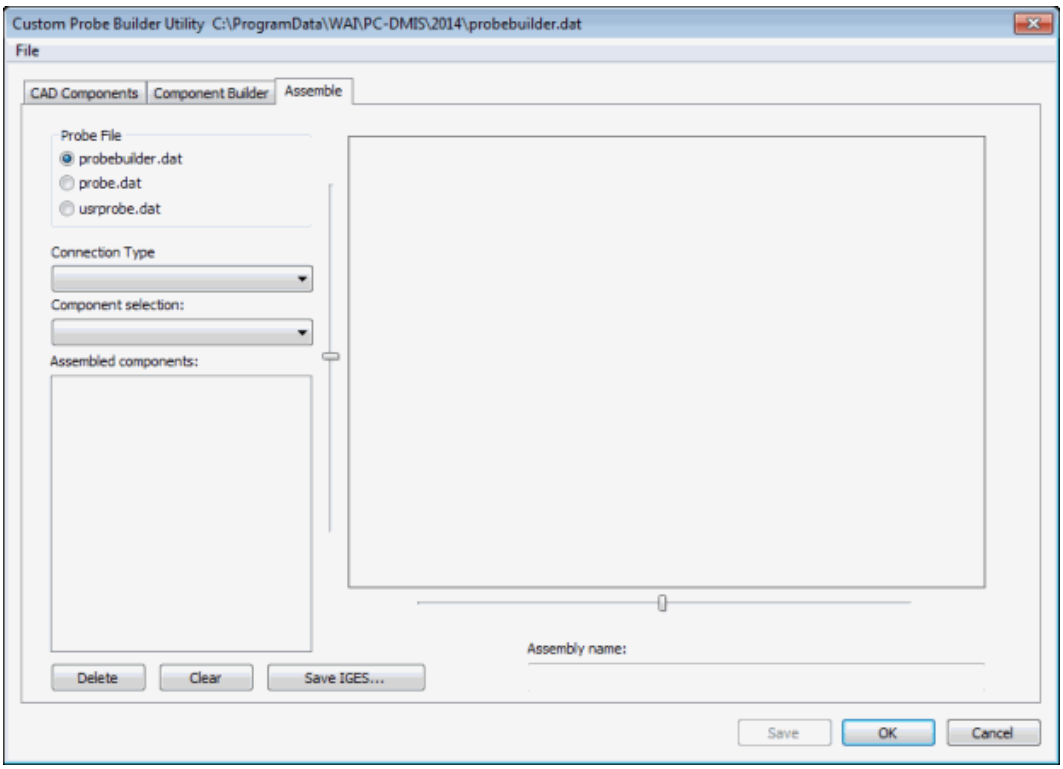


Custom Probe Builder Utility - Component Builder tab showing a sample Cut Cylinder

7. Click the **Save** button to save the new component.

Custom Probe Builder Utility - Assemble Tab

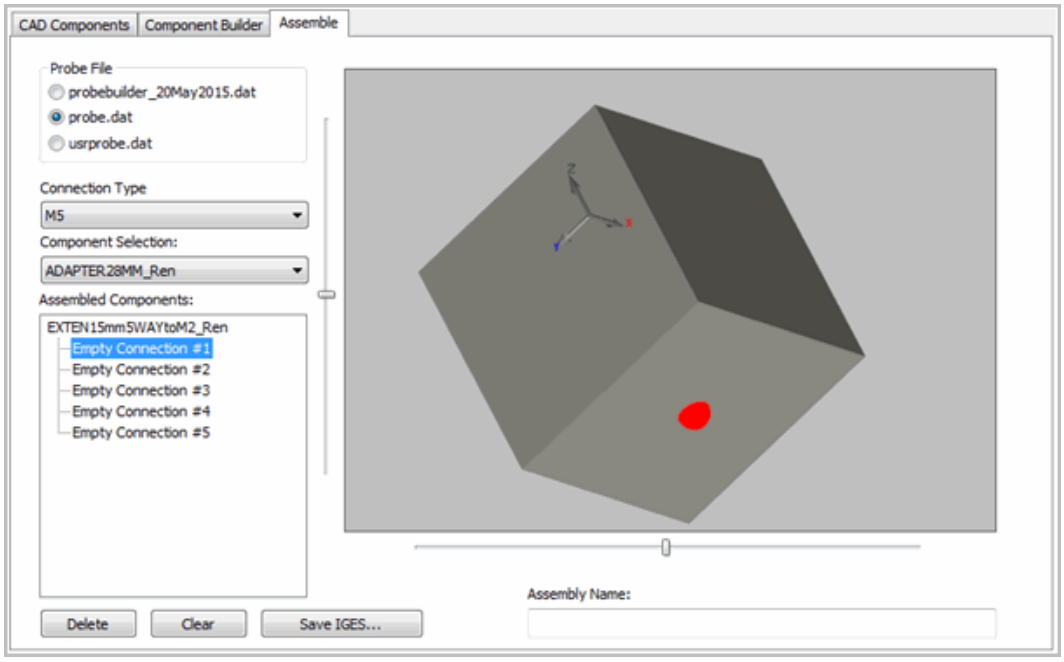
You can use the **Assemble** tab to assemble a custom probe component.



Custom Probe Builder Utility Dialog Box - Assemble Tab

To create an assembly from various components:

1. From the **Probe File** section, select a file.
2. From the **Connection Type** list, select the appropriate type of connection.
3. From the **Component Selection** list, select a component. The software updates the list based on the **Probe File** selection. The software uses the list to create an assembled component from the files, and then it sorts all the items in the list by their connection types. For example, if Connection #1 is defined as an M8 connection, only items that can connect to an M8 connection are available in the list.



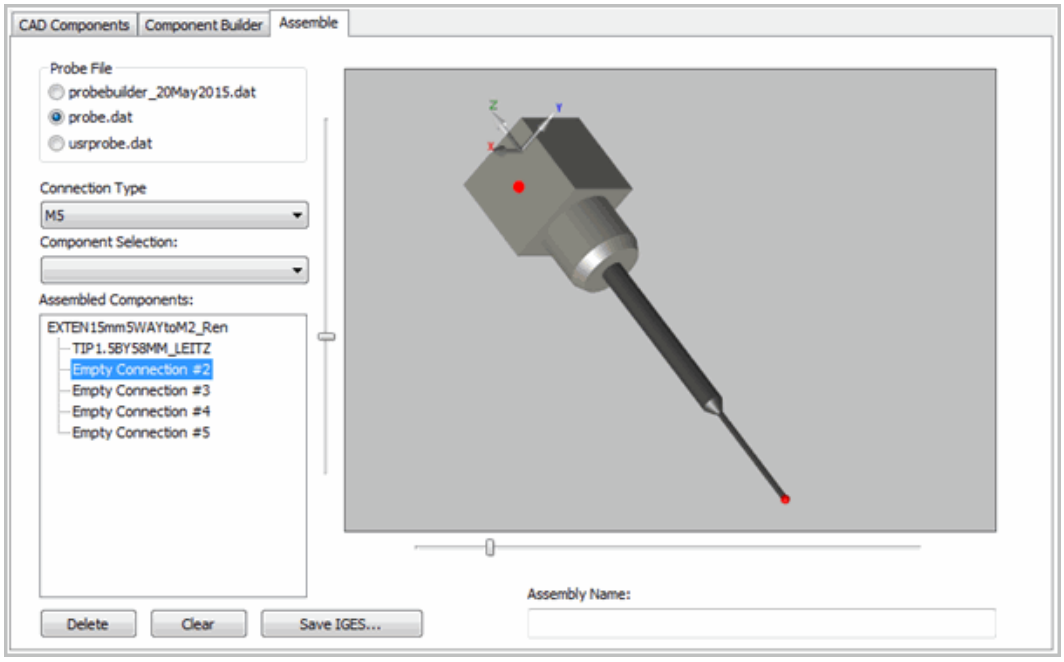
Custom Probe Builder Utility - Assemble tab showing the start of an assembly build

For example, in the **Assemble** tab shown above:

- The "probe.dat" file is selected in the **Probe File** area.
- The "ADAPTER28MM_Ren" component (found in the "probe.dat" file) is selected from the **Component Selection** list.
- The component is drawn in the graphic area. The red dot identifies the first available (empty) connection. You can zoom and rotate the image. For example, if you press Ctrl + Z on your keyboard, the software zooms the image to fit entirely within the graphic area.
- The **Assembled Components** area displays a tree view of the selected component and highlights the first empty connection. Each time you fill an empty connection, the software automatically selects the next available empty connection port. To remove a connection, you can left-click the item, and then click **Delete**.
- Once you add a component to the **Assembled Components** tree, the software enables the **Delete**, **Clear**, and **Save IGES** buttons.

4. From the **Probe File** area, select another file or use the currently selected file to continue to build the assembly.

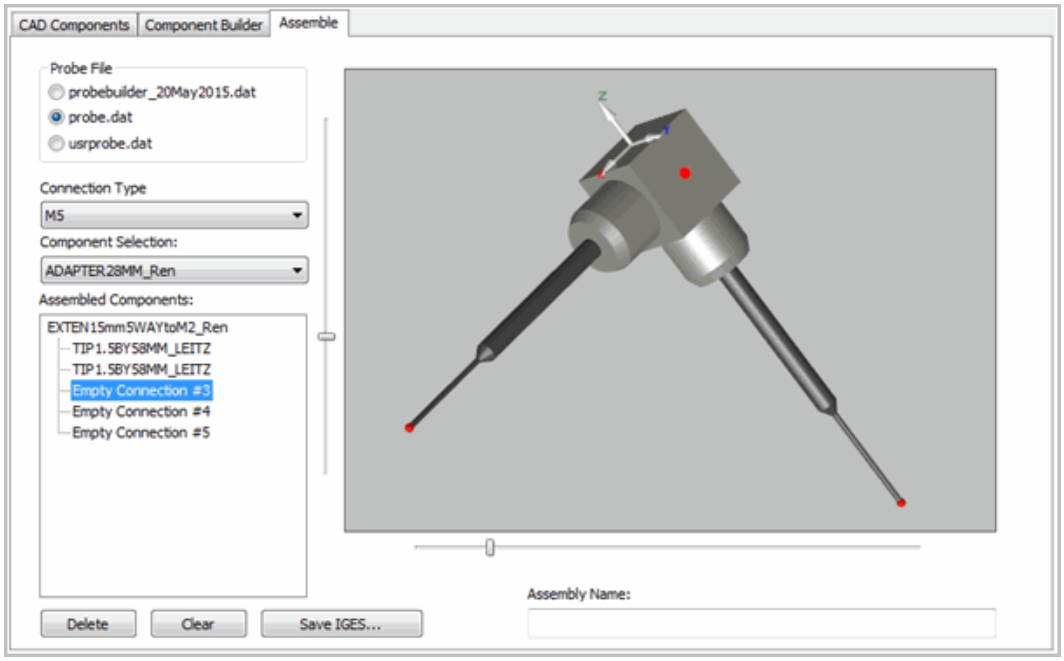
> At any point, you can change the **Probe File** option and select items you created in the **Component Builder** tab.



Custom Probe Builder Utility - Assemble tab showing the continuation of an assembly build

5. From the **Connection Type** list, select a new connection type. If an assembly already exists, a prompt asks if you want to clear the current build. Click **Yes** to clear the build and start over, or click **No** to return to the current build.

6. From the **Component Selection** list, select new components to add to the connections. Some components may have more connections that you can add onto, while others may end with terminating elements (Spheres, Disks and Shanks) as shown below:



Custom Probe Builder Utility - Assemble tab showing the completion of an assembly build

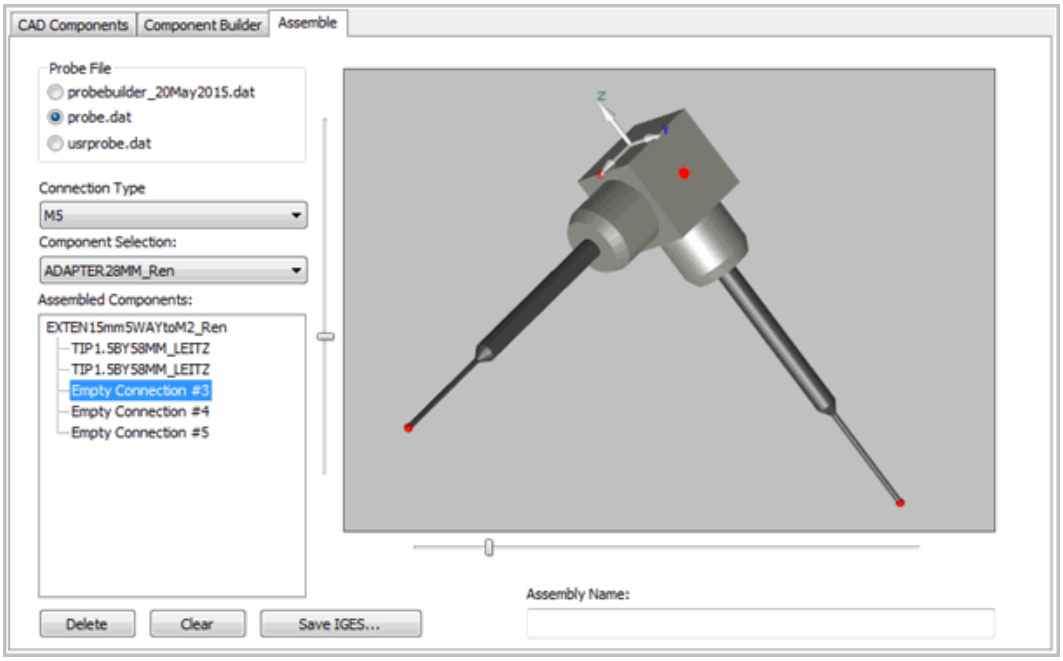
7. Update the assembly with any of these options:

- The **Delete** button works similar to the **Delete** button on the **Probe Utilities** dialog box. If you build a branch on the tree that contains multiple items, you can delete the entire branch or just portions of it. The **Delete** button only removes the selected item in the tree along with any attached components.
- The **Clear** button clears the entire tree and the graphic area.
- The **Save IGES** button saves the imported model to an IGES file. A standard **Save As** dialog box appears so you can specify a unique file name.

8. In the **Assembly Name** box, enter a name for the assembly, and then click **Save**.

Modifying the Rotation Angle of a Component

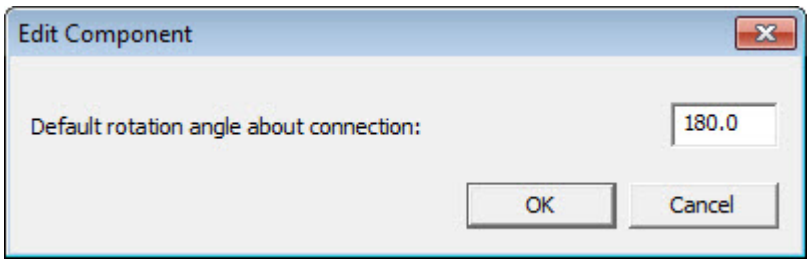
If a connected component has an incorrect orientation, you can rotate it.



Custom Probe Builder Utility - Assemble tab showing connectors requiring an orientation change

To correct the orientation of a connection:

1. Right-click an item in the connector tree to open the **Edit Component** dialog box.



Edit Component dialog box

2. In the **Default rotation angle about connection** box, enter the angle (any angle from +180° to -180° inclusive). 0 (zero) is the default angle.

3. Click **OK** to rotate the component in the graphic area.
4. Click **Save**.

Creating and Using Connection Points on a Polygon Element Example

Creating Connection Points

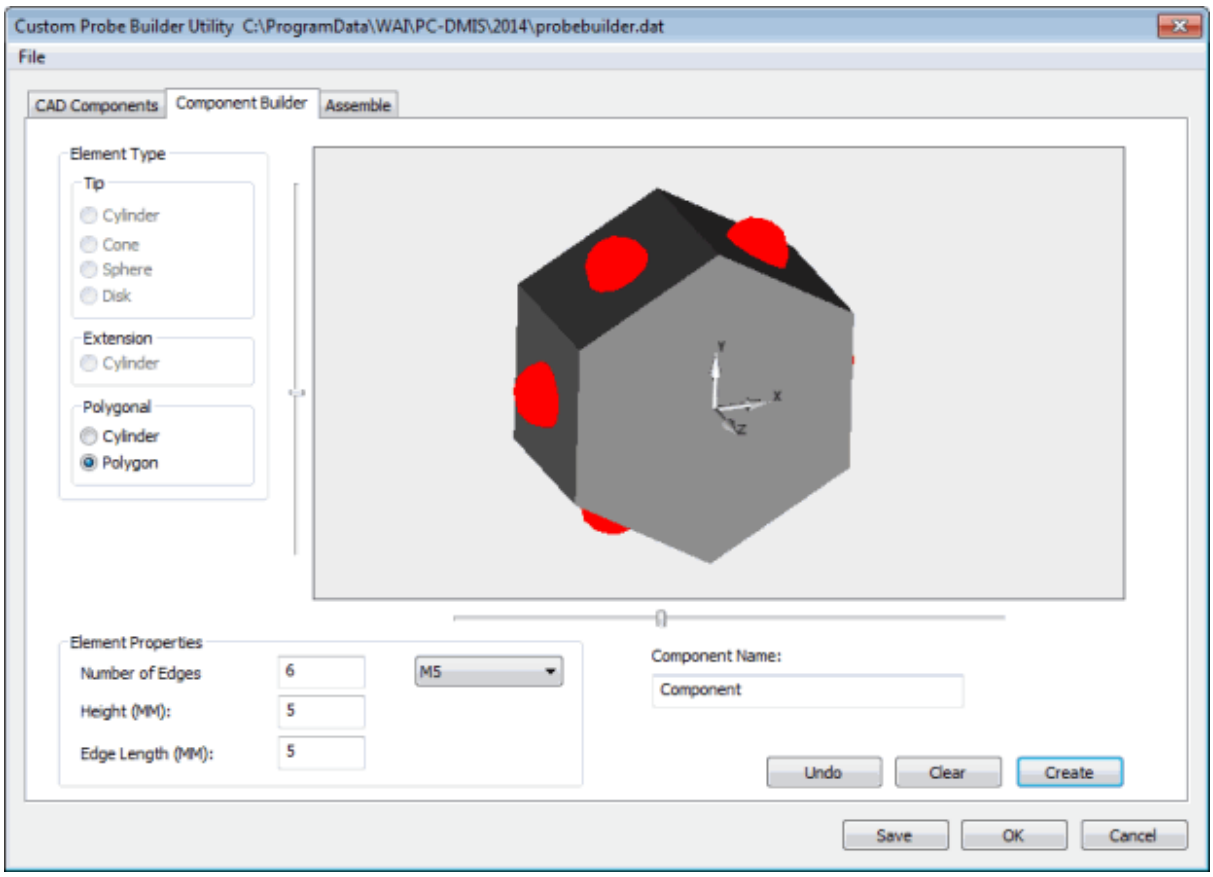
Connection points are used to define the faces to be used to attach components to when building an assembly. This allows more control over your assembly.

To create connection points on a Polygon element:

1. Select the **Polygon** option from the **Polygonal** section of the **Component Builder** tab.
2. Adjust the **Element Properties** as needed.

   The **Number of Edges** box accepts a minimum value of 3 and a maximum value of 20. If you enter a value less than 3, the box is reset to 3; if you enter a value greater than 20, the box is reset to 20.

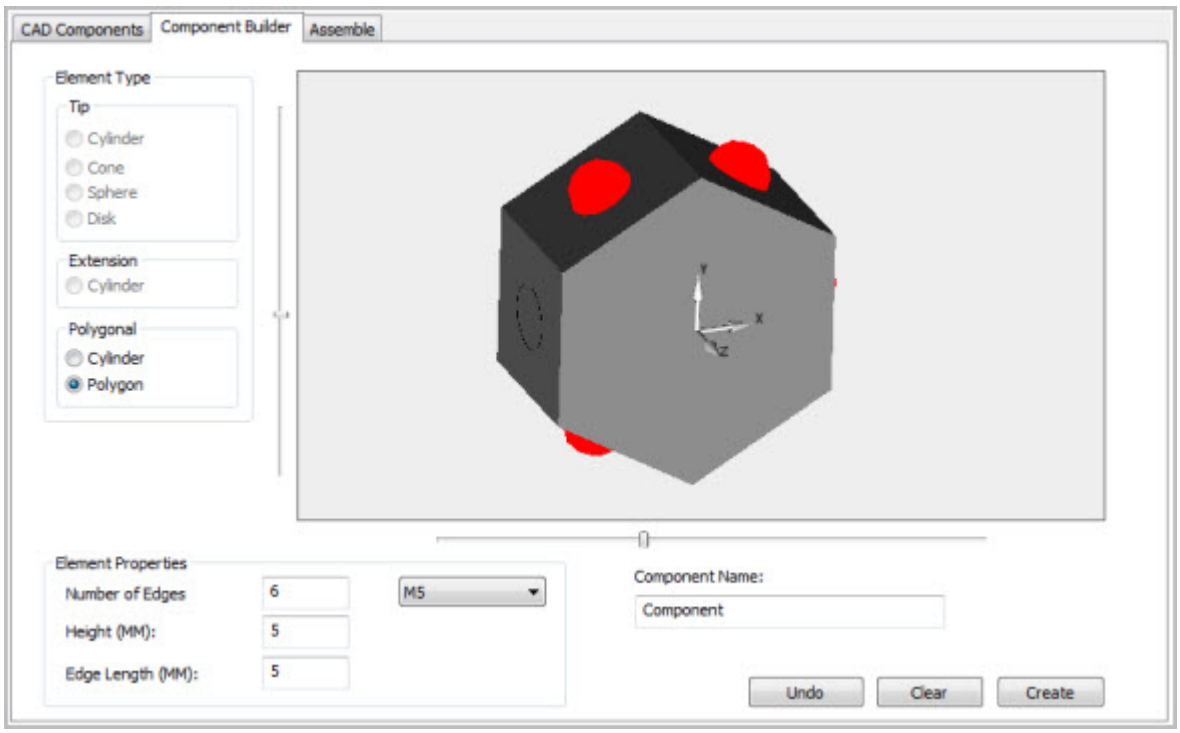
3. Enter a unique name for your component in the **Component name** section then click **Create**.



Custom Probe Builder Utility - Component Builder tab showing a polygon and its default connection points

A basic polygon is displayed with red connection dots. The red connection dots are displayed on five of the sides. Each dot represents a connection point that can be used in an assembly.

4. To manage which connection points are available for a particular assembly, click any of the connection point dots to disable or enable. In the image below, the front (left side) face connection is disabled. Clicking a disabled point re-enables it.



Custom Probe Builder Utility - Component Builder view of a connection point disabled

5. When all the properties and connection points have been defined, click **Save**. If this is a new component, enter a name in the **Component Name** field and then click **Save**.

Using Connection Points

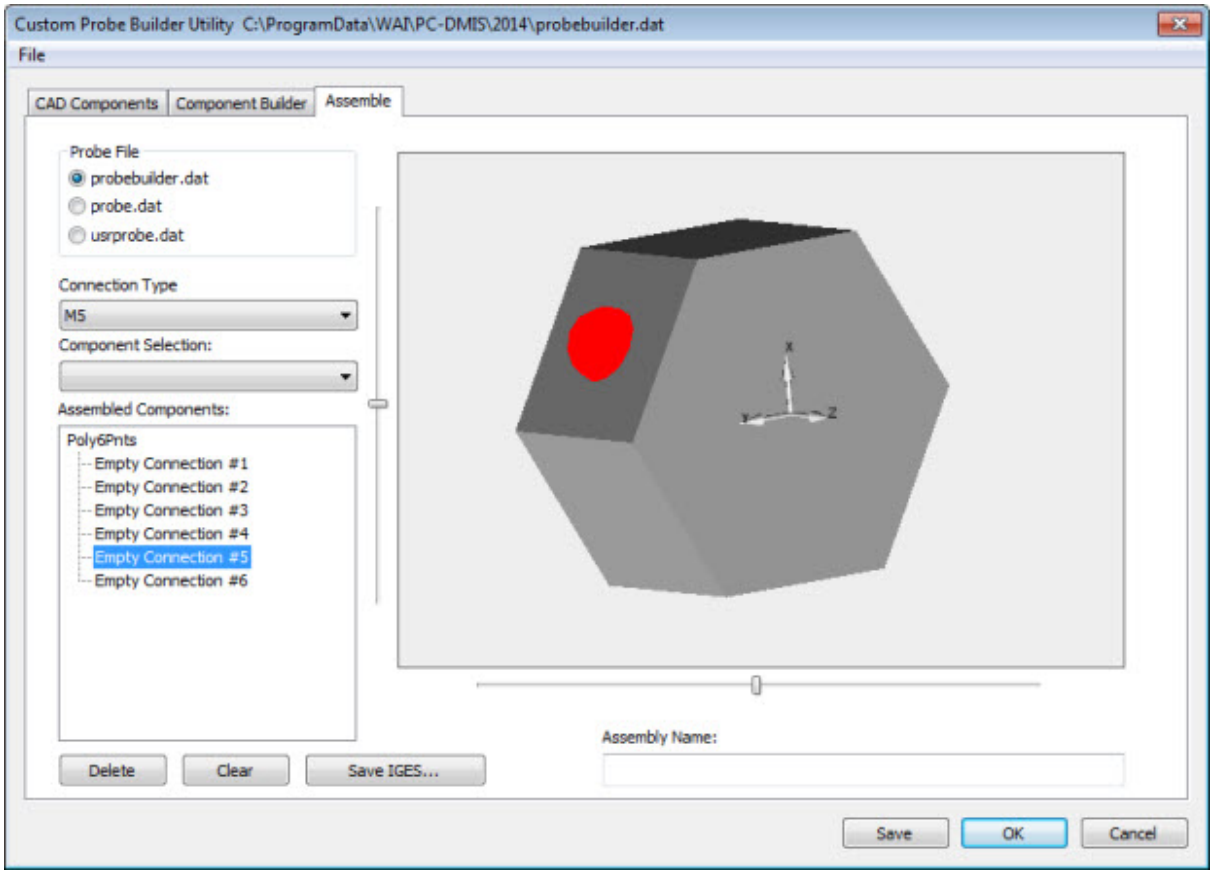
Once the changes in the **Component Builder** tab have been saved, they can be used to assemble a custom probe.

To assemble a custom probe using the built component:

1. Click the **Assemble** tab.
2. In the **Probe File** section, click the file name used to save your changes.
3. Click the type of connection from the **Connection Type** list. The components in the **Component Selection** list update based on the type of connection selected. Only items that connect to this connection type are listed.
4. The component assembly is detailed in the **Assembled Components** section. Click the connection you want to begin building on in the **Assembled Components** tree. The first "empty" connection is always selected by default and is highlighted. If the selected connection face is empty, it shows the red connection dot. Rotate the image if you can't initially see it.
5. Select a component from the **Component Selection** list.

   The selected connection in the **Assembled Components** section is the face that is updated when you begin building your assembly. To work on another face, click a different connection in the **Assembled Components** section. The selected face is

updated with the red connection dot to signify it is the face the next selected component is built onto.



Custom Probe Builder Utility - Assemble tab showing a polygon and the Connection #5 connection point

A face that already has a component connected to it can be replaced with a new component:

a. Click the component to be replaced in the **Assembled Components** tree.

b. Select a new component from the **Component Selection** list to replace it.

Remove a component from a connection point without replacing it:

a. Click the component in the tree to be deleted.

b. Click **Delete**.

6. Once you've selected the face to work on, click the next component from the **Component Selection** list. The selected item is automatically attached to the targeted face - the face with the red connection dot on it.

7. Repeat as needed until your assembly is complete.

8. When finished, type a name for your assembly in the **Assembly Name** box.

9. Save the changes:

- Click **Save IGES** to save the assembly as an IGES file.
- Click **Save** to save the assembly into the .dat file created or opened in the current session

Migrating Measurement Routines Between Machines Using PH10 and Tesa Star M Probes

In the **Option** section of the PC-DMIS Settings Editor, PC-DMIS provides the `AutoAdjustTesaM5Degrees` entry to assist in migrating measurement routines between two machines that use a PH10 probe on one machine and a Tesa Star M probe on the other. Both machines must have probe files with the same name (although the probe files themselves will be different).

- If the entry is set to 1, when a measurement routine that was created initially on one of these machines is then loaded onto the other, PC-DMIS converts any active `TIP/` commands to the closest AB tip angle based on the index increment. If any angles are converted, a message box appears displaying them.
- If the entry is set to 0, PC-DMIS does not perform this angle adjustment.

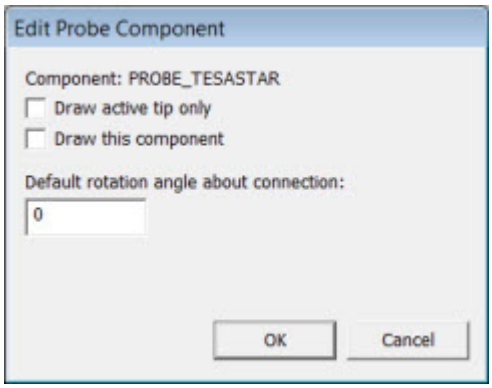
For details, see the "AutoAdjustTesaM5Degrees" topic in the "Option" section of the PC-DMIS Settings Editor documentation.

Showing and Hiding Probe Components

You can show or hide probe components in one of two ways:

From within the Probe Utilities Dialog Box

1. From the Edit window, select a `LOADPROBE` command.
2. Press F9 to access the **Probe Utilities** dialog box.
3. From the **Probe description** area, double-click on the probe component you want to show or hide to open up the **Edit Probe Component** dialog box.



Edit Probe Component Dialog Box

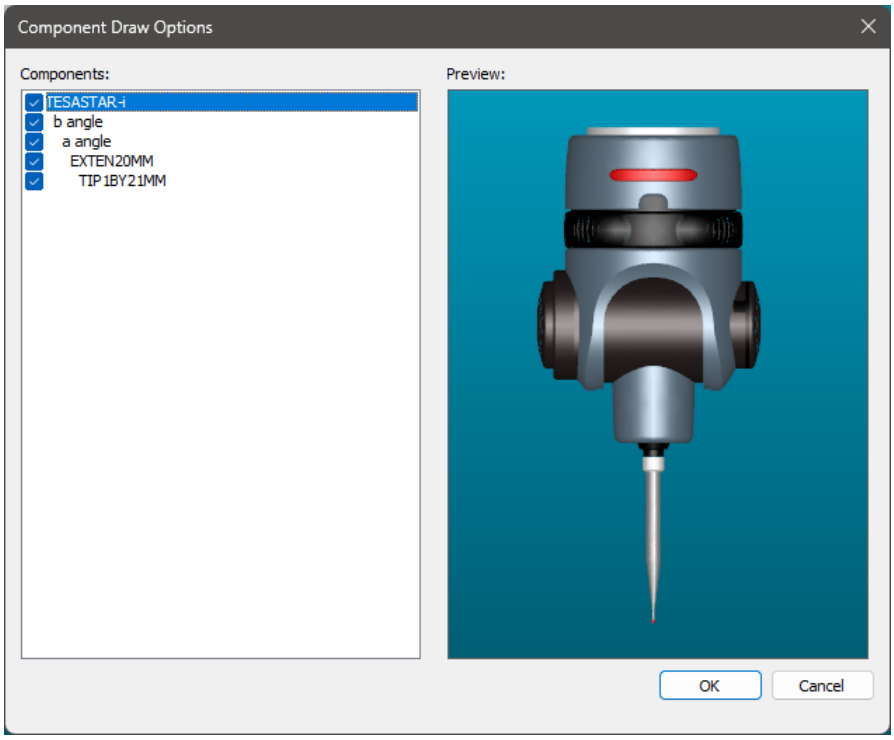
4. Mark or clear the **Draw this component** check box to show or hide that component.
5. Click **OK**.

PC-DMIS organizes the different components in a hierarchical list. This means:

- When you choose to hide a component, PC-DMIS also hides any components above that component.
- When you choose to show a component, PC-DMIS also shows any components below that component.

From the Graphic Items Toolbar

1. From the toolbar area, right-click and choose **Graphic Items** to show the **Graphic Items** toolbar.
2. From the toolbar, click the **Probe Component Draw Options** button to show the **Component Draw Options** dialog box.



Component Draw Options Dialog Box

3. From the **Components** list, mark or clear the component's check box to show or hide that component.
4. Click **OK**.

PC-DMIS organizes the different components in a hierarchical list. This means:

- When you choose to hide a component, PC-DMIS also hides any components above that component.
- When you choose to show a component, PC-DMIS also shows any components below that component.

Editing the usrprobe.dat Data File

This topic describes how probes are created and stored in the probe.dat when you define probes in PC-DMIS. You can use this information as a model to manually create probes in the usrprobe.dat file.

Definition of the Probe.dat

The **probe.dat** file controls the graphical and numerical representation of the probe system for PC-DMIS. If you need a custom probe or probe system within PC-DMIS, you need to first define it in the probe.dat file. You need to strictly follow a specific format to define it.

The content below describes the format and rules of the format with a list of the commands and their definitions.

```
ITEM:TIP2BY20MM M2
```

In the above code snippet, the following rules apply:

1. The first set of characters following the colon "**ITEM:**" is the name that appears in the **Probe Utilities** dialog box when you construct a probe setup. The text following the colon after the keyword **ITEM:** is a unique name of your choice to identify the probe configuration.

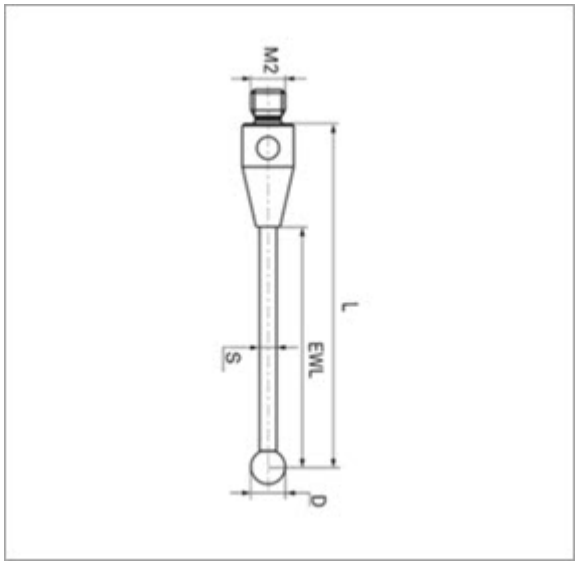
   You can find details in the **Format of the Probe or Probe System ITEM** section below.

2. The second set of characters, "**M2**" in the above example, defines the type of thread or connection type this item has.

The following information lists and defines the probe builder commands:

`EffectiveWorkingLength`
This parameter defines the depth that you can insert the probe assembly into a feature that allows a valid measurement with the probe tip. This is a way for you to define the usable length of the probe assembly to something less than its total length if needed or desired and only applies to cylinder features.

The default Effective Working Length (EWL) value is the distance from the center of the tip's ball to the end of the stylus.



Example details of an M2 stylus showing the Effective Working Length (EWL)

If you do not set a value, PC-DMIS uses the default value for this parameter.

`begintip`
The start of a tip definition.

`endtip`
The end of a tip definition.

`ribcount N`
Defines the number of lines in a circular feature.

N - The number of lines seen in a circular feature. This is an integer from 2 to 1000, inclusive.

`line x1 y1 z1 x2 y2 z2`
Defines a line feature.

x1, **y1**, **z1** - The coordinate location starting point of the line.

x2, **y2**, **z2** - The coordinate location end point of the line.

`sphere x y z d`
Defines a sphere feature.

x, **y**, **z** - The center coordinate location of the sphere.

d - The diameter of the sphere.

`cutsphere x y z i j k d t b`
Defines a cut sphere feature.

x, **y**, **z** - The center coordinate location of the sphere.

d - The diameter of the sphere.

i, **j**, **k** - The vector perpendicular to the sphere top and bottom.

t, **b** - The location of the top and bottom as follows:

- Top = (x,y,z) + t * (i,j,k)
- Bottom = (x,y,z) - b * (i,j,k)

`cylinder x1 y1 z1 x2 y2 z2 d`
Defines a cylinder feature.

x1, **y1**, **z1** - The coordinate location of the top of the cylinder.

x2, **y2**, **z2** - The coordinate location of the bottom of the cylinder.

d - The diameter of the cylinder.

`cone x1 y1 z1 d1 x2 y2 z2 d2`
Defines a cone feature

x1, **y1**, **z1** - The coordinate location of the top of the cone.

x2, **y2**, **z2** - The coordinate location of the bottom of the cone.

d1 - The diameter at the top of the cone.

d2 - The diameter at the bottom of the cone.

`ring x1 y1 z1 i1 j1 k1 d1 d2`
Defines a ring feature.

x1, **y1**, **z1** - The center coordinate location of a ring.

i1, **j1**, **k1** - The vector of the ring with respect to the current coordinate system of the probe definition.

d1 - The diameter of the outside of the ring.

d2 - The diameter of the inside of the ring.

`comment text`
An author message that describes the code or important information related to the code. The probe system does not act upon the text that follows the comment command.

`hotspot x1 y1 z1 i1 j1 k1 d1 t1 type`
Defines a hotspot command.

x1, **y1**, **z1** - The coordinate location of the center of the probe tip.

i1, **j1**, **k1** - The vector of the probe tip, usually 0,0,1.

d1 - The diameter of the tip.

t1 - The thickness of the tip (for disk probe).

type - The type of probe (ball, shank, disk, and so on).

`opticalcenter x1 y1 z1 i1 j1 k1 sch=1 minf maxf optics_type min_na max_na mag type`
Defines an optical center.

Example 1:

```
opticalcenter x1 y1 z1 i1 j1 k1 sch=2 ot magsteps fov1
fov2...fovmagsteps, na1, na2,...namagsteps, auxlensmag
type
```

x1, **y1**, **z1** - The center location of the center of the probe tip.

i1, **j1**, **k1** - The vector of the probe tip, usually 0,0,1.

sch - The schema number for the optical center data.

ot - The optics_type (1: encoder infinite zoom, 2: fixed optics, 3:nonencoder zoom, 4:encoder discrete zoom, 5: turret microscope, 6: DCC turret microscope).

magsteps - The count of FOV (Field of View) and numeric aperture values.

fov - The X FOV size for each magnification step or min and max if infinite zoom.

na - The numeric aperture for each magnification step or those corresponding to the min and max FOV size if infinite zoom.

auxlensmag - The magnification of any attached auxiliary lens.

type - The type of probe (ball, shank, disk, camera, and so on).

Example 2:

opticalcenter x1 y1 z1 i1 j1 k1 sch=3 ot magsteps fov1 fov2...fovmagsteps, na1, na2,...namagsteps QRDsteps qrdfovsize1 qrdfovsize2… qrdfovsizeQRDsteps qrd1 qrd2…qrdQRDsteps auxlensmag type

x1, **y1**, **z1** - See above.

i1, **j1**, **k1** - See above.

sch - See above.

ot - See above.

magsteps - See above.

fov - The X FOV size for each magnification step or min and max if infinite zoom.

na - See above.

QRDsteps - The count of qrdfovsize (QRD FOV size in X) and qrd (quadratic region depth) values.

auxlensmag - See above.

type - See above .

`connect x1 y1 z1 i1 j1 k1 take`

Defines a connection point.

x1, **y1**, **z1** - The location of a connection point on the probe system.

i1, **j1**, **k1** - The vector of the connection based on its orientation to the current coordinate system of the probe system. It always points toward the center.

take - The type of thread or connection type that the connect point is compatible with.

`autojoint x1 y1 z1 I1 j1 k1 ina inc sa ea label`

Defines an auto joint feature.

x1, **y1**, **z1** - The location of the joint with respect to the current coordinate system.

i1, **j1**, **k1** - The vector of the joint based on the current coordinate system. It points back toward the startup location.

ina - The default initial angle of this joint.

inc - The increments of this joint.

sa - The starting angle of this joint.

ea - The ending angle of this joint.

label - The name of the joint that appears in the **Probe Utilities** dialog box.

`manualjoint x1 y1 z1 i1 j1 k1 ina inc sa ea label`

Defines a manual joint feature.

x1, **y1**, **z1** - The location of the joint with respect to the current coordinate system.

i1, **j1**, **k1** - The vector of the joint based on the current coordinate system. It points back toward the startup location.

ina - The default initial angle of this joint.

inc - The increments of this joint.

sa - The starting angle of this joint.

ea - The ending angle of this joint.

label - The name of the joint that appears in the **Probe Utilities** dialog box.

```
face N x1 y1 z1 ... xN yN zN
```
Defines a face feature.

N - The number of corner points on the given face.

x1, **y1**, **z1** through **xN**, **yN**, **zN** - The coordinates of each corner point of the face.

```
solid N
```
Defines a solid feature.

solid - Applies shading. If you don't use the **solid** command, the faces appear as wireframes. You use this with the "**face**" command to indicate the number (**N**) of "face" commands.

N - The number of face commands which follow. This determines the number of faces to which to apply shading.

```
cadgeom X Y Z II IJ IK JI JJ JK KI KJ KK Level FileName
```
Defines the CAD geometry transformation movements for a specific CAD file.

X, **Y**, **Z** - The coordinate location start point. Moves the origin of the CAD file to the start point of the drawing for the probe file.

II - **KK** - The set of three transformation matrices to rotate the CAD csy to the probe csy. If no changes are required, the nominal values would be:
```
cadgeom X Y Z 1 0 0 0 1 0 0 0 1 Level FileName
```

Level - The CAD level number as assigned from within PC-DMIS. If no CAD level exists or if there are multiple levels, you may need to create a new level inside PC-DMIS.

FileName - A .draw file which is nothing more than a renamed .cad file.

```
cutcylinder X1 Y1 Z1 X2 Y2 Z2 I2 J2 K2 Diameter
```
Defines the cut cylinder geometry transformation for a specific CAD file.

X1, **Y1**, **Z1** - The coordinate location start point for the cut cylinder. This moves the origin of the CAD file to the start point of the drawing for the probe file.

X2, **Y2**, **Z2** - The cylinder edge coordinate location of the drawing for the probe file.

I2, **J2**, **K2** - The edge vector of the cut cylinder.

Diameter - The diameter value of the cut cylinder.

Format of the Probe or Probe System ITEM

Each new probe configuration that is defined in PC-DMIS is added to the file "probe.dat". It begins with the "**ITEM:**" command. This command contains both a distinct naming description and a connection identifier. This identifier specifies the type of object to which it can connect, as described above. Below, you can see the available connection identifiers with a brief description. For a probe tip, only the second line of the probe configuration must be a "**begintip**" command, and the very last line a "**endtip**" command.

To define a probe configuration, you must work with a top-down approach. This means that your initial starting point is x=0, y=0, z=0 and everything is drawn from that point down. Consequently, all z values should have a negative value. Also, for most items, the 0, 0, 0 location is the center of the item. This makes it possible to have both positive and negative numbers in both the X and Y directions. Positive X always moves from left to right initially, and Positive Y always moves from front to back as if you were looking FROM the positive Z direction.

The only two commands that define an active configuration are "**hotspot**" for tips and "**connect**" for all others. You must have one of these two commands present in any configuration, but they are never present at the same time.

- "**hotspot**" defines the center location of any portion of a configuration that can be used to actually touch a part for measurement purposes.
- "**connect**" defines any location where another item can be added to the current item. Each connection must include a Connection Identifier to define the type of item it can take.

The two commands that can change the coordinates system of the probe configuration are "**autojoint**" and "**manualjoint**". Take special care when you use any of these two commands, because each command transfers the origin of the configuration to the center of the joint. Keep in mind that the first "joint" command changes the axes of the coordinate system. What was the -Z direction now becomes the +X direction, and what was the +X direction now becomes the +Z direction. Y remains the same. These commands are usually used in pairs. For example, one command controls the B rotation of a PH9 while a second controls the A rotation of a PH9. Also, the first command of the pair always controls the B rotation and the second always controls the A rotation.

Connection Identifier Listing

M2
Specifies that it can connect a M2 threaded device.

M3
Specifies that it can connect a M3 threaded device.

M4
Specifies that it can connect a M4 threaded device.

M5
Specifies that it can connect a M5 threaded device.

M8
Specifies that it can connect a M8 threaded device.

QC
Specifies that it can connect a Quick Connect device.

ARM
Specifies that it connects directly to the machine arm.

OPT
Specifies that it can connect to a Renishaw Optical lens that works with the VP2 and VP1 optical probes.

DEA
Specifies that it connects with the DEA old style probe head including their rotary head.

LEITZ1
Specifies that it can connect a Leitz M5 threaded device.

LEITZ1ANALOG
Specifies that it can connect a Leitz M5 threaded device on an analog probe head.

FIXEDOPTICS
-

ZOOMOPTICS
-

TESAOVP
-

TKJ
Specifies that it is a Tesa Kinematic Joint.

SENMATION
Specifies that it is a Senmation joint (a joint to change sensors).

Example Code for a Tip

```
ITEM:TIP1.5BY11MM M2
begintip
ribcount 10
cylinder 0 0 0 0 0 -3 3
cone 0 0 -3 3 0 0 -7 0.65
cylinder 0 0 -7 0 0 -11 0.65
color 255 0 0
sphere 0 0 -11 1.5
hotspot 0 0 -11 0 0 1 1.5 1.5 ball
endtip
```

Example Code for a Probe Head

```
ITEM:PROBEPH9A ARM
color 0 0 0
comment THE FOLLOWING 12 LINES CREATE THE BOX OF THE PH9
line -30 -30 0 30 -30 0
line 30 -30 0 30 30 0
line 30 30 0 -30 30 0
line -30 30 0 -30 -30 0
line -30 -30 0 -30 -30 -60
line -30 -30 -60 30 -30 -60
line 30 -30 -60 30 30 -60
line 30 30 -60 -30 30 -60
line -30 30 -60 -30 -30 -60
line 30 -30 0 30 -30 -60
line 30 30 0 30 30 -60
line -30 30 0 -30 30 -60
comment THE FOLLOWING CREATES THE LED
ring 0 -30 -30 0 1 0 20 4
ribcount 10
color 255 0 0
sphere 0 -30 -30 4
comment THE FOLLOWING STARTS THE ROTATING SPHERE
autojoint 0 0 -77 0 0 1 0 7.5 -180 180 b angle
color 0 0 0
ribcount 10
sphere 0 0 0 50
autojoint 0 0 0 -1 0 0 0 7.5 0 105 a angle
ribcount 10
```

```
color 0 0 0
cylinder -22 0 0 -38 0 0 25
connect -38 0 0 1 0 0 QC
```

Analog Probe Scanning Parameters

You can download a .pdf document from the link below to review analog probe scanning parameters for the following probes:

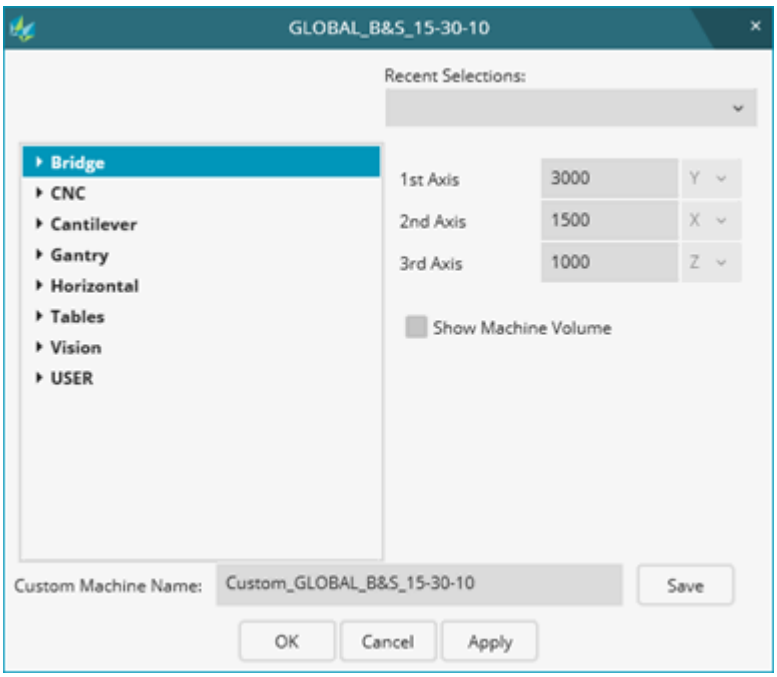
- SP600
- SP80
- SP25
- LSP-X3c/t
- LSP-X5
- LSP-X1s/c/h
- LSP-X1m

To get the file:

1. In your Internet browser, access this site: https://downloads.ms.hexagonmi.com/docs/FilesInHelps
2. Download the file "Scanning Parameters - H006908 PRB Params.pdf".

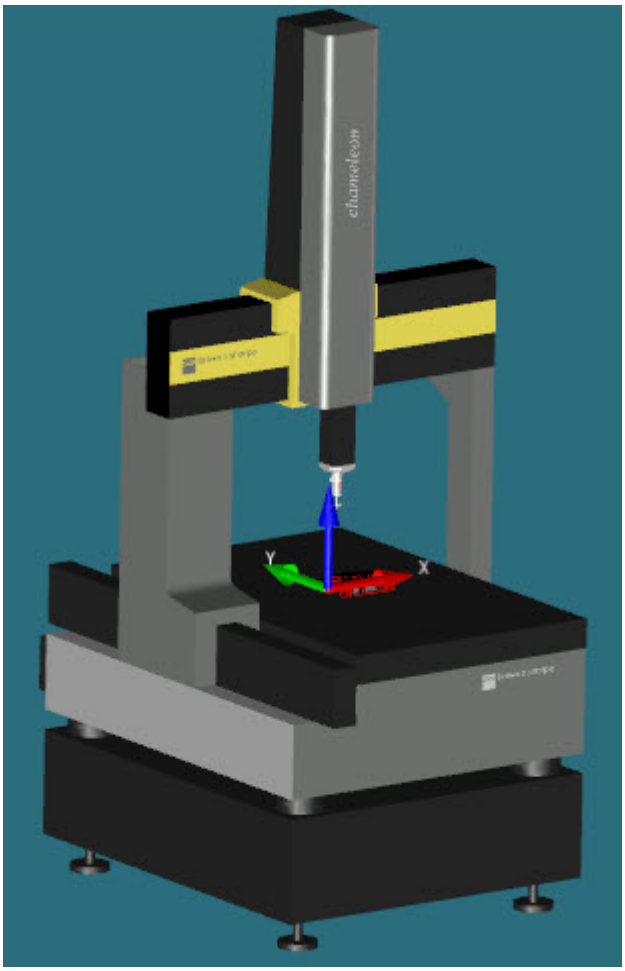
Defining Machines

If you use PC-DMIS in offline mode, the **Insert | Hardware Definition | Machine** menu option displays the **Load Virtual Machine** dialog box.



Load Virtual Machine Dialog Box

With this dialog box, you can select or create a virtual measuring machine or virtual table that is displayed and animated in the Graphic Display window along with the probe and part model.

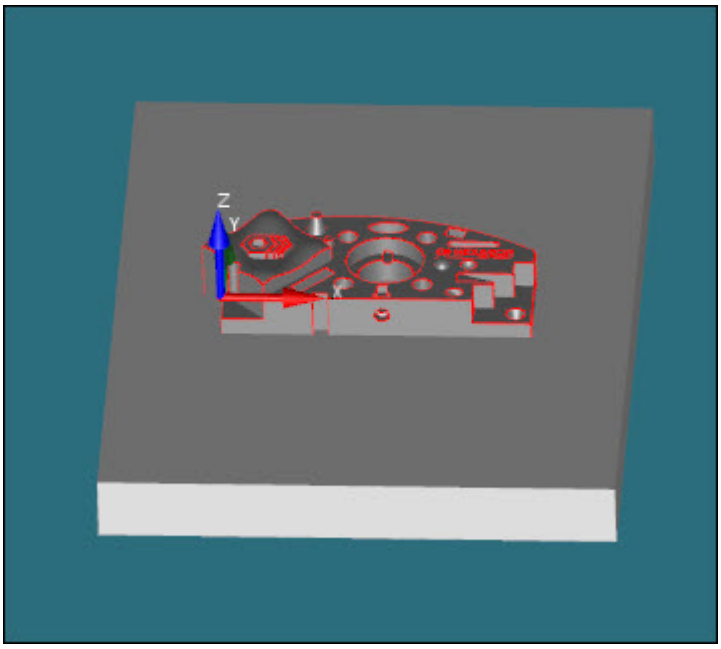


A sample virtual machine

If you import both a part model and a machine model into the Graphic Display window, and you don't have any outstanding manual machine offsets or rotations, PC-DMIS automatically positions the part model to the machine model's table.

If you have manual x,y,z, offsets or rotations from the **Part/Machine** tab of the **Setup Options** dialog box, you need to use that tab's **Auto Position** button to position the part model to the machine table. For more information on the **Auto Position** button, see the "Part Setup area" topic in the "Setting Your Preferences: Introduction" chapter.

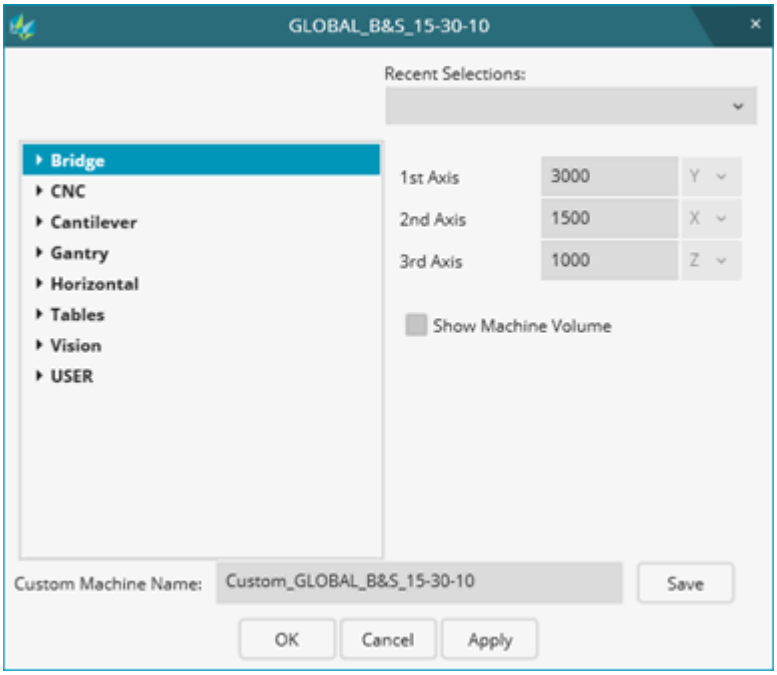
You may not want to show a full virtual machine and instead may only want to display a virtual table (or CMM baseplate). You can do this by choosing an option from the **Tables** category from the list of machines.



A sample virtual table

The virtual table comes from a list of tables defined in the tablesonly.dat file. You can modify the .dat file to contain tables that match what you need.

About the Load Virtual Machine Dialog Box



Load Virtual Machine Dialog Box

This dialog box is available only if you use PC-DMIS in offline mode.

List of machines - The expandable list on the left of the dialog box, shows the major categories of machines as well as virtual tables (in the **Tables** category).

When you expand a list, it shows the various machine brands. When you expand a brand, you can see the major machines available on the market for that brand. When you select a machine from this list, a preview of the machine appears in the Graphic Display window. PC-DMIS also creates an automatic custom name in the **Custom Machine Name** box for the machine in case you want to modify the machine's limits.

Recent Selections - This list contains the most recent (last five) machines that you selected with **OK** or **Apply**. This list remains across later versions of PC-DMIS. You can remove them from this list by resetting the product with the **Reset Product** button in the PC-DMIS Settings Editor.

1st, **2nd**, and **3rd** Axis - These boxes define the travel limits of the virtual machine. For more information, see "Additional Information on the Machine's Axes" below.

Show Machine Volume - This check box shows the machine's current volume as a green box. The green box appears when you click **OK** or **Apply**.

Custom Machine Name - This shows an automatic name you can use whenever you preview a machine. The name is the machine's name with a "Custom_" prefix. You can modify this to something different if needed.

Save - This saves the custom machine in a **USER** category of the list of machines. Custom machines are saved in the usermachine.dat file.

OK - This accepts the selected machine and any modifications and closes the dialog box. It inserts a `LOADMACHINE` command into the Edit window.

Apply - This does the same as **OK** except it does not close the dialog box.

Cancel - This closes the dialog box without accepting any changes.

Selecting or Creating a New Animated Machine for Display

If you use PC-DMIS in offline mode, you can use the **Load Virtual Machine** dialog box (**Insert | Hardware Definition | Machine**) to select or create a virtual machine or table to use in PC-DMIS's Graphic Display window.

Selecting an Existing Machine or Table

The **Load Virtual Machine** dialog box contains a list of machine and table models that you can add into the Graphic Display window. These are organized by type in an expandable / collapsible **Name** list.

To add a virtual machine:

1. From the list of machines, expand a category type to see the brands within that category.
2. Expand the desired brand name.
3. Select a machine to see a preview of the machine in the Graphic Display window.
4. Click **Apply** or **OK** to accept a machine and insert a `LOADMACHINE` command into the Edit window. This command also shows the machine's model into the Graphic Display window.

To add only a virtual table:

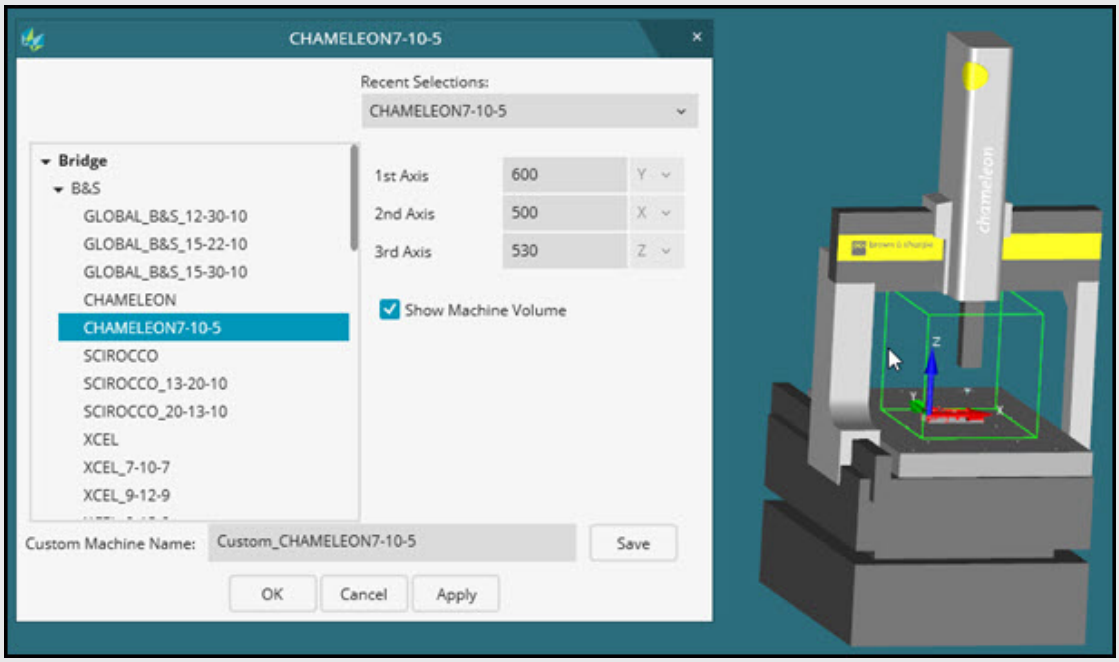
1. From the list of machines, expand the **Table** category.
2. Expand the **Size** list.
3. Choose a table to see a preview of the table in the Graphic Display window.
4. Click **Apply** or **OK** to accept a table and insert a `LOADMACHINE` command into the Edit window. This command also shows the table into the Graphic Display window.

Modifying and Saving a Custom Machine

You can change a machine's limits on the fly for any machine and store it as a custom machine.

1. From the list of machines, select an existing machine. PC-DMIS automatically shows a preview of the current machine. PC-DMIS also automatically creates a temporary custom machine name for you in the **Custom Machine Name** box. The custom name is the machine's name preceded by a "Custom_" prefix.
2. If you don't like the automatic name, type a new name for the machine. If you use spaces, the software converts them to underscore characters.
    - Specify the 1st axis by selecting X, Y, or Z from the **1st axis** list. Type the length of the machine's base in the **1st axis** box.
    - Specify the 2nd axis by selecting X, Y, or Z from the **2nd axis** list. Type the width of the machine's base in the **2nd axis** box.
    - Specify the 3rd axis by selecting X, Y, or Z from the **3rd axis** list. Type the height of the machine in the **3rd axis** box.
3. Click **Apply** to insert a `LOADMACHINE` command and a machine model in the Graphic Display window with those values. The "usermachine.dat" file is not actually modified until you click **Apply** or **OK**. If you click **Cancel** all changes are ignored.
4. If you want to save the customized machine, click **Save** to add the saved machine into the dialog box. PC-DMIS adds the machine under the **USER** category.

For example, suppose that you have a machine that's similar to another machine, but it has a smaller volume. You don't have to recreate the entire machine. Instead, you can modify the axis values of the similar machine to reflect your machine's volume for the current measurement routine. When you click **Apply**, PC-DMIS applies these changes internally to the `LOADMACHINE` command.



A Chameleon machine with a modified, smaller volume

You cannot edit the `LOADMACHINE` command in the Edit window. To make changes to the machine, press F9 on the **Load Virtual Machine** dialog box and make your changes there.

Additional Information on the Machine's Axes

The 1st axis defines the length of the machine's base in the direction that the bridge moves (for bridge machines), or the length of the base in the direction that the vertical carriage moves (for horizontal arm machines). This is usually the X axis for most horizontal arm machines, and either the X or the Y axis for most vertical arm machines.

The 2nd axis defines the width of the base in the direction that the carriage moves (for a bridge machine) or the direction the arm moves (for a horizontal arm machine). This is the vertical Z axis for most horizontal arm machines and either the X or the Y axis for most

vertical arm machines. This differs from the machine volume. The volume defines the distance the arm can reach. Most arm machines have a volume just over 50% of the width.

The 3rd axis defines the height of the machine from the top of the base to the bottom side of the bridge for a bridge machine, or the extent of the vertical axis for a horizontal arm machine. This is the vertical Z axis for most bridge machines and the Y axis for most horizontal arm machines. At this time only the following axis configurations are allowed:

- Generic Bridge with Y, X, Z or X, Y, Z
- Generic Arm with X, Y, Z

Additional Information for Creating Multiple Arm Machines

If you use PC-DMIS in offline mode, you can also create and place a simulated multiple-arm machine into the Graphic Display window.

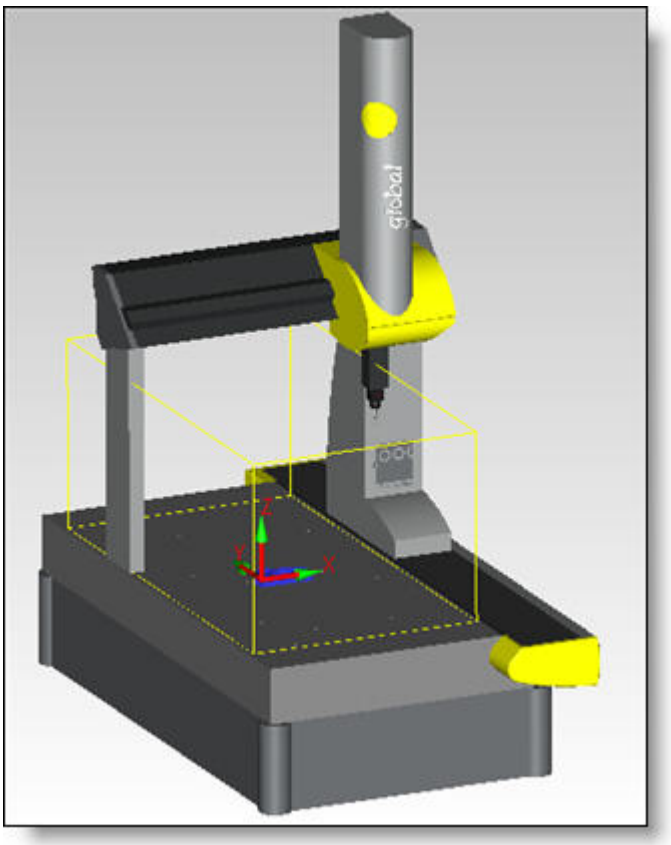
1. Follow PC-DMIS multiple-arm setup procedures.
2. Enter multiple arm mode. The **Active Arms** toolbar appears. This enables the **Dual Horizontal Arm** option from the **Machine Type** area.
3. Create a multiple arm measurement routine.
4. Follow the "Selecting or Creating a New Animated Machine for Display" procedure (make sure you select the **Dual Horizontal Arm** option button from the **Machine Type** area).
5. The Graphic Display window should now show a dual arm machine when finished.

Displaying Existing Animated Machines in the Graphic Display Window

If you use PC-DMIS in offline mode, you can display a representation of an animated measuring machine in the Graphic Display window. To do this, access the **Machine** dialog box (**Insert | Hardware Definitions | Machine**), and select a machine from the **Name** list.

PC-DMIS automatically displays the selected machine in the Graphic Display window along with the part's CAD model.

To display the machine's volume, click the **Show Machine Volume** check box, and click **Apply**. This draws a yellow cube on the machine's tabletop that represents the range of motion the probe can move to using that machine (see the example graphic below).



Sample machine and machine volume shown in the Graphic Display window

Click **OK** to accept the display of the machine.

PC-DMIS also inserts a `LOADMACHINE/NAME` command into the Edit window, where NAME is the machine's name.

If you import both a part model and a machine model into the Graphic Display window, and you don't have any outstanding manual machine offsets or rotations, PC-DMIS automatically positions the part model to the machine model's table.

If you have manual x,y,z, offsets or rotations from the **Part/Machine** tab of the **Setup Options** dialog box, you need to use that tab's **Auto Position** button to position the part model to the machine table. For more information on the **Auto Position** button, see the "Part Setup area" topic in the "Setting Your Preferences: Introduction" chapter.

Creating a Relationship between the Machine and the Part

You can create a relationship between the machine model and the CAD model by changing the XYZ offsets and rotations to establish the proper orientation between the two models. You can do this by using the **Quick Fixture Mode** to change the part's orientation

or by using the **Part/Machine** tab of the **Setup Options** dialog box. Or you can use a combination of the two approaches.

Using the Quick Fixture Mode

1. From the **Graphic Modes** toolbar, click **Quick Fixture Mode** ( ) to enter Quick Fixture Mode and display the **Quick Fixture** toolbar.
2. Use your mouse and the **Quick Fixture** toolbar to properly orient your part.
    - From the drop-down icon on the **Quick Fixture** toolbar, select **Level** ( ) to level your part to the machine model's table surface.
    - From the drop-down icon on the **Quick Fixture** toolbar, select **Drop** ( ) to set your part onto the machine model's table surface.
    - Right-click on the part model to rotate and move your part as needed.

When you then switch to a different mode, the part model remains fixed at its final position.

For more information on how to use this mode, see "Using Quick Fixture Mode" in the "Defining Hardware" chapter.

For information on the **Quick Fixture** toolbar, see "Quick Fixture Toolbar" in the "Using Toolbars: Introduction" chapter.

Using the Part/Machine Tab

1. Press F5 to access the **Setup Options** dialog box.
2. Select the **Part/Machine** tab.
3. In the **Part Setup** area, click the **Auto Position** button to have PC-DMIS automatically position your part model on the machine model's table surface.
4. If you want more direct control over placement of your part model, modify the items directly in the **Part Setup** area.

See "Setup Options Mode: Part / Machine tab" topic in the "Setting Your Preferences" chapter.

Removing the Animated Machine from the Graphic Display Window

1. Access the Edit window.
2. Place it in Summary Mode or Command Mode.

3. Delete the entire `LOADMACHINE` command. PC-DMIS removes the display of the machine.

Permanently Deleting an Animated Machine

1. Access the **Load Virtual Machine** dialog box (**Insert | Hardware Definition | Machine**).
2. Select the machine to delete from the **Machine Name** list.
3. Click the **Delete** button.
4. Click **Apply** or **OK**. PC-DMIS removes the machine from the **Machine Name** list and removes the display of the machine. This also deletes the machine from the "usermachine.dat" file.

The "usermachine.dat" file is not actually modified until you click **Apply** or **OK**. If you click **Cancel** all changes are ignored.

Editing the usermachine.dat Data File

The machines listed in the dialog box come from two .dat files:

- The file "machine.dat" contains the default machine descriptions supplied by the PC-DMIS developers.
- The file "usermachine.dat" contains the machine descriptions of any machines the user has created and saved.

If there is no "machine.dat" file, the list of machines only displays the **No Machine** item.

You can use the information in machine.dat as a model to manually create your own machines in the usermachine.dat file. For information on where these files are stored, see "Understanding Data Files" in the "Setting Your Preferences" chapter.

Definitions of the machine.dat and usermachine.dat Files

The machine.dat file controls the graphical and numerical representation of the default machines available in PC-DMIS. If you need a custom machine or machine system within PC-DMIS, you need to define it in the usermachine.dat file. The usermachine.dat file follows the same format, but it holds custom machines.

You need to strictly follow a specific format to define the machine.

This example describes the format and rules of the format with a list of the commands and their definitions.

```
ITEM:2000_20-12-18_ARM1 ARM
```

In the above code snippet, the following rules apply:

1. The first set of characters following the colon "**ITEM:**" is the name that appears in the **Load Virtual Machine** dialog box when you construct a machine setup (see the example dialog box image below). This is a unique name of your choice to identify the machine configuration.

   You can find details in the **Format of the Machine or Machine System ITEM** section below.

2. The second set of characters, "**ARM**" in the above example, defines the type of machine connection.

The following information lists and defines the machine builder commands:

```
ITEM:<machine name>
```

Specifies the machine name displayed in the **Load Virtual Machine** dialog box in PC-DMIS.

```
ITEM <connection type>
```

Defines the type of connection. The `ARM` connection type is used in the above example. This doesn't mean much in the case of a machine start, however.

```
cadgeom n1 n2 n3 n4 n5 n6 n7 n8 n9 n10 n11 n12 n13
<draw_file_name.draw>
```

Defines the position and rotation of the layers within the DRAW file:

```
n1 n2 n3
```

The X Y Z coordinate location from the origin of 0,0,0.

```
n4 n5 n6
```

The I J K vector for the X+ side of the layer.

```
n7 n8 n9
```

The I J K vector for the Y+ side of the layer.

```
n10 n11 n12
```

The I J K vector for the Z+ side of the layer.

```
n13
```

The layer to display. All previous "n" values are applied to this layer.

```
draw_file_name.draw
```

The DRAW file that the software uses to get the layer data. This file is located in the "models\cmm" sub-folder in the PC-DMIS install directory.

```
tablecenter n1 n2 n3 n4 n5 n6
```

Defines the position of the machine table within the DRAW file:

```
n1 n2 n3
```

The distance of the center (XYZ) of the plate from the origin of the DRAW file.

```
n4 n5 n6
```

The I J K vector of the plate.

```
<joint type> n1 n2 n3 n4 n5 n6 n7 n8 n9 n10 <joint name>
```

Defines the type of joint used within the DRAW file:

```
<joint type>
```

The type of joint used. The options are "autojoint", "linearjoint", and "manualjoint".

```
n1 n2 n3
```

The X Y Z coordinate location from the origin 0,0,0.

```
n4 n5 n6
```

The I J K vector.

```
n7 n8
```

The angular offset value (`n7`) and the increment value (`n8`). These are mainly used for autojoint configurations and are typically set to 0 0 in most other configurations.

```
autojoint 244.147 -151.141 -586 0 0 1 0 0.01 -360 360 w axis
```

The example shows that the configuration starts at 0 (`n7`) and rotates in 0.01 (`n8`) degree increments.

```
n9 n10
```

The minus and plus motion for the defined axis. The first number (`n9`) is always a negative value or zero while the second number (`n10`) is always a positive number or zero.

```
<joint name>
```

A unique user-defined name for the joint for each ITEM definition. You can use space characters in the joint name.

```
connect n1 n2 n3 n4 n5 n6 <probe connection type> <joint type>
```

The machine position in relation to the last linear joint at which the probe connects:

```
n1 n2 n3
```

The X Y Z coordinate location of the probe connection from the last linear joint position.

```
n4 n5 n6
```

The I J K vector for the probe connection.

```
<probe connection type>
```

The machine-to-probe connection type. This is typically defined as `ARM` .

`connectprev <command command_args>`
The number of joints to back up to in a machine configuration. The limit is three.

`connectprev` - Back up to the first previous joint.

`connectprevprev` - Back up to the second previous joint.

`connectprevprevprev` - Back up to the third previous joint.

`<command command_args>` - The command related to the joint type and its arguments.

```
ITEM:GLOBAL_DEA_7-7-5_ROTAB ARM
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 1 global_DEA_7-7-5.draw
autojoint 0 0 0 0 1 0 0 0 -700 0 y axis
cadgeom 0 0 0 0 1 0 1 0 0 0 1 0 2 global_DEA_7-7-5.draw
autojoint 0 0 0 -1 0 0 0 0 0 700 x axis
cadgeom 0 0 0 0 1 0 0 0 1 1 0 0 3 global_DEA_7-7-5.draw
autojoint 0 0 0 0 1 0 0 0 -500 0 z axis
cadgeom 0 0 0 0 -1 0 1 0 0 0 0 1 4 global_DEA_7-7-5.draw
connect 0 0 0 0 0 1 ARM
connectprevprevprev autojoint 244.147 -151.141 -586 0 0 1 0
0.01 -360 360 w axis
cylinder 0 0 0 0 0 25 500
tablecenter 0.0 0.0 25.0 0 0 1
```

Format of the Machine or Machine System ITEM

Each new machine configuration defined in PC-DMIS is added to the machine.dat file. It begins with the "**ITEM:**" command. This command contains a distinct and descriptive name and a connection identifier. This identifier specifies the type of object it can be connected to as described above. Below, you can see the available connection identifiers with a brief description.

To define a probe configuration, you must work with a top-down approach. This means that your initial starting point is x=0, y=0, z=0, and everything is drawn from that point down. Consequently, all z values should have a negative value. Also, for most items, the 0, 0, 0 location is the center of the item. This makes it possible to have both positive and negative numbers in both the X and Y directions. Positive X always moves from left to right initially, and positive Y always moves from front to back as if you were looking FROM the positive Z direction.

The only command that defines an active configuration is "**connect**". You must have this command in your configuration. The "**connect**" command defines any location where another item can be added to the current item. Each connection must include a Connection Type identifier to define the type of connection the machine can take. This is typically an Arm connection type.

The commands that can change the coordinate system of the configuration are "**linearjoint**" and "**autojoint**". Take special care when you use either of these commands because each command transfers the origin of the configuration to the center of the joint. Keep in mind that the first "joint" command changes the axes of the coordinate system. What was the -Z direction now becomes the +X direction, and what was the +X direction now becomes the +Z direction. Y remains the same. These commands are usually used in pairs. The joints control the motion of the machine's components.

Connection Identifier Listing

ARM - This connection type specifies that the connection occurs directly to the machine arm. Although there are many possible connection types, PC-DMIS uses the ARM connection type in almost all machine configurations.

This is a typical Machine Code example:

```
ITEM:2000_20-12-18_ARM1 ARM
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 1 2000_20-12-18.draw
tablecenter 1345 1200 -2112 0 0 1
linearjoint 0 0 0 1 0 0 0 0 0 2690 x axis
cadgeom 0 0 0 1 0 0 0 0 1 1 0 0 2 2000_20-12-18.draw
linearjoint 0 0 0 -1 0 0 0 0 -1800 0 z axis
cadgeom 0 0 0 1 0 0 1 0 0 0 0 1 3 2000_20-12-18.draw
linearjoint 0 0 0 0 1 0 0 0 0 1200 y axis
cadgeom 0 0 0 0 1 0 1 0 0 0 1 0 4 2000_20-12-18.draw
connect 0 0 0 0 0 -1 ARM
```

Editing the tablesonly.dat File

The default tables are created and stored in a text file named tablesonly.dat. You can use the information in that file to create your own tables. For information on where this file is stored, see "Understanding Data Files" in the "Setting Your Preferences" chapter.

To edit tablesonly.dat, open it in a text editor and modify the data.

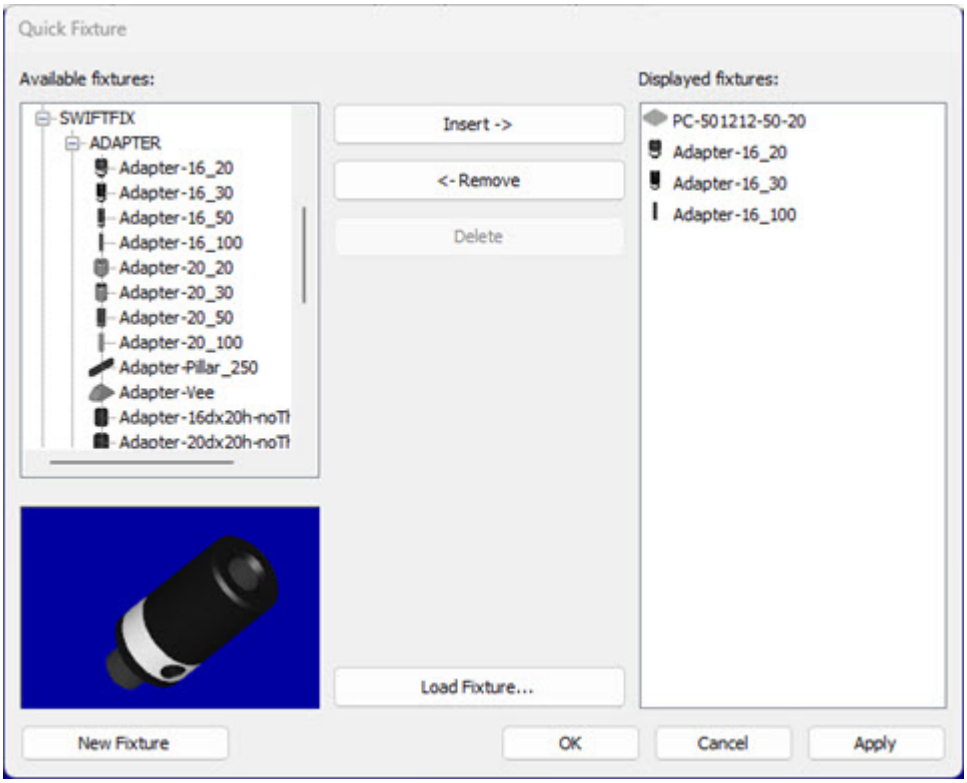
This example shows three tables. The first table has a width of 300 mm, a length of 200 mm, and a height of 50 mm.

```
[Tables]

Count=3

1=300,200,50

2=400,500,50

3=500,500,75
```

If you want to just have one table that matches your actual machine's dimensions, you can set `Count=1` , delete all but the first table, and then type your table's dimensions.

Inserting Quick Fixtures

Select **Insert | Hardware Definition | Quick Fixture** to display the **Quick Fixture** dialog box. You can use this dialog box to insert predefined or custom fixtures into the Graphic Display window.



Quick Fixture Dialog Box

This dialog box contains these options:

Available fixtures - Displays the fixtures that you can import into the Graphic Display window. These fixtures are stored in the "Models\QuickFix\" subfolder in the PC-DMIS installation directory. When you select a fixture, PC-DMIS displays a preview of the fixture in the preview pane below the list.

Displayed fixtures - Displays fixtures currently shown in the Graphic Display window. To reposition fixtures in the Graphic Display window, see "Using Quick Fixture Mode".

Insert - Moves the selected fixture from the **Available fixtures** list to the **Displayed fixtures** list, allowing the fixture to appear in the Graphic Display window.

Remove - Removes the selected fixture from the **Displayed fixtures** list.

Delete - After you add a custom fixture using the **New Fixture** option, a **User** section appears in the **Available fixtures** list. The **Delete** button is available when you select one of these custom fixtures from the **User** section. Click **Delete** to permanently remove the selected fixture from the list and the userquickfix.dat file.

Load Fixture - Loads a previously saved quick fixture (as a .qfxt extension) into the Graphic Display window. In the **Open** dialog box, select the fixture to load and click **Open**. The **Quick Fixture** dialog box displays the loaded fixture in the **Displayed fixtures** area. You can load only one .qfxt file for each measurement routine.

New fixture - Opens the **Open** dialog box, where you can browse to and select a fixture file. Click **Open** to add the selected file to the **Available fixtures** list. You can add any supported fixture file.

OK - Applies the current changes and closes the dialog box.

Apply - Applies the current changes without closing the dialog box.

When you select one or more Quick Fixture components in the **Quick Fixture** dialog box, the selected components are highlighted in red in the Graphic Display window.

In PC-DMIS Offline mode, if an animated machine has been added to the measurement routine, PC-DMIS inserts the new Quick Fixture components at the -X, -Y, -Z corner of the machine's work volume. Otherwise, PC-DMIS adds them at the origin.

Inserting and Removing Quick Fixtures

To insert a quick fixture into the Graphic Display window, follow these steps:

1. Select **Insert | Hardware Definition | Quick Fixture** to open the **Quick Fixture** dialog box.
2. Select the fixture that you want to add from the **Available fixtures** list. PC-DMIS displays a preview of the fixture in the preview pane.
3. Click **Insert**. The fixture name appears in the **Displayed fixtures** list, and the fixture model appears in the Graphic Display window.

   If an animated machine has been added to the measurement routine, PC-DMIS inserts the new Quick Fixture components at the -X, -Y, -Z corner of the machine work volume. Otherwise, PC-DMIS inserts them at the origin.

4. Click **OK**.

- If the machine table is empty, importing a base plate or CAD model automatically places it at the center of the table.
- If you insert a base plate before importing a CAD model, PC-DMIS places the CAD model at the center of the base plate.
- When you insert multiple fixtures, make sure they do not overlap.

To remove a quick fixture from the Graphic Display window, follow these steps:

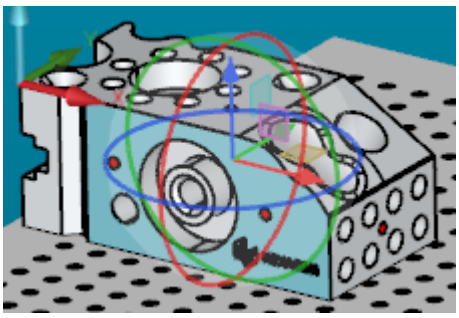
1. Select **Insert | Hardware Definition | Quick Fixture** to open the **Quick Fixture** dialog box.
2. Select one or more fixtures from the **Displayed fixtures** list. PC-DMIS highlights the selected fixtures in red in the Graphic Display window.
3. Click **Remove**. PC-DMIS removes the selected fixtures from the **Displayed fixtures** list and the Graphic Display window.
4. Click **OK**.

To reposition quick fixtures, see "Using Quick Fixture Mode".

Using Quick Fixture Mode

In **Quick Fixture Mode**, you can move, rotate, align, and attach fixture and CAD models in the Graphic Display window by using the Quick Fixture gizmo. To enter this mode, click the **Quick Fixture Mode** icon ( ) on the **Graphic Modes** toolbar.

When you select a fixture or CAD model, the Quick Fixture gizmo appears at the center of the selected object. The gizmo contains three axis arrows, three plane handles, and three rotation rings that you can use to move and rotate the selected object.



Quick Fixture gizmo

When you double-click a gizmo handle, the operation is based on the currently highlighted element.

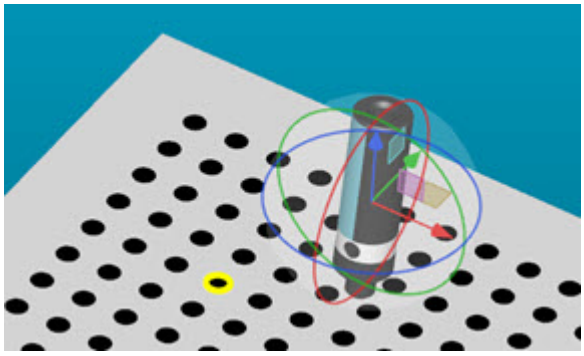
- If you highlight a rotation ring, the corresponding rotation operation (Level or Square) takes priority, even if the pointer is over an underlying axis.
- If you highlight a handle, the corresponding plane operation takes priority, even if it overlaps a single-axis handle.
- PC-DMIS uses the same hover-priority behavior for drag and double-click operations.

Attaching Fixtures

You can attach a fixture to a circle feature on another fixture or on the CAD model.

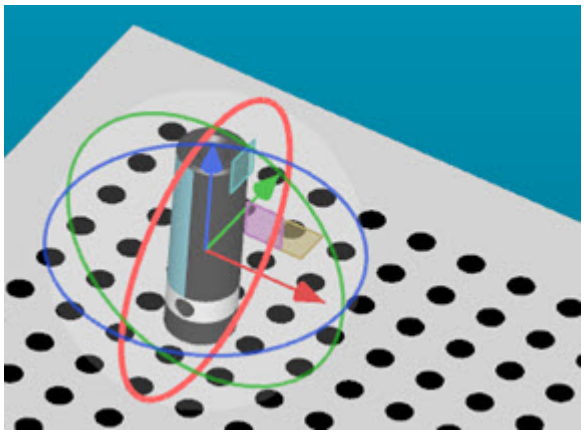
To attach a fixture:

1. Select a fixture.
2. Press and hold **Shift**.
3. Hover over the target circle until it is highlighted in yellow.



Example showing the selected fixture and highlighted circle.

4. Click the highlighted circle.



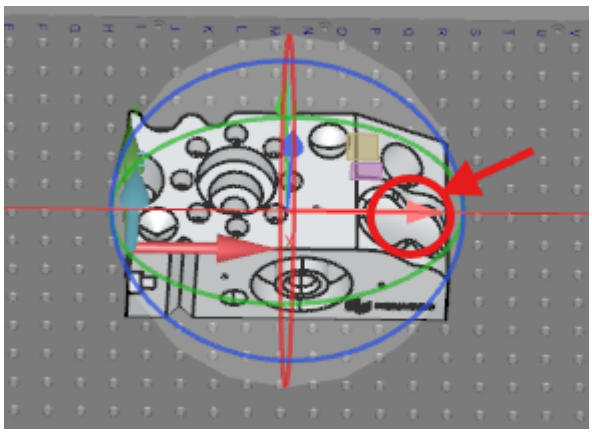
Example of a repositioned fixture attached to a circle feature.

PC-DMIS inserts the selected fixture into the hole.

Moving Objects

Move Along an Axis

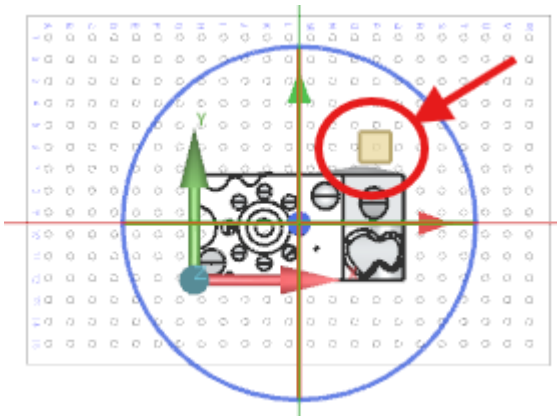
1. Select the fixture or CAD model.
2. Hover over an axis arrow until it is highlighted.



3. Click and drag the highlighted arrow to move the selected object along that axis.

Move Within a Plane

1. Select the fixture or CAD model.
2. Hover over a plane handle until it is highlighted.

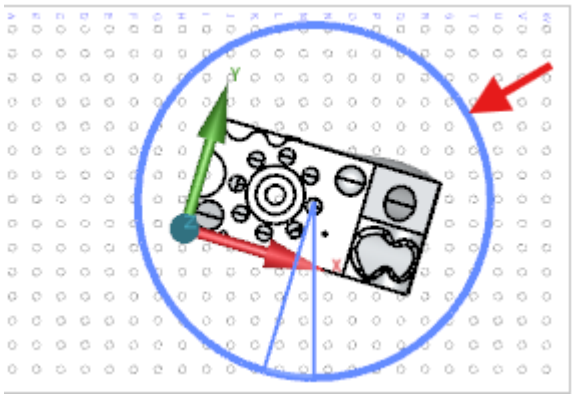


3. Click and drag the highlighted plane handle to move the selected object within the corresponding plane.

Rotating Objects

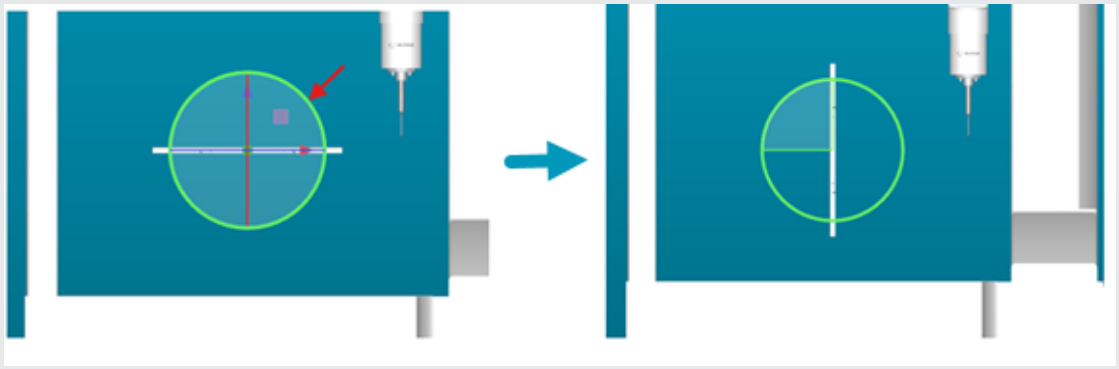
Rotate About an Axis

1. Select the fixture or CAD model.
2. Hover over a rotation ring until it is highlighted.



3. Click and drag the highlighted ring to rotate the selected object about the corresponding axis.

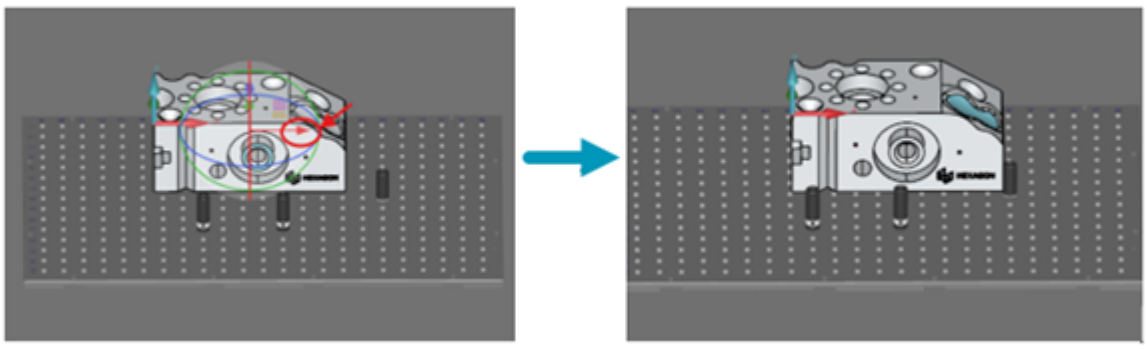
While dragging a rotation ring, press and hold **Shift** to snap the rotation to the nearest 90-degree position.



Dropping Objects

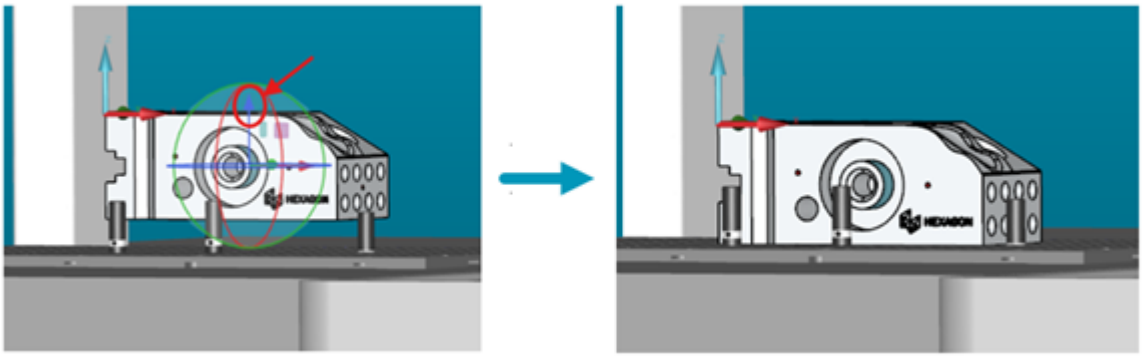
Drop Along an Axis in the Positive Direction

1. Select the fixture or CAD model.
2. Double-click an axis arrow to move the selected object in the positive direction until it contacts the nearest object.



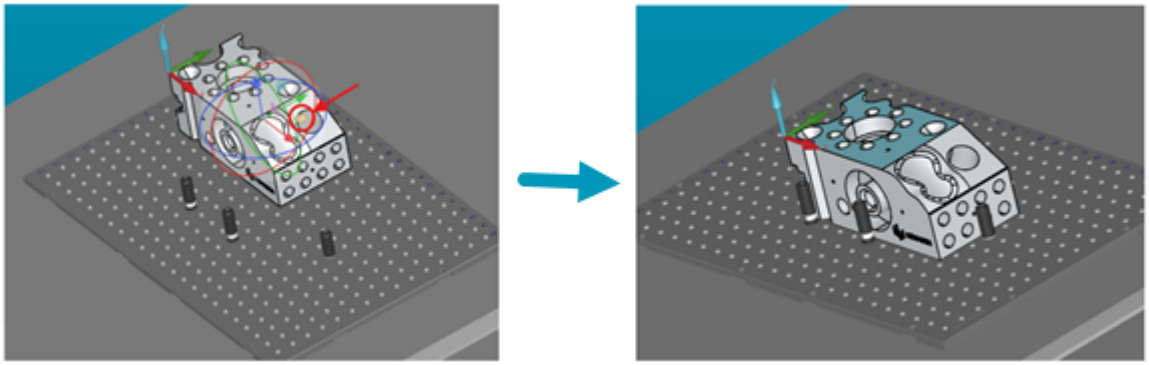
Drop Along an Axis in the Negative Direction

1. Select the fixture or CAD model.
2. Press and hold **Ctrl**, and then double-click an axis arrow to move the selected object in the negative direction until it contacts the nearest object.



Corner Drop

1. Select the fixture or CAD model.
2. Double-click a plane handle to move the selected object simultaneously along all three axes until it contacts the nearest object on each axis.



Corner drop uses the following collision detection logic:

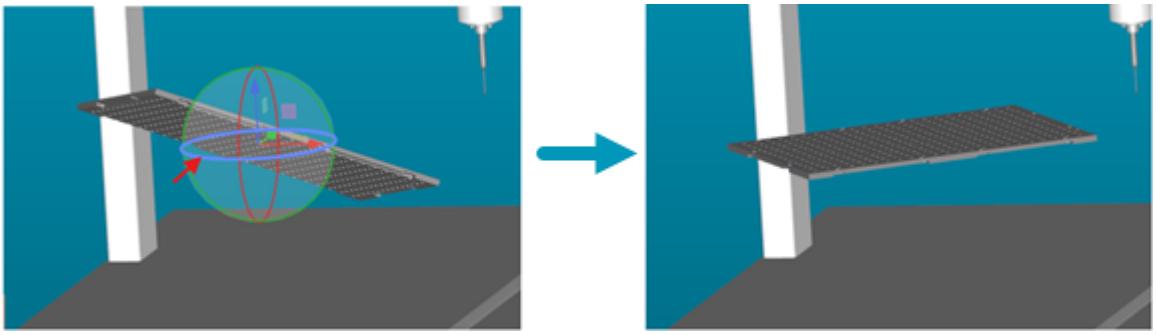
- For each axis, PC-DMIS first checks for collisions in both the positive and negative directions without moving the object.
- If collisions are detected in both directions, PC-DMIS moves the object toward the nearest collision.
- If a collision is detected in only one direction, PC-DMIS moves the object in that direction.
- If no collision is detected, PC-DMIS does not move the object along that axis.
- PC-DMIS processes each axis independently and sequentially.

Aligning Objects

Level

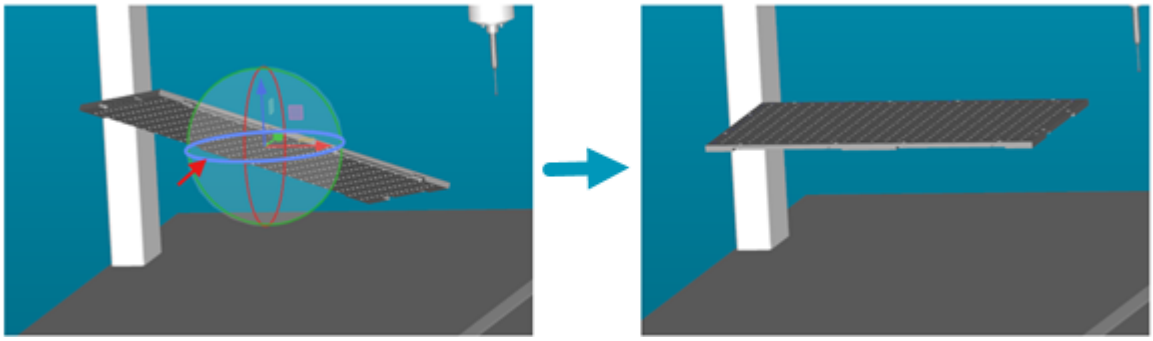
1. Select the fixture or CAD model.

2. Double-click a rotation ring to align the selected object's surface normal with the machine coordinate system.



Square

1. Select the fixture or CAD model.
2. Press and hold **Ctrl**, and then double-click a rotation ring to align the selected object's axes with the CAD coordinate system shown by the trihedron.



Adding Custom Fixtures

To add a custom fixture to the **Available fixtures** list:

1. Open the **Quick Fixture** dialog box (**Insert | Hardware Definition | Quick Fixture**).
2. Click **New Fixture**. In the **Open** dialog box, browse to and select a fixture model in a supported graphical format, and then click **Open**.
3. PC-DMIS adds the fixture to the **User** section of the **Available fixtures** list and is available for use in the current and future measurement routines.

Using Quick Fixtures Files

Using .DRAW Files

PC-DMIS stores quick fixture files with a .DRAW filename extension in the "Models\QuickFix\" sub-folder where you installed PC-DMIS.

In addition, PC-DMIS stores a definition of each quick fixture in a special data file (.dat filename extension) located in the directory where you installed PC-DMIS.

- If the quick fixture was originally shipped with PC-DMIS, it is stored in the *QuickFix.dat* file.
- If the quick fixture is a user-created fixture, its definition is stored in the *UserQuickFix.dat* file.

A typical quick fixture data file consists of two lines for each quick fixture, an **ITEM:** line and a **cadgeom** line. In a text editor, this file might look something like this:

```
ITEM:R20-501-SO RAYCO-STANDOFF
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 R20-501-SO.draw
ITEM:R20-5050-SO RAYCO-STANDOFF
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 R20-5050-SO.draw
ITEM:R20-7550-SO RAYCO-STANDOFF
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 R20-7550-SO.draw
ITEM:Adapter-16dx20h-noThread SWIFTFIX-ADAPTER
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 Adapter-16dx20h-noThread.draw
ITEM:Adapter-20dx20h-noThread SWIFTFIX-ADAPTER
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 Adapter-20dx20h-noThread.draw
ITEM:Adapter-16dx20h-withPin SWIFTFIX-ADAPTER
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 Adapter-16dx20h-withPin.draw
ITEM:Adapter-20dx20h-withPin SWIFTFIX-ADAPTER
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 Adapter-20dx20h-withPin.draw
ITEM:Adapter-Joint SWIFTFIX-ADAPTER
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 0 Adapter-Joint.draw
```

A sample QuickFix.Dat file.

To Use Custom Quick Fixture .DRAW Files from a Previous Installation

1. Close down PC-DMIS and open Windows Explorer.
2. Before you create new custom fixtures for your new installation, copy the .DRAW files from the "Models\QuickFix\" sub-folder of your previous installation, and paste them into the "Models\QuickFix\" sub-folder of your current PC-DMIS installation.
3. Copy the userquickfix.dat file from the directory of your previous installation, and paste it into the directory of your current PC-DMIS installation.
4. Restart PC-DMIS. The custom quick fixtures will appear in the **Quick Fixture** dialog box along with the fixtures supplied by your current PC-DMIS installation.

The .DRAW quick fixture files will not appear inside of the **Quick Fixture** dialog box unless the *userquickfix.dat* file contains the necessary definitions for the file. These definitions get added whenever you import your own custom IGES files as quick fixtures using the **Quick Fixture** dialog box. For information, see "Adding Custom Fixtures".

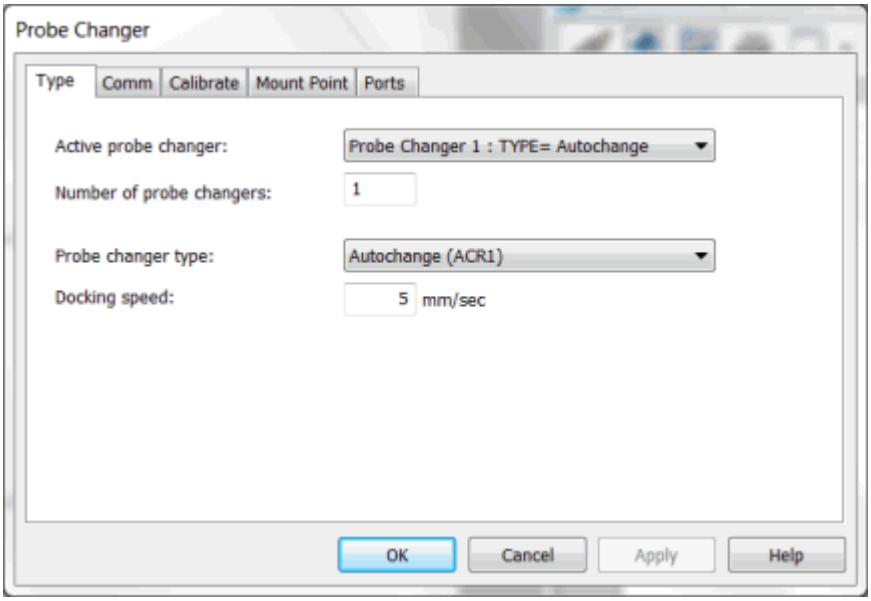
If you want to directly use the .DRAW files created by another user, but you don't want to overwrite your own quick fixtures by copying the other user's data file, you would need to use a text editor to merge the necessary **ITEM:** and **cadgeom** lines from the other user's data file into your data file.

Using .QFXT Files

You can also save and load fixture files as .QFXT files by using the **Save QuickFixture file** and **Load QuickFixture file** buttons on the **Quick Fixture** toolbar. For more information, see "Quick Fixture Toolbar".

Defining Probe Changers

Before you can begin to use your probe changer, you must select it in PC-DMIS, define its port locations, and set the options that you can use on it. To perform these tasks, select **Edit | Preferences | Probe Changer**. The **Probe Changer** dialog box opens:



Probe Changer Dialog Box

Initially, the **Probe Changer** dialog box displays only the **Type** tab. Some or all of the following tabs become available after you select your probe changer and click **Apply**:

Type - Use this tab to select and define the type of probe changer for your system.

Comm - Use this tab to define the serial port to use with the probe changer.

Calibrate - Use this tab to select the active tip to calibrate.

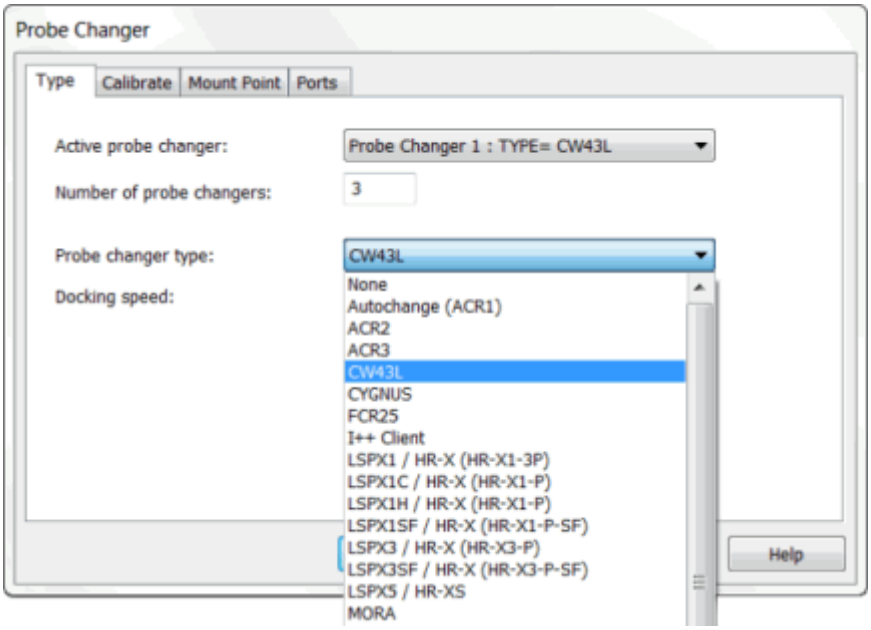
Mount Point - Use this tab to change the probe head wrist angle values and define a safe move location for the CMM to move to after each measurement cycle.

Ports - Use this tab to define the probe or stylus configuration for existing ports in the probe changer rack.

PC-DMIS supports various probe changer hardware, and terms are used by their respective manufacturers. "Slot", "port", and "garage" are used interchangeably. They all mean the location that holds the probe configuration.

Type Tab

Before you can begin the calibration process, you must select and define the type of probe changer for your system on the **Type** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Type Tab

Active probe changer

The **Active probe changer** list contains numbered items (Probe Changer 1, Probe Changer 2, Probe Changer 3, and other items) to which you can assign a probe changer and its values. The number of items that appears in the list depends on the value in the **Number of probe changers** box. Initially, PC-DMIS usually displays only Probe Changer 1. To add additional items to the list, increase the value in the **Number of probe changers** box.

PC-DMIS stores the probe changer type that you selected in the **Probe changer type** list, plus any other values, for the numbered item in the **Active probe changer** list.

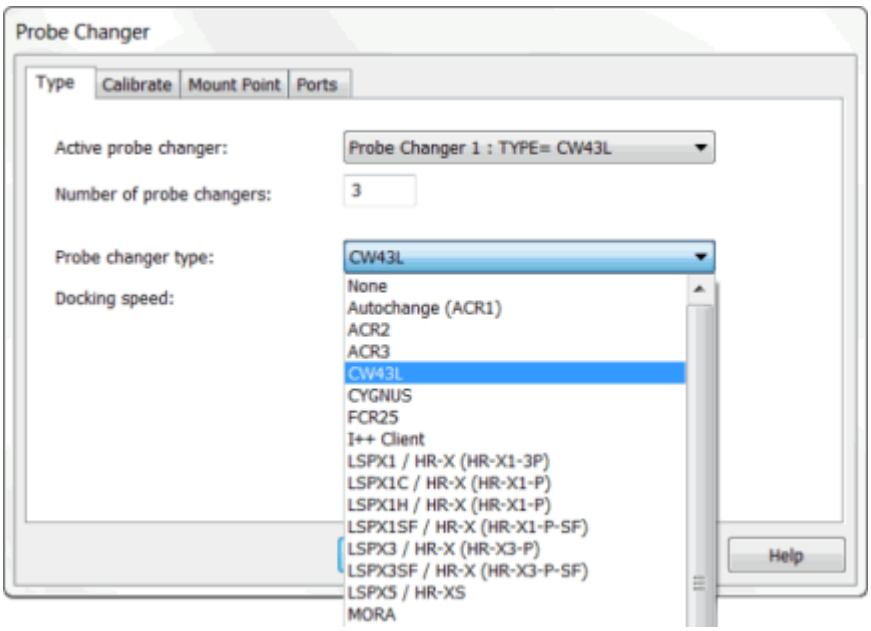
Number of probe changers

Type the number of different types of probe changers in the **Number of probe changers** box. The number that you enter becomes the number of available probe changers in the **Active probe changer** list.

If you type a smaller value in the **Number of probe changers** box, PC-DMIS removes all the probe changers after the probe changer in the new value position. If any of the probe changers to be removed have their type set, PC-DMIS prompts you to confirm their removal.

Probe changer type

The **Probe changer type** list contains the various probe changer types. For example:



CW43L Probe Changer Type

To select a probe changer:

1. Select the probe changer in the **Probe changer type** list.
2. Click **Apply** to make this probe changer active and load settings that apply to this probe changer.

Docking speed

You can change the docking speed to fine-tune the operation of the probe change cycle. To set the docking speed, type a new percentage value in the **Docking speed** box. The default value of 5% is appropriate for most machine configurations.

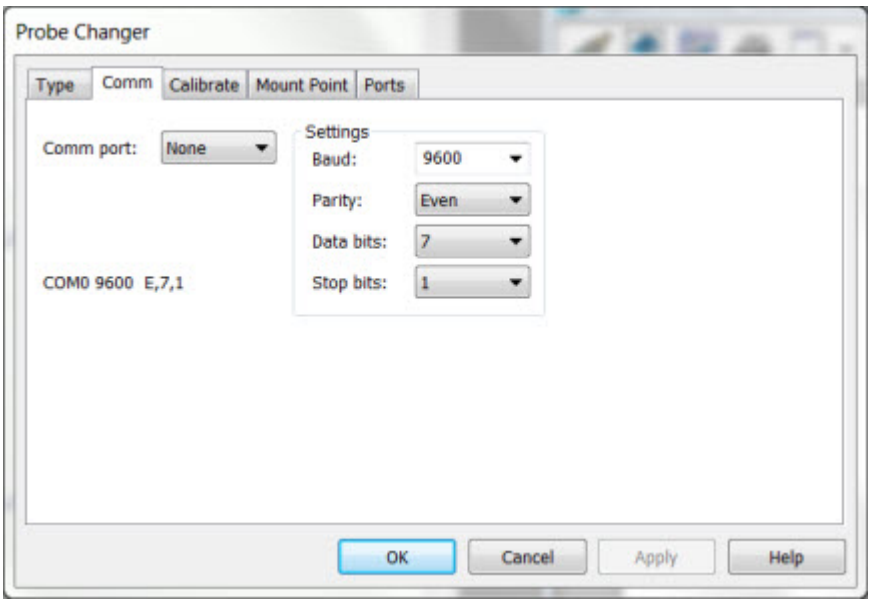
> The docking speed may not be available for some types of probe changers. The controller manages the probe change cycle when the **Docking speed** option is not available. For example, the controller manages the probe change cycle for the LSPX5, and other LSPX probe types. For these probe types, the controller controls the slow docking speed, and you cannot change it.

Apply button

To save your changes, click **Apply**.

Comm Tab

Some types of probe changers have a serial connection to the computer. Use the **Comm** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**) to choose your communications port (COM port) and modify your COM port settings. For information on your COM port settings, check the documentation that came with your computer hardware.

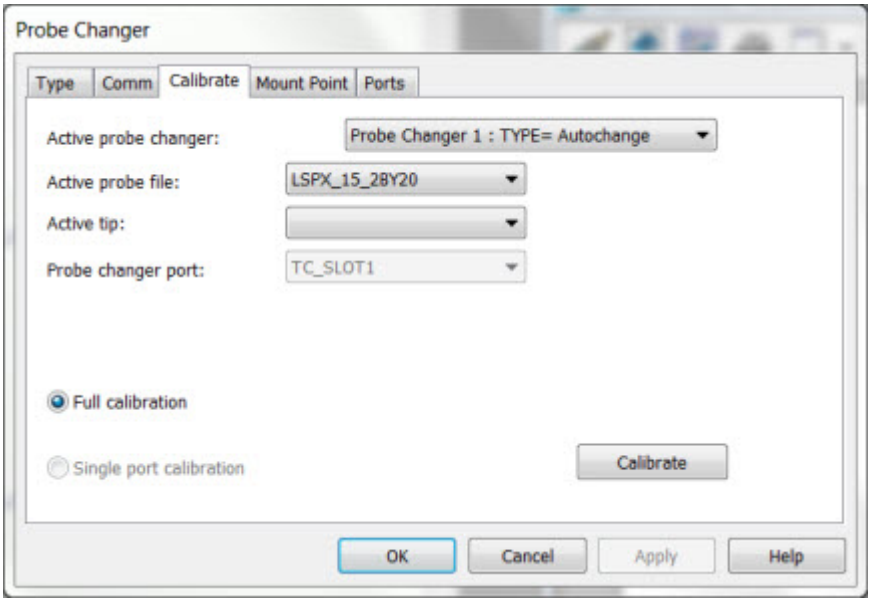


Probe Changer Dialog Box - Comm Tab

The **Comm** tab does appear if your probe changer does not have a serial connection.

Calibrate Tab

To open the **Calibrate** tab in the **Probe Changer** dialog box, select **Edit | Preferences | Probe Changer**.



Probe Changer Dialog Box - Calibrate Tab

Before you begin the calibration process, consider the following:

- You must choose the probe changer type from the **Type** tab.
- Before you start this process, verify that the stylus has been calibrated. For more information, see "Defining Probes" in the "Defining Hardware" chapter.

- The rack should be squared to one of the CMM's axes.

Generally, you should physically align probe changers along one of the machine’s axis. However, for the TP20, TP200, and SP600 Probe Changers, this isn't required. For these probe changers, the rack still needs to be level, but you can now rotate it so that the length of the rack is no longer aligned along a machine axis.

Active probe changer

From this list, select the probe changer to calibrate. For more information, see "Type Tab".

Active probe file

From this list, select a probe to use for the calibration process.

Active tip

From this list, choose a tip configuration for the probe that you selected.

Probe changer port

This list works in conjunction with the **Single port calibration** option. It enables you to choose a single port to calibrate. These dialog box items remain unavailable for selection until you define a probe changer that supports calibrating an individual port.

Full calibration

If you select the **Full Calibration** option, PC-DMIS measures the entire probe changer. This is the most often used method of calibration, and for some it is the only method that is available. It is recommended that the operator use the full calibration method.

Partial calibration

This option only calibrates a portion of the probe changer. This option only appears for the changer types that support it.

Single port calibration

Some probe changers, such as the ACR1, enable you to measure only a single port after a successful full calibration. The **Single port calibration** option only appears for the changer types that support it.

Calibrate Button

Before you can use your probe changer, you must define the changer's port locations through the appropriate probe changer calibration procedure. The following topics discuss the calibration processes for several probe changers. If you have a probe changer type that is different from the ones discussed, use the calibration process for the FCR25 Probe Changer as a guide. It should be similar enough for all supported types.

Calibrating the FCR25 Probe Changer

The calibration process for the Renishaw FCR25 Probe Changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the FCR25 Probe Changer.

Two inserts are used with the FCR25 rack to accommodate different probe types: PA25-SH and PA25-20.

PA25-SH and PA25-20 Inserts

For example, the following image of an FCR25 Probe Changer shows three ports: two with inserts and one without (probes are also shown). The left port shows a PA25-SH insert, and the right port shows a PA25-20 insert. The middle port does not have an insert.

FCR25 Probe Changer inserts and probe types

> The FCR25 Probe Changer must be mounted on the machine table parallel to either the X or Y axis for successful calibration. The FCR25 Probe Changer can be mounted to your table with the MRS rack or the three and six port stand-alone racks. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the FCR25 Probe Changer

To select the FCR25 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **FCR25**.
3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** list, select the item that represents the probe changer you are defining. If this is the first probe changer, it appears as **Probe Changer 1 : TYPE=None**.
5. In the **Number of probe changers** box, type the number of different types of probe changers.
6. In the **Docking speed** box, type a value. A value between 15-20% is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the location your probe body moves to when you use the probe changer to switch probe components.

Step 2 - Define the Mount Point

The mount point for the FCR25 Probe Changer is the location in front of your probe changer where the machine moves to before it picks up or drops off a probe. You should determine a location that avoids collisions with the probe changer or part.

To define the mount point for your probe changer:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).
2. In the **Active probe changer** list, select **TYPE=FCR25**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Typically, but not always, these values are 0 and 0, respectively. You need to use a

calibrated probe rotation that ensures the probe has the ability to move in and out of the probe rack during the necessary steps of the probe rack calibration procedure.

4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also manually type these values.
6. To save your changes, click **Apply**.

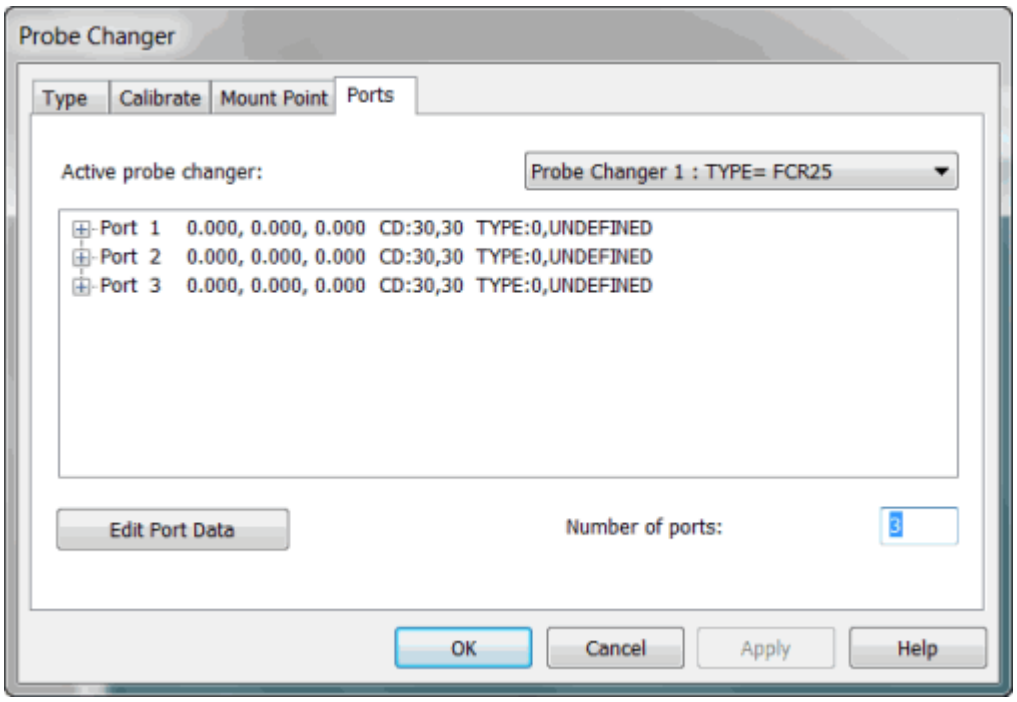
In the next step, you define the ports that your probe changes uses.

Step 3 - Define the Ports

The example described below has a PA25-SH insert in port 1(left), NO INSERT in port 2 (middle), and a PA25-20 insert in port 3 (right).

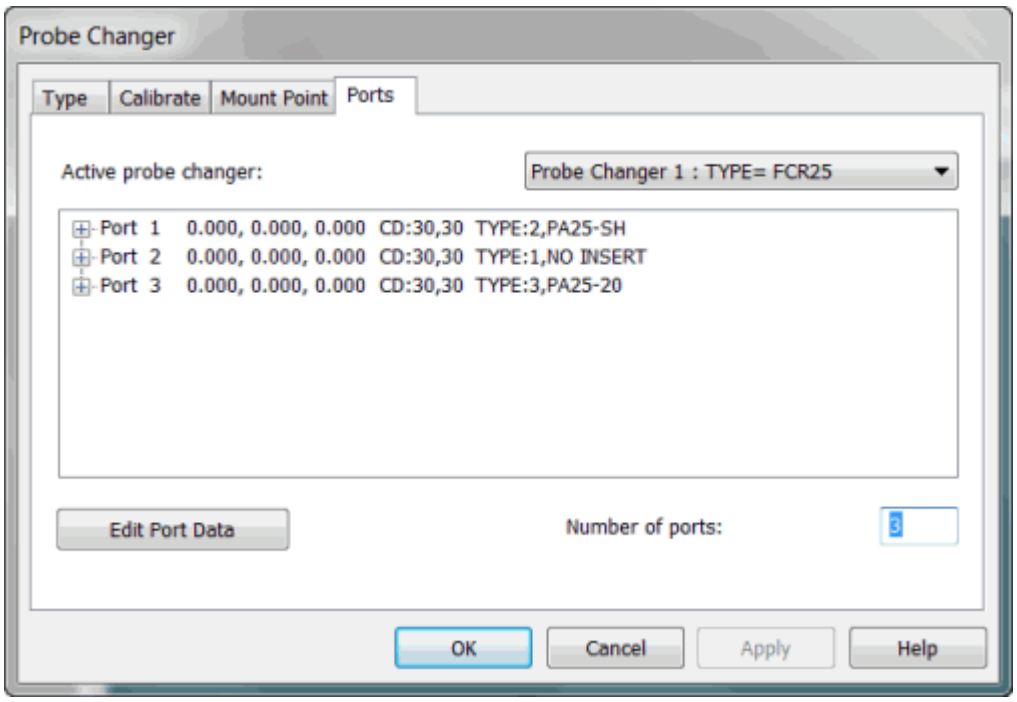
To define the ports of your FCR25 Probe Changer, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).
2. In the **Active probe changer** list, select **TYPE=FCR25**.
3. In the **Number of ports** box, in multiples of three, specify the number of ports for your FCR25 Probe Changer. PC-DMIS then lists the specified number of ports as "ports" (for example, port 1, port 2, port 3, and so forth). Until you define the ports, PC-DMIS lists the "port" entries as "UNDEFINED". You must define all of the ports in the rack before you start.



Ports tab with undefined ports.

4. Select a port from the list, and click **Edit Port Data**. The **Probe Changer Port Data** dialog box opens.
5. In the **Port type** list, select **NO INSERT**, **PA25-SH**, or **PA25-20**.
6. You may specify the **XYZ** values for the center position of the port or leave those values empty. In any case, PC-DMIS automatically populates these values upon successful calibration. See "Step 9 - Review Calibration Results".
7. To save the changes to the port data, click **OK**.
8. Repeat steps 4 through 6 for all ports in your changer.



Ports Tab

9. To save your changes, click **Apply**.

The next step prepares you to calibrate the probe changer.

Step 4 - Prepare the Probe Changer

The calibration process for the FCR25 Probe Changer may vary slightly depending on the types and locations of the inserts for each port. The process described here shows how PC-DMIS manages calibration for each port type.

Once you click the **Calibrate** button, the following message box appears:

PC-DMIS Message

Each FCR25 unit contains 3 ports. Different types of physical racks can hold varying numbers of FCR25 units. The current rack is configured to have a total of 3 ports, which means it should have 1 FCR25 units.

Please shim open all the lids and remove all modules and styli from the ports.

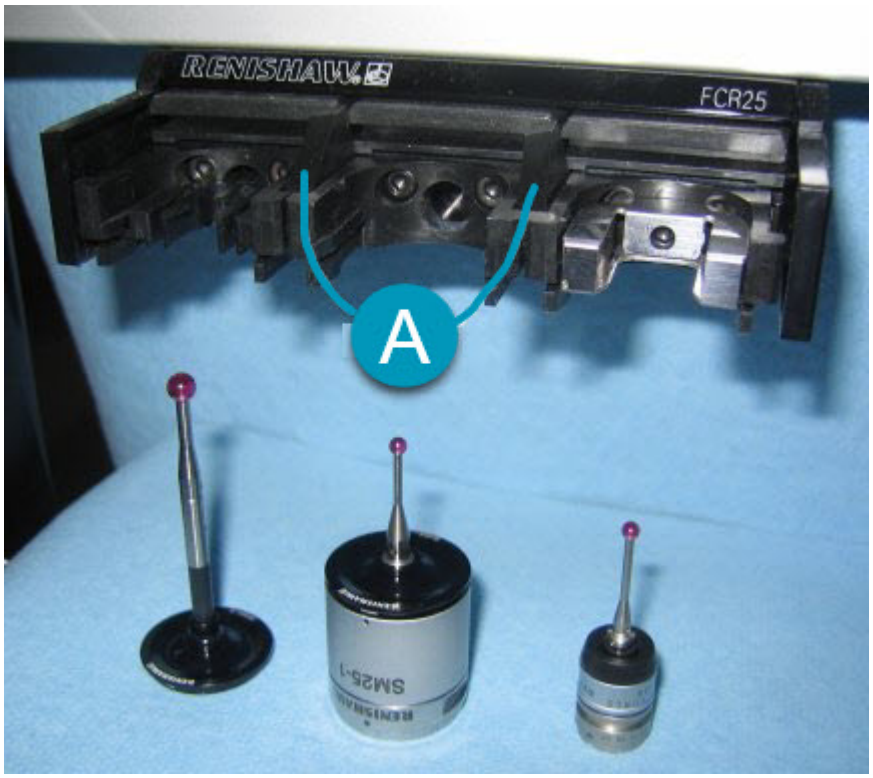
1. Read the instructions from the preceding prompt, and verify that you have the correct number of ports and FCR25 units (a unit is a set of three ports).

2. Open the lids to each port. Insert the shims, with the large ends of the shims toward the back of the ports to hold them open.

   A "shim" is a tapered piece of plastic that fits in between two ports to hold their lids open. The picture below shows a close-up view of a shim between ports 2 and 3, holding their lids open. Without a shim, the lids are shut, as you can see in port 1.

Shim holding the lids open on two ports

3. With the lids open, remove all of the modules and styli by sliding them forward and out of the ports, as shown below:



(A) - Shims

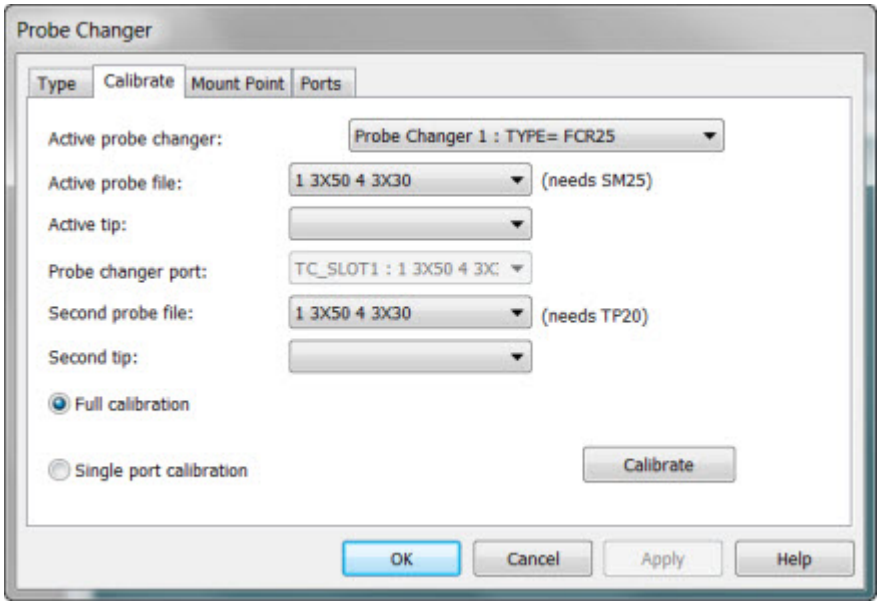
4. When you are ready to measure port 1, click **OK**.

The next step starts the calibration process.

Step 5 - Begin Calibration

Before you start the calibration process, you must specify or verify the probes to use in the calibration of your FCR25 Probe Changer.

1. Select the Calibrate tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **TYPE=FCR25**.
3. Determine the type of calibration. Select one of the following:
   - To calibrate one port, select the **Single port calibration** option. If you select the single port calibration, you must also select the needed port from the **Probe changer port** list.
   - To calibrate all FCR25 ports, select the **Full calibration** option. This documentation describes the **Full calibration** option.
4. In the **Active probe file** list, select the currently attached probe that defines the current probe configuration.
5. In the **Active tip** list, select the current tip.
6. If any ports require a secondary probe file to pick up that type of stylus, select the required probe file that defines the secondary probe configuration from the **Second probe file** list. Then select the needed tip from the **Second Tip** list. For example, a PA25-20 insert would require you to specify something like the SO25TP20_3 to accommodate the size of stylus used with the insert.
7. When you are ready to begin the calibration, click **Calibrate**.

In the next step, PC-DMIS measures port 1.

Step 6 - Measure Port 1/PA25-SH Insert

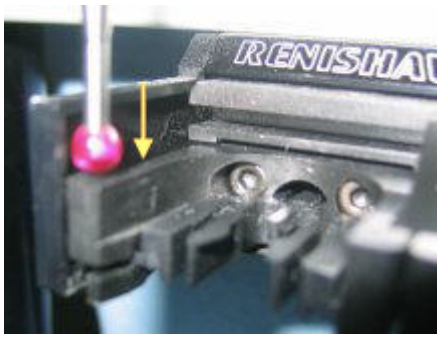
For the FCR25 Probe Changer, PC-DMIS prompts you through the process of measuring port 1 (the left-most port) through a series of message boxes. Follow the prompts and take the needed hits as directed in the instructions for each hit.

1. Hit 1 on top surface:

PC-DMIS Message

Please take a hit on the top face of the front left corner for port 1 (which is local port 1 on FCR25 unit number 1).

Use your machine's jog box to measure the first hit on the top surface of port 1 as shown in the image below:



First hit on the top surface of port 1

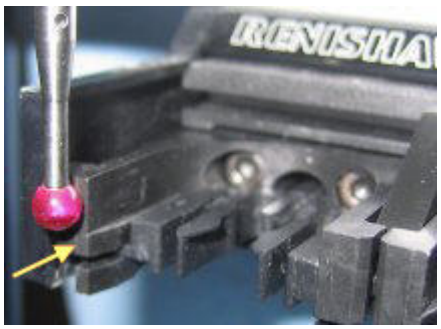
When you click **OK**, you are prompted to take the hit with the **Execution** dialog box.

2. Hit 2 on the front surface:

PC-DMIS Message

Please take a hit on the front face of the front left corner for port 1 (which is local port 1 on FCR25 unit number 1).

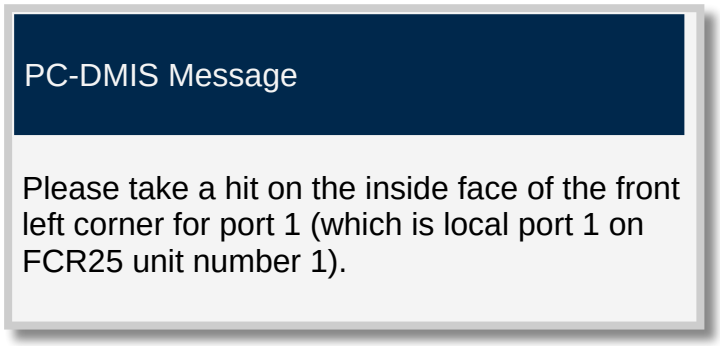
Use your machine's jog box to measure the second hit on the front surface of port 1, as shown in the image below:



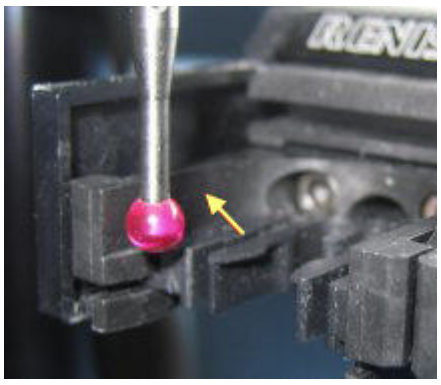
Second hit on the front surface of port 1

When you click **OK**, you are prompted to take the hit with the **Execution** dialog box.

3. Hit 3 on the inside surface:



Use your machine's jog box to measure the third hit on the inside surface of port 1 as shown in the image below:



Third hit on the inside surface of port 1

When you click **OK**, you are prompted to take the hit with the **Execution** dialog box.

This set of three hits establishes the location for the tool changer. These three hits would be the same if the port didn't have an insert. If you used a PA25-20 insert in this port, the hits would be taken on the insert in a similar fashion.

You are then asked to complete a few steps through the following message box:

PC-DMIS Message

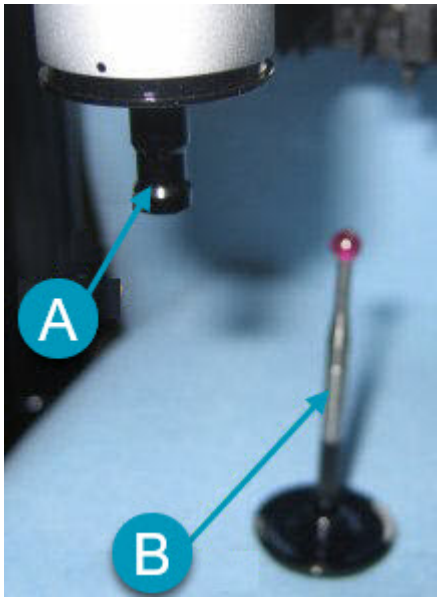
Please do the following steps in the indicated order.

1. Remove the current SH-1/2/3 stylus.
2. Attach the SHSP (Stylus Holder Setting Piece).
3. Jog the probe to a safe location with a clear line of approach to the port(s) being calibrated.
4. Then click **OK**.

After you click **OK** the machine will begin DCC measurement.

Complete the instructions in the message box:

1. Remove the specified stylus (in this case, the SH-1/2/3 stylus).
2. Attach the SHSP as shown in the image below:



WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Attach the SHSP
(A) - SHSP
(B) - SH-1/2/3

> Whenever the instructions direct that you should jog the probe to a "safe location" or a location "with a clear line of approach", you should move the probe to a position that is in front of and slightly above the rack.

3. When you have completed these steps, click **OK** to begin DCC measurement.

> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

- PC-DMIS automatically measures the three hits with the SHSP that were previously taken with the SH-1/2/3 stylus.
- It also takes a hit on the opposite inside face.
- This completes the measurement of port 1.

In the next step, you measure port 3.

Step 7 - Measure Port 3 / PA25-20 Insert

Before PC-DMIS can measure port 3 (the right-most port) for the FCR25 Probe Changer, you must first change the probe to the probe file you specified for the **Second Probe File** in step 5.

1. When prompted, remove the current module, and add the TM25-20 module to the end of the probe body. The image below shows the TM25-20 module and TP20 type stylus after this change:

TM25-20 module and TP20 type stylus

> Changing probes may not be necessary depending on your FCR25 Probe Changer configuration. For example, if there are no inserts in any of the ports, then this probe change may not be necessary. The change specified in this step is only required to accommodate the calibration of port 3 with the PA25-20 insert.

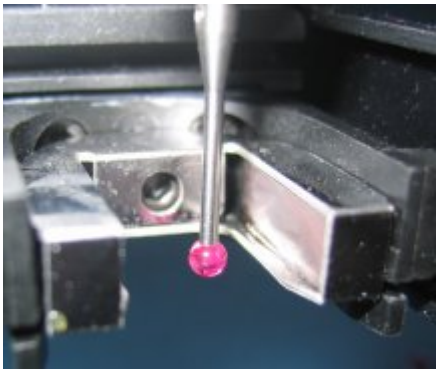
2. After you change the probe, click **OK**. PC-DMIS displays the following prompt:

> PC-DMIS Message
>
> Please jog the probe to a safe location with a clear line of approach to the port(s) being calibrated, then click OK.
>
> When you click OK the machine will begin DCC operation.

3. After you move the probe to a safe location, click **OK** to begin DCC measurement of the insert in port 3.

> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

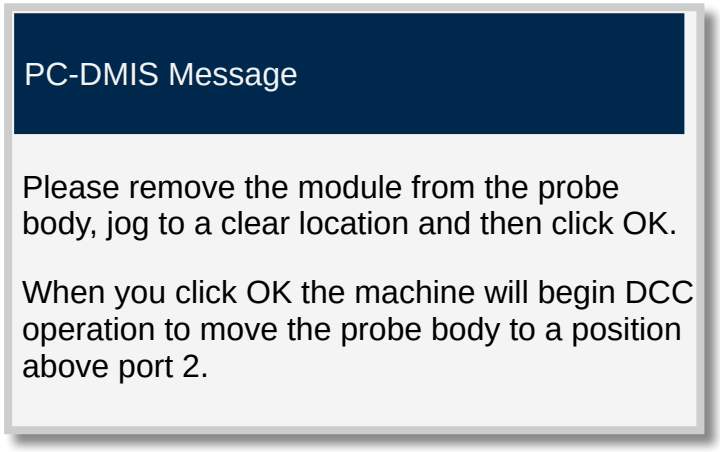
The following images show the probe taking measurements. PC-DMIS automatically takes hits to determine the location of the insert.



In the next step, you measure port 2.

Step 8 - Measure Port 2 / No Insert

1. Before you measure port 2 for the FCR25 Probe Changer, PC-DMIS prompts you to remove the module that PC-DMIS used for the measurement of port 3:



2. Once you have removed the module and moved the probe to a safe location, click **OK** to continue the process.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

PC-DMIS moves the probe body to a position that is centered above port 2 as shown in the image below. (The image also shows the module that PC-DMIS expects you to add in the next prompt.)

Position centered above port 2

The procedure for measuring port 2 would be used for all ports if you didn't have an insert in any of the ports. The ports would also be measured in a different order (port 1 first, followed by port 2, and then port 3).

PC-DMIS Message

Place the desired module into the port and slowly lower the probe body towards the module while being careful not to collide with the port.

Continue lowering until the module jumps upwards slightly due to the magnetic attraction. Observe to see if the module jumps straight up (indicates good alignment) or tilts (indicates poor alignment).

Reposition and repeat as necessary until satisfied with the alignment, then click OK.

3. Follow the prompt to place the module into the port. Then, slowly lower the probe body towards the module until the magnetic attraction causes the module to jump up towards the probe body. If you have a poor alignment, use the jog box to reposition the probe body, and repeat this process until you get a good alignment.

   The following images show the process described above.

   - Slowly lowering the probe body:

   - Module jumps upward for a good alignment:

- Module tilted for a poor alignment:

4. When you have established a good alignment, click **OK**. The following prompt appears:

PC-DMIS Message
Slowly lower the probe body onto the module. IMMEDIATELY stop when the LED on the probe head illuminates and then click OK. When you click OK the machine will begin a small DCC movement to finish seating the probe body into the module.

The image below shows the probe body that has been lowered just before the LED is lit:

5. Notice the slight gap between the probe body and the SM25-2 module. From this point, continue to lower the probe body and stop when the LED is lit. This does not close the gap completely. To finish the process, click **OK**.

   WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

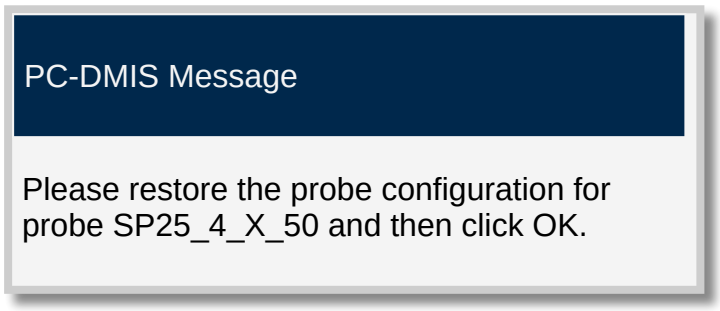
   At this point, the probe body automatically moves down the rest of the way to seat the probe module and close the gap. The following prompt appears:

   PC-DMIS Message

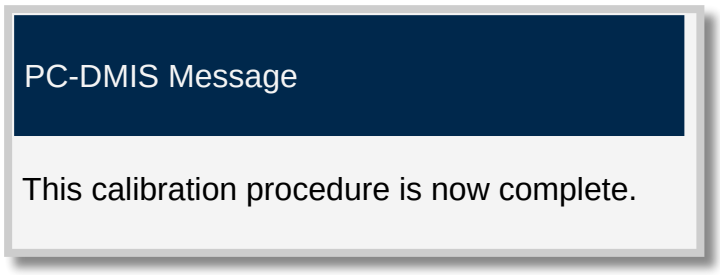
   Moving in only the one axis (as much as possible) please move the probe clear from the port and then click OK.

6. Move the probe straight out from the port to a position in front of the port as shown in the following image:

7. Click **OK**. This completes the measurement of port 2. PC-DMIS then prompts you to set the configuration back to the original probe configuration:



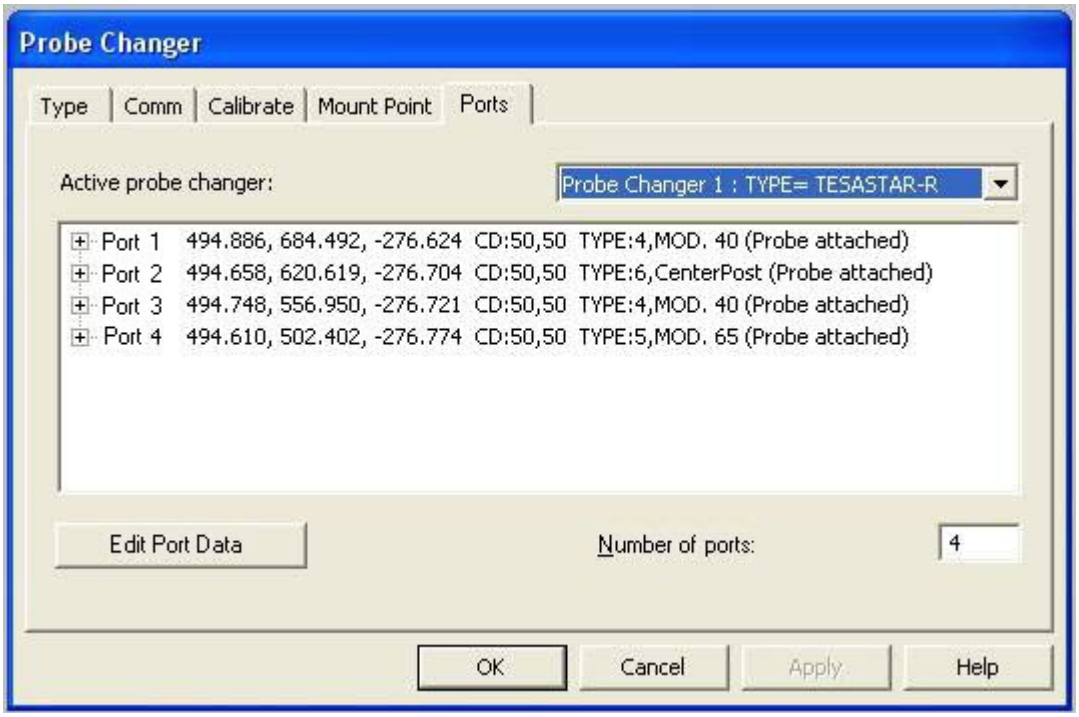
8. If needed, remove the current module, and add the modules and tips that make up the requested probe file. When you are done, click **OK**. The following prompt appears:



This step completes the calibration of the FCR25 Probe Changer. The next step describes how to review your calibration.

Step 9 - Review the Calibration Results

1. When the calibration of the FCR25 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.
    - The probe changer must be aligned parallel to either the X or the Y axis of the CMM.
    - The X and Y values should show equal spacing between ports (roughly 40 mm apart).
    - The Z values should be nearly identical, since the ports are all at the same height.

How a `LOADPROBE` command works with the probe changer:

1. During the execution of a measurement routine, the probe entities that you added to each port are automatically picked up from the port whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. Before it picks it up, the probe body moves to the mount point and then into the empty port to drop off the current probe.
3. The rack's keys rotate to the release position. The current probe remains in the port, while the probe body lifts away to detach.
4. The probe body moves over the load position above the port containing the probe.
5. The probe body moves down onto the new probe. The keys rotate again to automatically engage the new module.
6. The probe body moves back out of the port and over to the rack’s mount point.

7. The CMM continues to measure the part with the newly-loaded probe.

Calibrating the ACR1 Probe Changer

This section describes how to define and calibrate the ACR1 Probe Changer.

No inserts are used for any of the port positions. However, if you intend to use probe extensions in any of the ports, you must define them as part of the port type before you begin.

ACR1 probe changer showing two different probe extensions in ports 7 and 8.

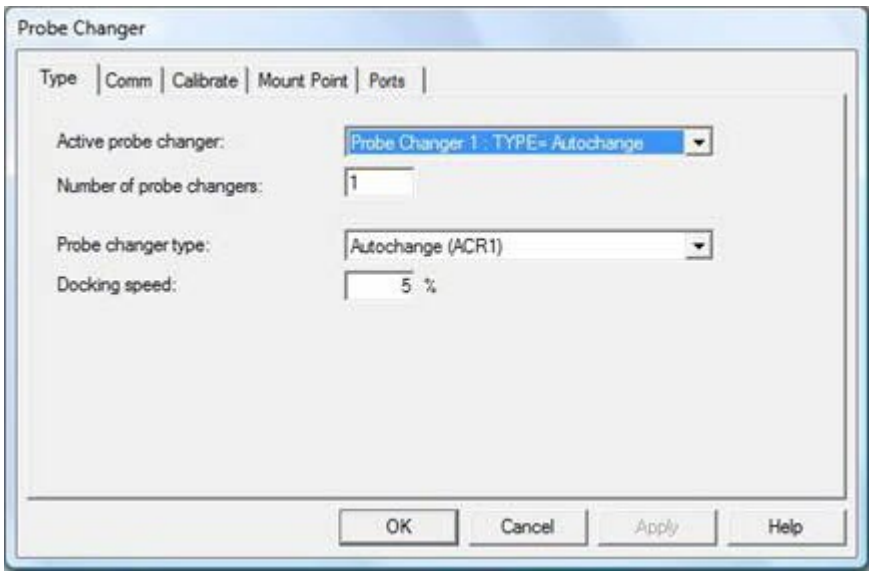
> The ACR1 Probe Changer MUST BE mounted on the machine table parallel to either the X or Y axis for successful calibration. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the ACR1 Probe Changer

To select the ACR1 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **Autochange (ACR1)**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the item that represents the probe changer you are defining. If this is the first probe changer, it appears as **Probe Changer 1 : TYPE=None**.

5. In the **Number of probe changers** box, type the number of different types of probe changers.

6. In the **Docking speed** box, type a value. A value between 15-20% is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the communications port.

Step 2 - Define the Communications Port

You can use the ACR1 Probe Changer in one of two modes:

- Host mode - In this mode, the rack controller connects to a communications port on the computer via a cable. All of the functions are based on signals between the two devices. The probe changer needs to connect to a serial communications port if you use it in host mode. You need to identify this communications (Comm) port before calibration can begin.

  If you use host mode, follow the steps below.

- Stand-alone mode - In this mode, there is no direct communications with the computer or PC-DMIS. This means that all rack functions are based on timing. The probe changer relies on timing to do a change cycle (to deposit and collect hardware from the probe changer). Respective modes are configured with the dip switches on the rear of the rack controller(s). They are covered in the Renishaw documentation or in the prints for the specific CMM configuration and are not covered here.

  If you use stand-alone mode, skip to "Step 3 - Define the Mount Point".

To define the communications port:

1. Select the **Comm** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).

2. In the **Comm port** box, select port **2** (typically if you use Host mode) or **None** (if you use Stand-alone mode). The setting depends on the machine's configuration.

Probe Changer Dialog Box - Comm Tab

3. Unless otherwise indicated, use the following default settings:

   Baud: 9600

   Parity: Even

   Data bits: 7

   Stop bits: 1

4. Click **Apply** and then **OK** to close the dialog box.

5. Restart PC-DMIS to force the new port settings to be read.

> If you change the port settings, you must restart PC-DMIS. Otherwise, the system may not work properly.

6. Once PC-DMIS restarts, select **Edit | Preferences | Probe Changer** to open the **Probe Changer** dialog box.

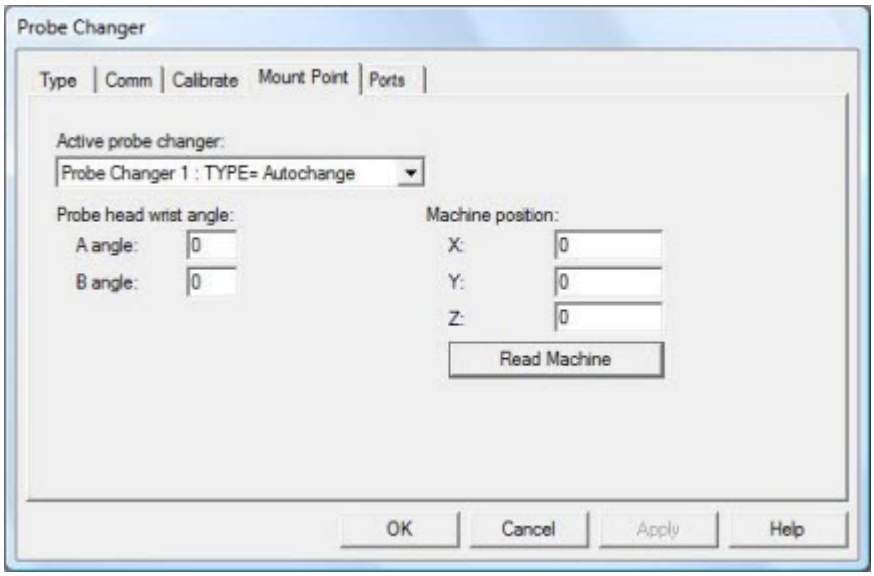
In the next step, you define the location of the probe body when you use the probe changer to switch probe components.

Step 3 - Define the Mount Point

The mount point for the ACR1 Probe Changer is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should determine a location that avoids collision with the probe changer or part.

To define the mount point for your probe changer:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=ACR1**.

3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Typically, but not always, these values are 0 and 0, respectively. You need to use a calibrated probe rotation that ensures the probe can move in and out of the probe rack during the necessary steps of the probe rack calibration procedure.

4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also manually type these values.
6. To save your changes, click **Apply**.

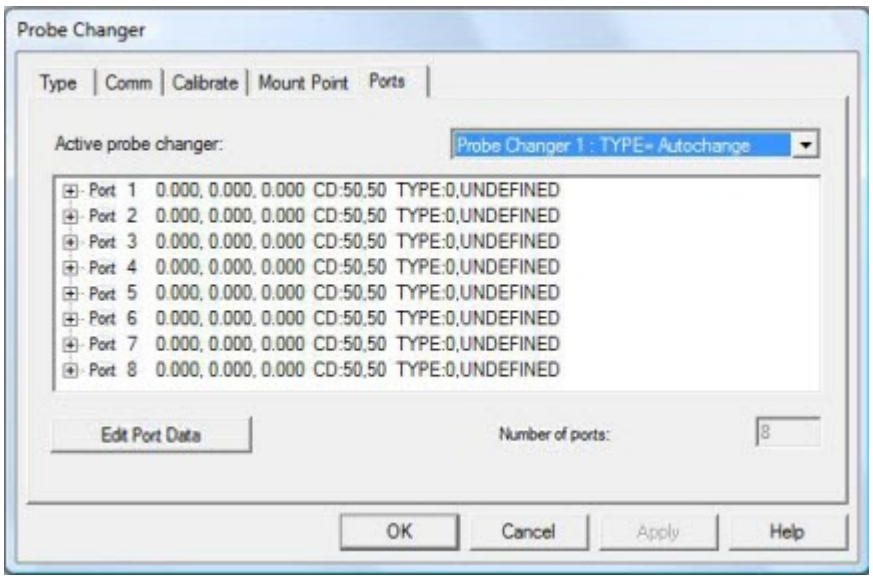
In the next step, you define the ports that your probe changer uses.

Step 4 - Define the Ports

The example described below has a probe extension in port 7 and port 8. When you use a probe extension such as these, you must identify and define it prior to calibration.

To define the ports of your ACR1 Probe Changer, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). When you first identify the probe changer, the list shows all of the ports as UNDEFINED:



Example showing the undefined ports on the Ports tab of the Probe Changer dialog box.

You must define all of the ports in the probe changer before you start.

2. In the **Active probe changer** list, select **TYPE= Autochange**.

3. Select a port in the list, and click **Edit Port Data**. The **Probe Changer Port Data** dialog box appears:

Probe Changer Port Data Dialog Box - Port type Option

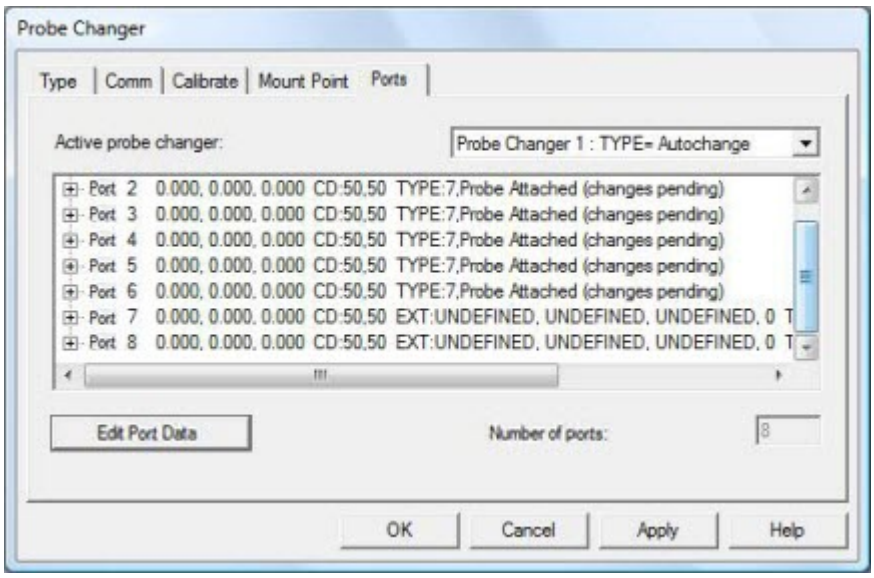
4. For each port, you need to classify its contents as either "Probe Attached" or "Extension Only". In the **Port type** list, select the appropriate option:

   - Probe Attached - Only probe bodies are in the ports, or the port is empty.
   - Extension Only - The port contains a probe extension (Renishaw PEM Autojoint to Autojoint Extension Bar, commonly called PEM). The PEM is available in various lengths, but the length is not significant at this point.

5. You can specify the **XYZ** values for the center position of the port, or leave those values empty. In any case, PC-DMIS automatically populates these values upon successful calibration. See "Step 10 - Review the Calibration Results".

6. To save your changes to the port data and close the **Probe Changer Port Data** dialog box, click **OK**. Repeat steps 4 and 5 for all ports in your probe changer.

   In the example below, ports 1 through 6 have been defined as containing only a probe body. Ports 7 and 8 are identified as each holding a PEM probe extension. It is not necessary that they be in adjacent ports; this is for illustration purposes only.



Example showing the defined ports on the Ports tab of the Probe Changer dialog box.

7. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

Step 5 - Prepare for Calibration

The calibration process for the ACR1 Probe Changer may vary slightly depending on the types and locations of the inserts for each port. The process described here shows how PC-DMIS manages calibration for each port type.

Once you click the **Calibrate** button, the following message box appears:

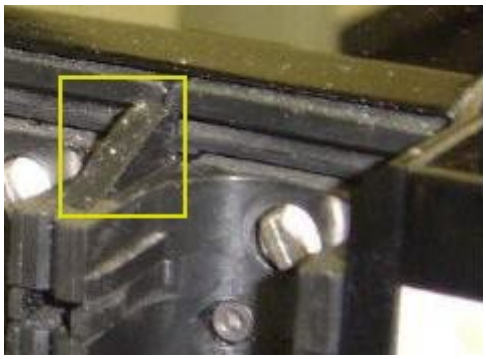
PC-DMIS Message

Please click OK, then shim open all the probe changer lids and remove all probes from the rack before proceeding with any measurement.

Note: You can safely ignore any cycle or lock errors that might occur on the rack status indicator lights during this calibration.

1. Read and follow the instructions from each prompt.
2. Open the lids to each port. Insert the shims, with the large ends of the shims toward the back of the ports to hold them open.

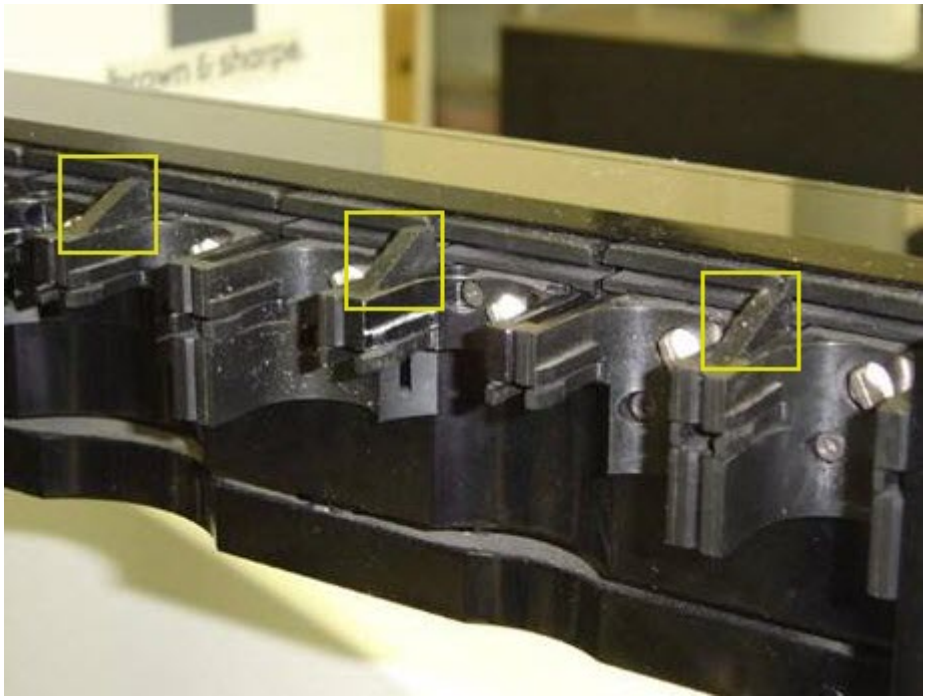
A "shim" is a tapered piece of plastic that fits in between two ports to hold their lids open. The picture below shows a close-up view of shims between multiple adjacent ports, holding open their lids.



Example showing shims between multiple adjacent ports.

Without a shim, the lids will shut.

3. With the lids open, remove all of the modules and styli by sliding them forward and out of the ports, as shown below:



Example showing the ACR1 probe changer with empty ports held open using shims.

4. When you are ready to measure port 1, click **OK**.

In the next step, you measure port 1.

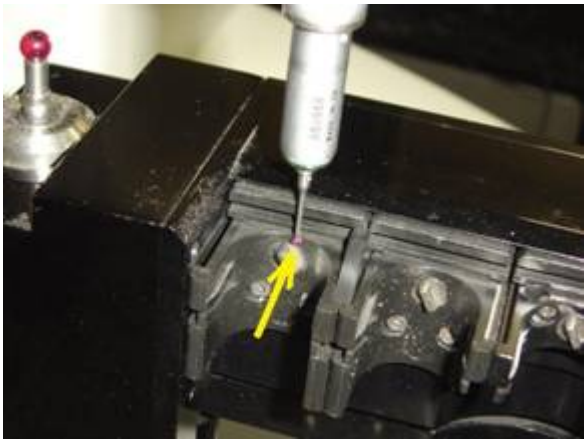
Step 6 - Measure Port 1

For the ACR1 Probe Changer, PC-DMIS prompts you through the process of measuring port 1 (the left-most port) through a series of message boxes. Follow the prompts and take the needed hits as directed in the instructions for each hit.

PC-DMIS Message

Please take a hit in the back of the circle for port 1. The hit should be taken in the middle just above the metal key.

Use your machine's jog box to measure the first hit on the back (arc) surface of port 1 as shown in the image below:



Example showing the back surface of port 1.

When you click **OK**, you are prompted to take the hit with the **Execution** dialog box.

In the next step, you measure port 8.

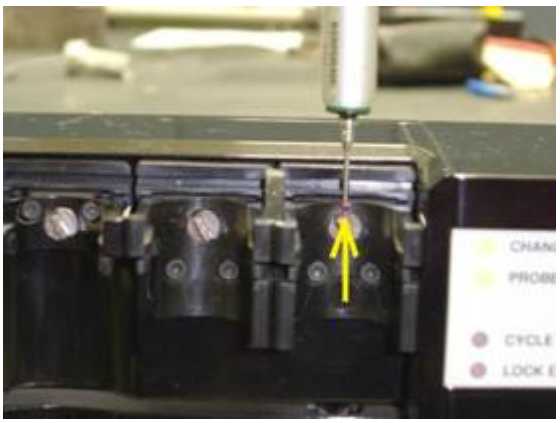
Step 7 - Measure Port 8

For the ACR1 Probe Changer, PC-DMIS prompts you through the process of measuring port 8 (the right-most port) through a series of message boxes. Follow the prompts and take the needed hits as directed in the instructions for each hit.

PC-DMIS Message

Please take a hit in the back of the circle for port 8. The hit should be taken in the middle just above the metal key.

Use your machine's jog box to measure the second hit on the back (arc) surface of port 8 as shown in the image below:



Example showing the back (arc) surface of port 8.

When you click **OK**, you are prompted to take the hit with the **Execution** dialog box.

This set of two hits establishes the location for the probe changer and would be the same if port 8 didn't have an extension.

At this point, the system goes into DCC Mode and measures a series of hits on all ports. It starts at port 8 and progresses down to port 1.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

After the DCC measurement of the ports, the next step measures the datum sphere.

Step 8 - Measure the Datum Sphere

You are now asked to complete a few steps for the ACR1 Probe Changer through the following message boxes:

PC-DMIS Message

Please jog to a clear location then click OK. If you have a moveable wrist it will then rotate to the A/B angles defined on the Mount Point tab.

Follow the instructions for this message and then click **OK**.

PC-DMIS Message

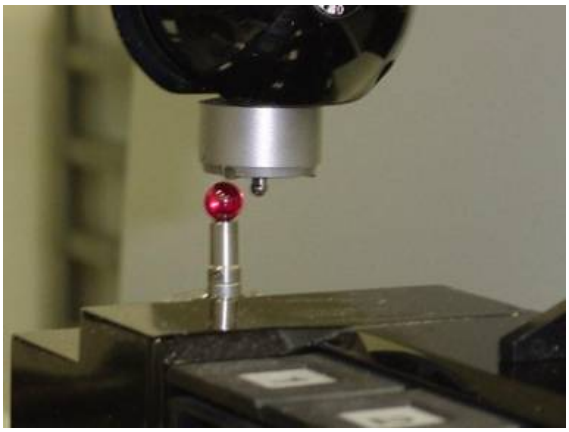
Please remove the shims and close all the probe changer lids. Then release the auto joint adapter and take 1 hit on the bottom of the auto joint with the probe changer datum sphere. Be careful to avoid the small pins and holes and take the hit on a flat area.

After you complete this point the auto joint diameter will be measured in DCC.

Follow the instructions for this message and then click **OK**.

At this point, a message displays in the **Machine commands** list in the **Execution** dialog box. It instructs you to take a hit with the stylus on top of the datum sphere (inverted probe stylus) on the left end of the probe changer.

Follow the instructions in the dialog box, remove the probe, and then take one hit on the datum sphere.



When you have completed these steps, click **OK** to begin DCC measurement.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

PC-DMIS automatically lowers the Autojoint extension to the side of the datum sphere. It then proceeds to take four hits in DCC Mode on the outside diameter of the Autojoint extension.

In the next step, you measure the datum sphere with PEM extension(s).

Step 9 - Measure the Datum Sphere with PEM Extension(s) (optional)

This example for the ACR1 Probe Changer uses two Autojoint extensions (PEM modules) in ports 7 and 8.

WARNING: At different times throughout this procedure, the machine moves in DCC mode. When this happens, to avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

The system displays this message:

PC-DMIS Message

Please attach only the extension to be used in port 7. When you click OK if you have a wrist it may rotate (if needed) to allow for any rotation of the bottom joint. When prompted then take 1 hit on the bottom of the auto joint with the probe changer datum sphere. Be careful to avoid the small pins and holes and take the hit on a flat area.

After you complete this point the auto joint diameter will be measured in DCC.

If you use PEM (extensions), it is necessary to also take a manual hit at the bottom of each one as shown in the picture below.

Example showing the manual hit at the bottom of each PEM.

When you have completed these steps, click **OK**. DCC measurement begins.

PC-DMIS Message

Please attach only the extension to be used in port 8. When you click OK if you have a wrist it may rotate (if needed) to allow for any rotation of the bottom joint. When prompted then take 1 hit on the bottom of the auto joint with the probe changer datum sphere. Be careful to avoid the small pins and holes and take the hit on a flat area.

After you complete this point the auto joint diameter will be measured in DCC.

The picture below shows the manual hit being taken with the 50 mm PEM identified as being stored in port 8.

When you have completed these steps, click **OK**. DCC measurement begins.

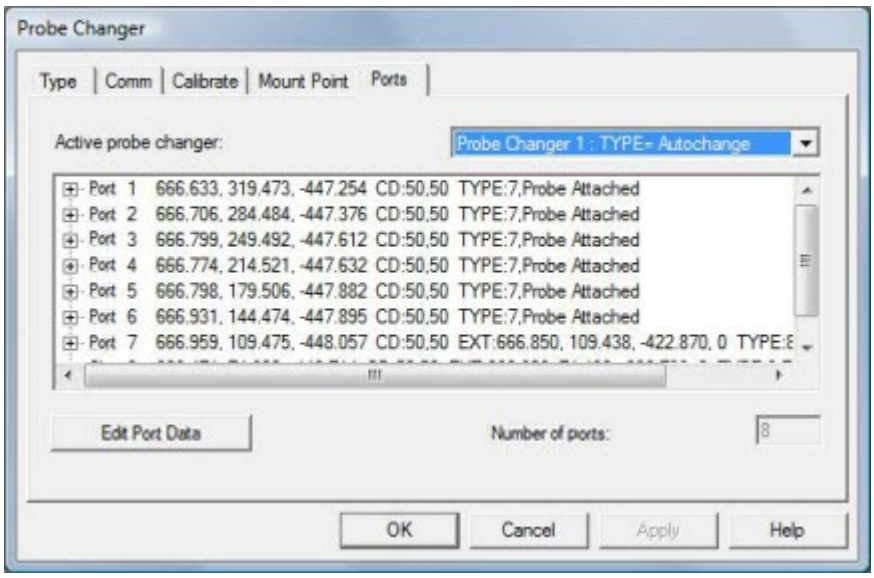
PC-DMIS Message

All measurement is now complete. Please reconnect the probe used to measure the ports and then click OK.

This completes the calibration of the ACR1 Probe Changer. The next step describes how to review your calibration.

Step 10 - Review the Calibration Results

1. When the calibration of the ACR1 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Example showing the calibration results on the Ports tab of the Probe Changer dialog box.

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

    - The position and spacing of the ports. For example, the ACR1 rack defined in this calibration process is aligned parallel to the Y axis of the CMM.
    - The X values of the ports should be, within reason, nearly identical since all of the ports are co-linear.
    - Similarly, the Z values should also be nearly identical, since the ports are all at the same height.
    - The Y values should be equally spaced (roughly 35 mm apart).

The results during measurement routine execution are:

1. During measurement routine execution, probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the port, which pushes the lid backwards.
3. The key mechanism automatically turns to disengage/engage, loads the new module, and continues to measure.

Calibrating the ACR3 Probe Changer

The calibration process for the Renishaw ACR3 Probe Changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the ACR3 Probe Changer.

No inserts are used for any of the port positions. However, if you intend to use probe extensions in any of the ports, you must define them as part of the port type before you begin.

Renishaw ACR3 Probe Changer rack

This probe changer MUST BE mounted on the machine table parallel to either the X or Y axis for successful calibration. For mounting instructions, consult the documentation that came with your probe changer.

Caution about Adaptors with an HP-L laser

You can use angular adapters with an HP-L laser in an ACR3 probe calibration. These adaptors let you use an HP-L laser at orientations that you cannot get with a standard build.

You must place the angular adapters in the same probe changer as the required probe build describes. For example, if you have an angular adapter and an HP-L laser, both components must be in the same probe changer. If you don't do this, a crash will occur that may destroy your sensor.

Be aware that once the machine picks up the angular adapter, PC DMIS does not move the rack back to the unlocked position if the HP-L is in another probe changer.

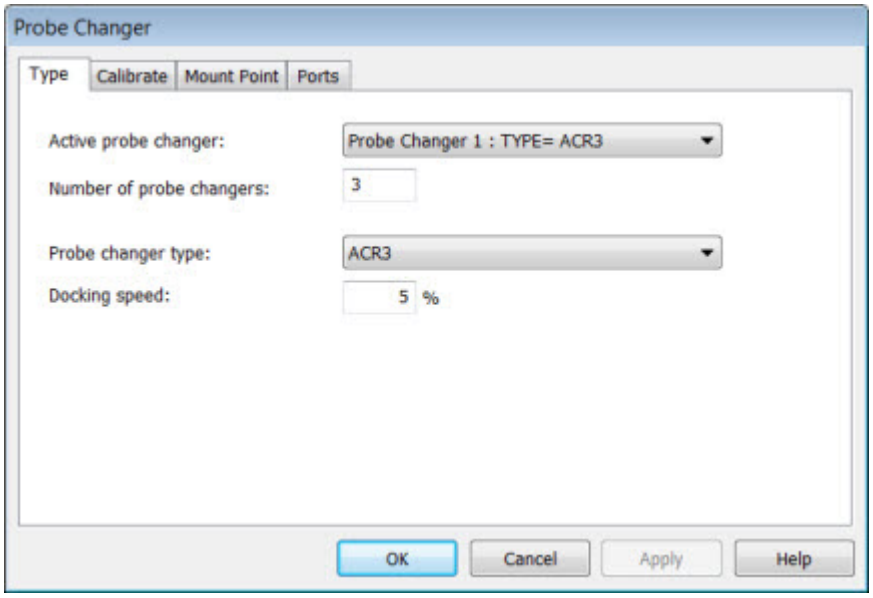
To calibrate your probe changer:

WARNING: At different times throughout this procedure, the machine moves in DCC mode. When this happens, to avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Step 1 - Select the ACR3 Probe Changer

To select the ACR3 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **ACR3**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** list, select the item that represents the probe changer you are defining. If this is the first probe changer, it appears as **Probe Changer 1 : TYPE=None**.
5. In the **Number of probe changers** box, type the number of different types of probe changers.
6. In the **Docking speed** box, type a value. A value between 15-20% is appropriate for this machine configuration. You can adjust this value any time to speed up or slow down the system.

> To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

> For details on controller configuration settings, see the documentation that came with your controller.

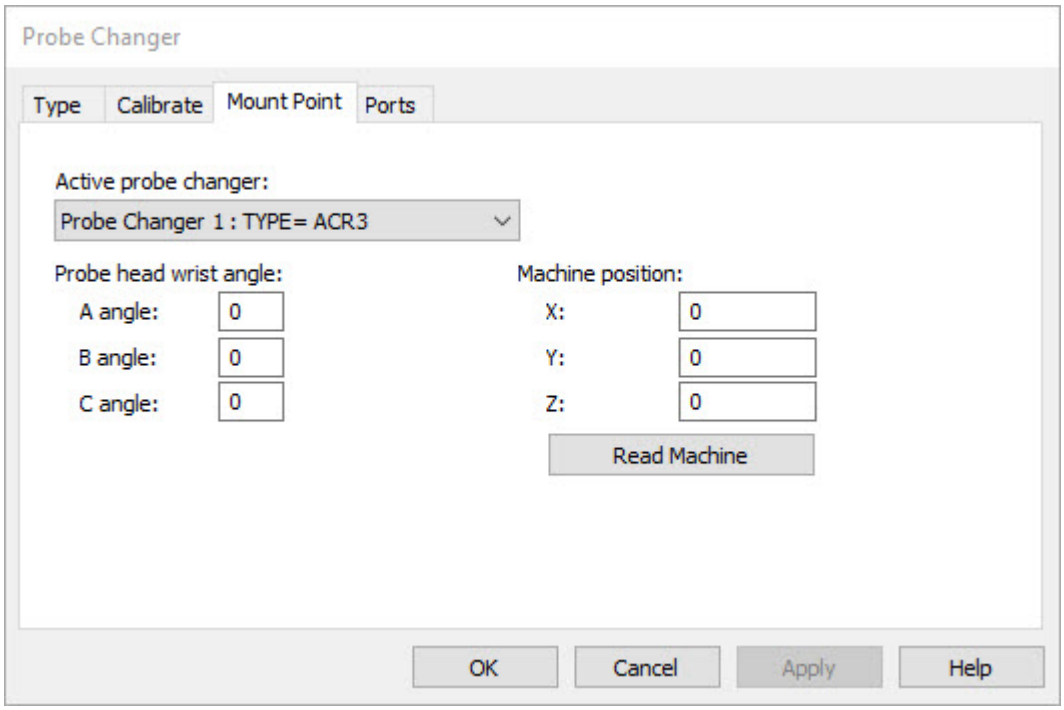
In the next step, you define the location of the probe body when you use the probe changer to switch probe components.

Step 2 - Define the Mount Point

The mount point for the ACR3 Probe Changer is the location in front of your probe changer where the machine moves to before it picks up or drops off a probe. You should determine a location that avoids collision with the probe changer or part.

To define the mount point for your probe changer:

1. From the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Mount Point** tab.



Probe Changer Dialog Box - Mount Point Tab

2. From the **Active probe changer** list, select **TYPE=ACR3**.

3. If necessary, alter the probe head wrist angles for **A angle**, **B angle**, and **C angle**. These values are required when you change probes in the probe changer. If an angle is not available in the probe head, the value for that angle should be 0 (zero). Before calibration, you need to set these values because part of the calibration procedure includes rotation of the probe head to these angles. You need to use a calibrated probe rotation that ensures the probe has the ability to move in and out of the probe changer during the necessary steps of the probe changer calibration procedure. After calibration, if you change the angles, PC-DMIS displays a warning message to inform you that recalibration is needed.

4. Use your jog box to manually move your machine to the desired mount point position.

5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also manually type these values.

6. Click **Apply** to save your changes.

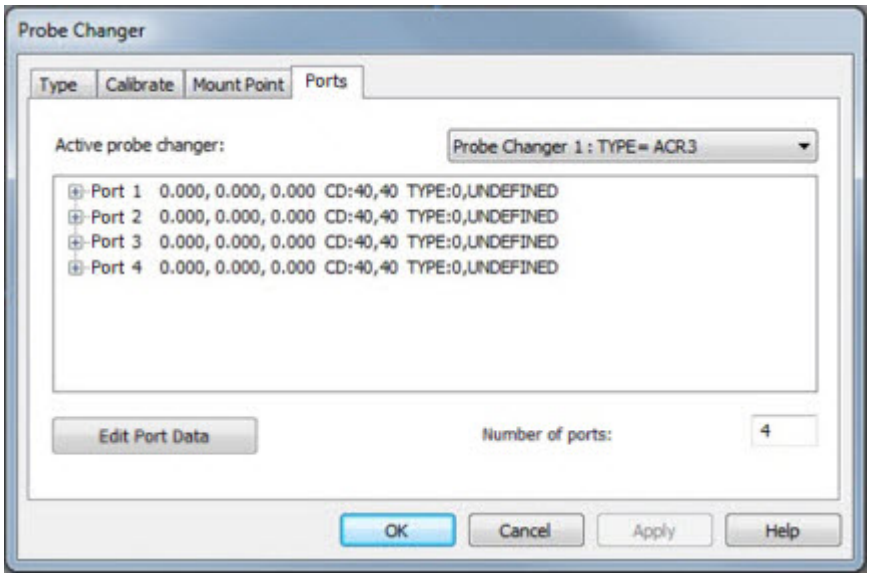
In the next step, you define the ports.

Step 3 - Define the Ports

The example described below has a probe extension in port 7 and port 8. When you use a probe extension such as these, you must identify and define it prior to calibration.

To define the ports of your ACR3 Probe Changer, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). When the probe changer is first identified, the list shows all of the ports as UNDEFINED:

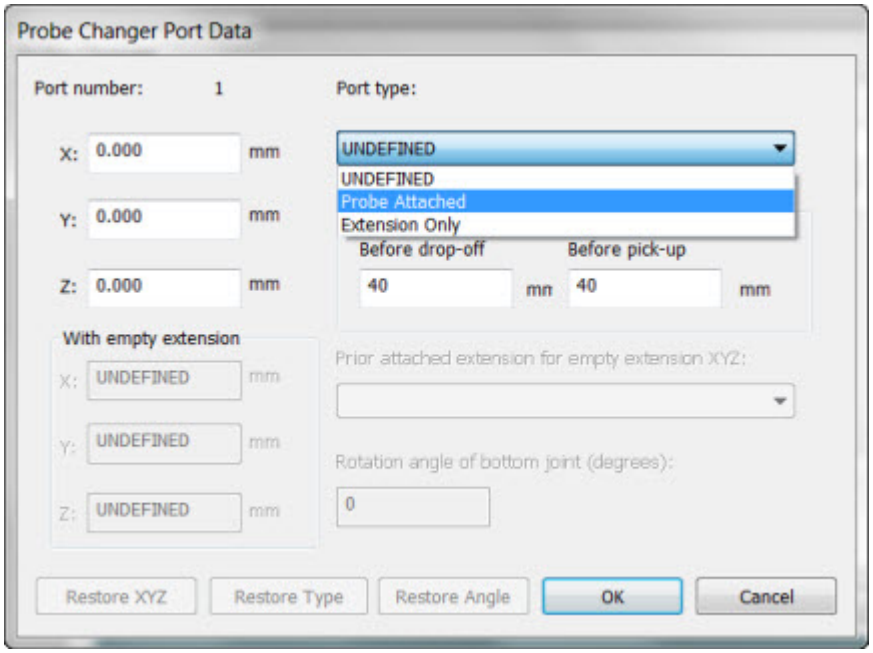


Example of the Ports tab with undefined ports.

   You must define all of the ports in the probe changer before you start.

2. In the **Active probe changer** list, select **TYPE= ACR3**.

3. Select a port in the list, and click **Edit Port Data**. The **Probe Changer Port Data** dialog box appears:



Probe Changer Port Data Dialog Box - Port type List

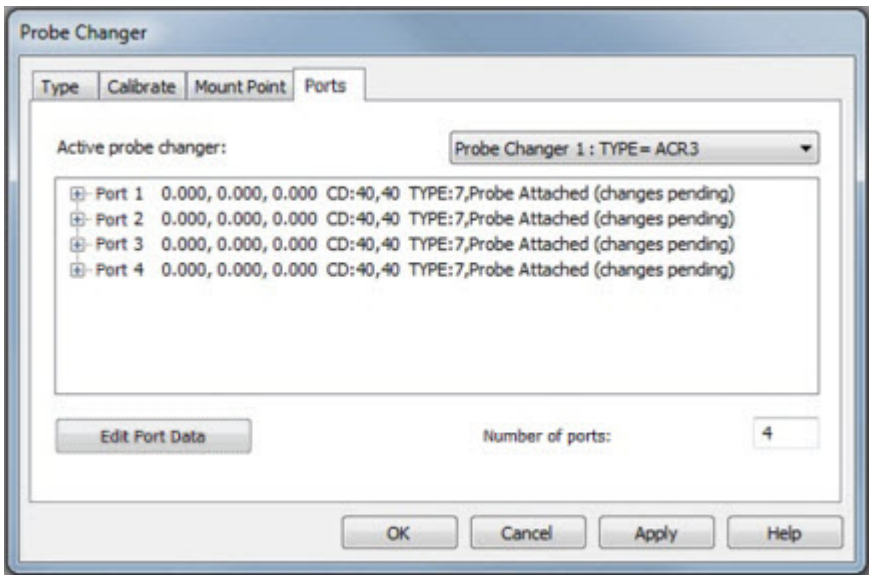
4. For each port, you need to classify its contents as either "Probe Attached" or "Extension Only". In the **Port type** list, select the appropriate option:

   - Probe Attached - Only probe bodies are in the ports, or the port is empty.
   - Extension Only - The port contains a probe extension (Renishaw PEM Autojoint to Autojoint Extension Bar, commonly called PEM). The PEM is available in various lengths, but the length is not significant at this point.

5. You can specify the **XYZ** values for the center position of the port, or leave those values empty. In any case, PC-DMIS automatically populates these values upon successful calibration. See "Step 12 - Review the Calibration Results".

6. To save your changes to the port data and close the **Probe Changer Port Data** dialog box, click **OK**. Repeat steps 4 and 5 for all ports in your probe changer.

   In the example below, ports 1 through 4 have been defined as containing only a probe body:



Example of the Ports tab showing the defined ports.

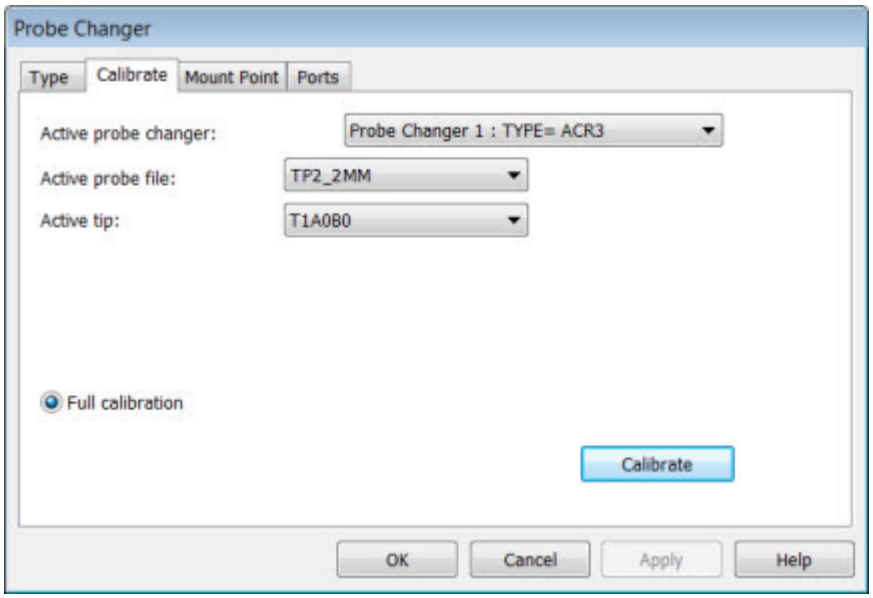
7. To save your changes, click **Apply**.

The next step starts the calibration process.

Step 4 - Prepare for Calibration

To start the calibration process for the ACR3 Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):

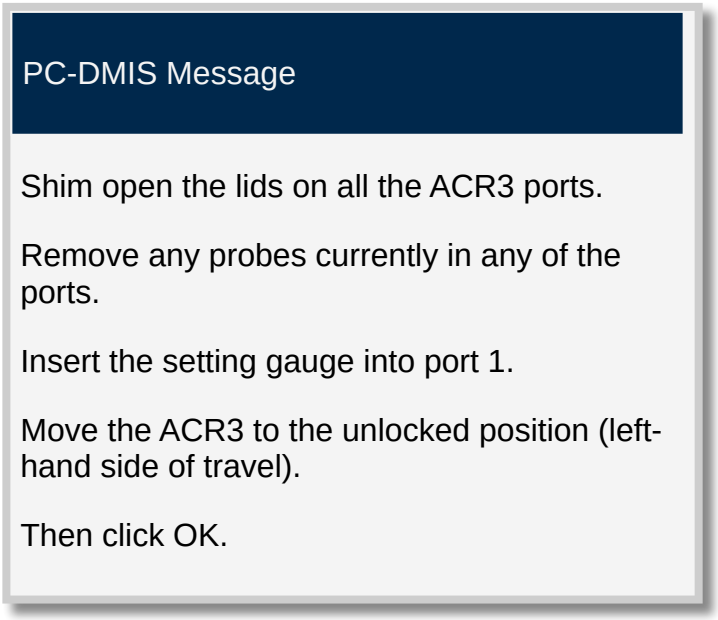


Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=ACR3**.
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what is to be used for the probe changer calibration, select the proper probe.

4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not what is to be used for the probe changer calibration, select the tip ID to be used for calibration. This is usually **T1A0B0**.

5. Click **Calibrate**. The following message box appears:



6. Read and follow the instructions from each prompt.

7. Hold the lids open by pushing them all the way back and inserting a shim before and/or after each port.

   A "shim" is a tapered piece of plastic that fits in between two ports to hold their lids open. The picture below shows a close-up view of shims between multiple adjacent ports, holding open their lids:

Example showing shims between multiple adjacent ports.

Without a shim, the lids will shut.

8. With the lids open, remove all of the modules and styli by sliding them forward and out of the ports, as shown below:

Example showing the ACR3 probe changer with empty ports held open using shims.

9. When you are ready to take the first measurement, click **OK**.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

When you start the calibration process for the ACR3 Probe Changer, a series of message boxes prompt you to record two positions and measure two manual hits. Three are required to fully define the orientation of this probe changer in the machine's XY plane.

The probe changer MUST BE ALIGNED parallel to either the X or Y axis. The manual hits locate the probe changer in the machine space. Follow the prompts and take the needed hits as directed in the instructions for each hit.

Renishaw provides a setting gauge with the probe changer. Take the first manual hit on the top face of the setting gauge that is inserted into port 1. Use the setting gauge to take hits on port 1 without having to use the probe body.

Example of a setting gauge.

The prompt for the first manual hit on the top face is:

PC-DMIS Message

Unlock the quick connect joint and remove the probe assembly.

After the probe is removed, use the jog box to move the autojoint above the central port of the setting gauge in port 1.

Slowly lower the probe head so that the autojoint just contacts the top face of the setting gauge which may be used as a feeler gauge as needed. Be careful not to move the position of the ACR3 during this process.

When in position, click OK.

PC-DMIS prompt for the first manual hit.

1. Use your machine's jog box to position the Autojoint directly on the top face of the setting gauge, as shown below:

Example showing the Autojoint positioned directly on top of the setting gauge.

2. When you are ready to take the first measurement, click **OK** with the probe in position.

In the next step, you take the second manual hit.

Step 6 - Take the Second Manual Hit

For the ACR3 Probe Changer, take the second hit with the Autojoint positioned directly in front of the setting gauge. The prompt for the second manual hit on the front edge of the setting gauge is:

PC-DMIS Message

Raise the probehead so that the autojoint is clear of the setting gauge.

Move the probehead so that the external surface of the autojoint is touching the front edge of the setting gauge.

Again be careful not to move the position of the ACR3 during this process.

When in position, click OK.

PC-DMIS prompt for the second manual hit.

1. Use your machine's jog box to position the probe directly in front of (and against) the setting gauge, as shown below:

Example showing the second manual hit.

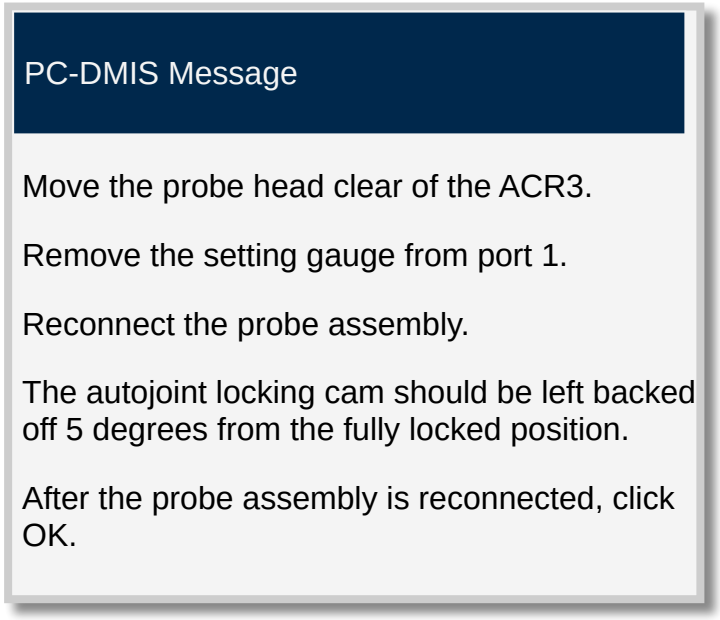
2. When you are ready to take the second measurement, click **OK** with the probe in this position.

In the next step, you remove the setting gauge and reconnect the probe assembly to the Autojoint.

Step 7 - Remove the Setting Gauge and Reconnect the Probe Assembly

For the ACR3 Probe Changer, after you have used the setting gauge to take the first (top face) hit and second (front edge) hit, do the following:

1. Remove the setting gauge from port 1 when you are prompted, as shown below:



PC-DMIS prompt to remove the setting gauge.

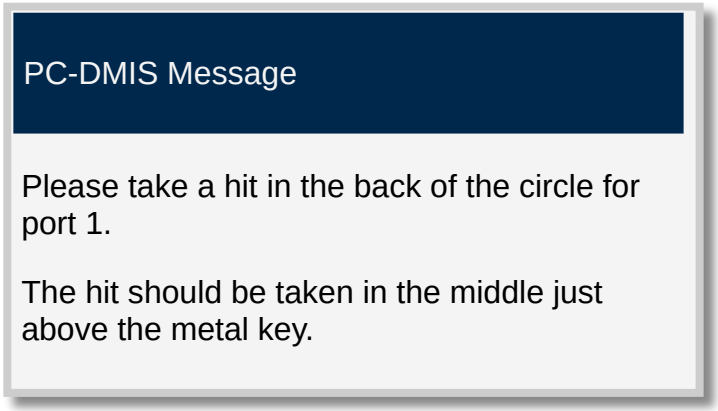
2. Reconnect the probe assembly to the Autojoint to prepare for the remaining calibration as the message instructs.

3. After you have reconnected the probe assembly, click **OK**.

In the next step, you take the first manual probe hit.

Step 8 - Take the First Manual Probe Hit

For the ACR3 Probe Changer, this is the first of two hits that you need to take to locate the probe changer. You will take the first hit in the XY plane inside the back of port 1 (just above the metal key). The prompt for the first manual probe hit is:



PC-DMIS prompt for the first manual probe hit.

1. When you are ready to take the first measurement, click **OK**.

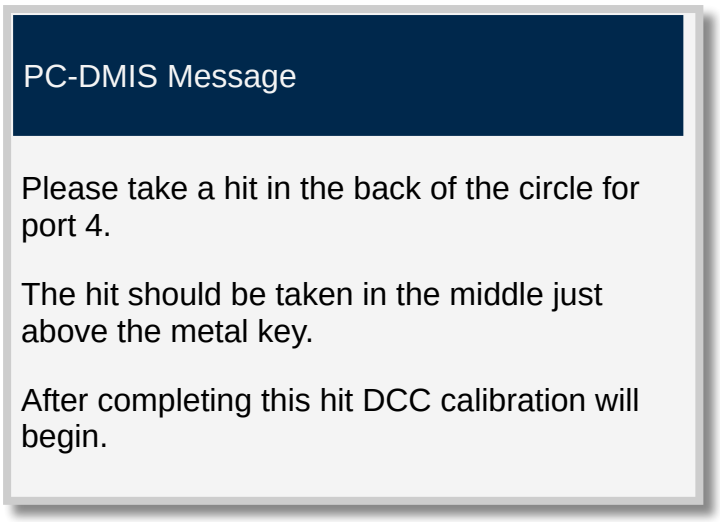
2. Use your machine's jog box to measure the first probe hit inside the back of port 1, as shown below.

Example showing the first probe hit inside the back of port 1.

In the next step, you take the second manual probe hit.

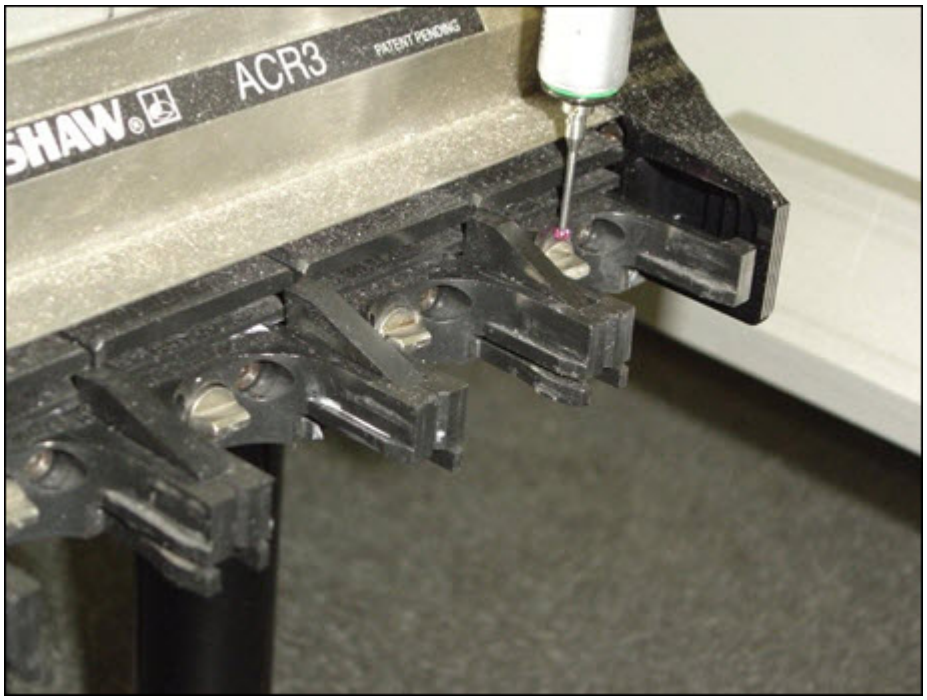
Step 9 - Take the Second Manual Probe Hit

For the ACR3 Probe Changer, this is the second of two hits that you need to take to locate the probe changer. You will take the second hit in the XY plane inside the back of port 4 (just above the metal key). The prompt for the second manual probe hit is:



PC-DMIS prompt for the second manual probe hit.

1. When you are ready to take the second measurement, click **OK**.
2. Use your machine's jog box to measure the second probe hit inside the back of port 4, as shown below:



Example showing the second manual probe hit inside the back of port 4.

This set of two manual probe hits establishes the location and orientation for the ACR3 Probe Changer.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

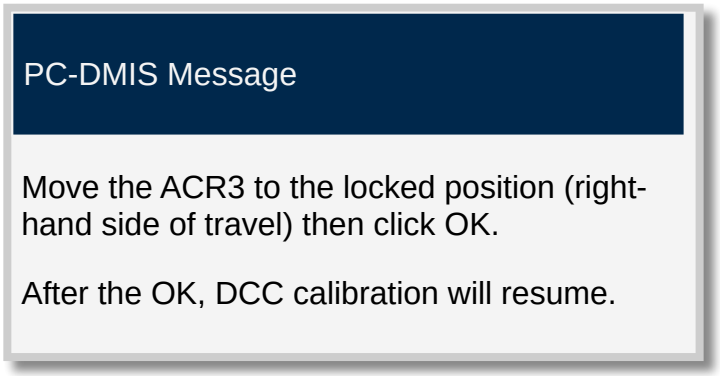
At this point, the system goes into DCC Mode and measures a series of hits that are required to accurately locate and orient the probe changer.

In the next step, you slide the ACR3 probe rack to the right side of its travel range.

Step 10 - Slide the Probe Rack to the Right Side of its Travel Range

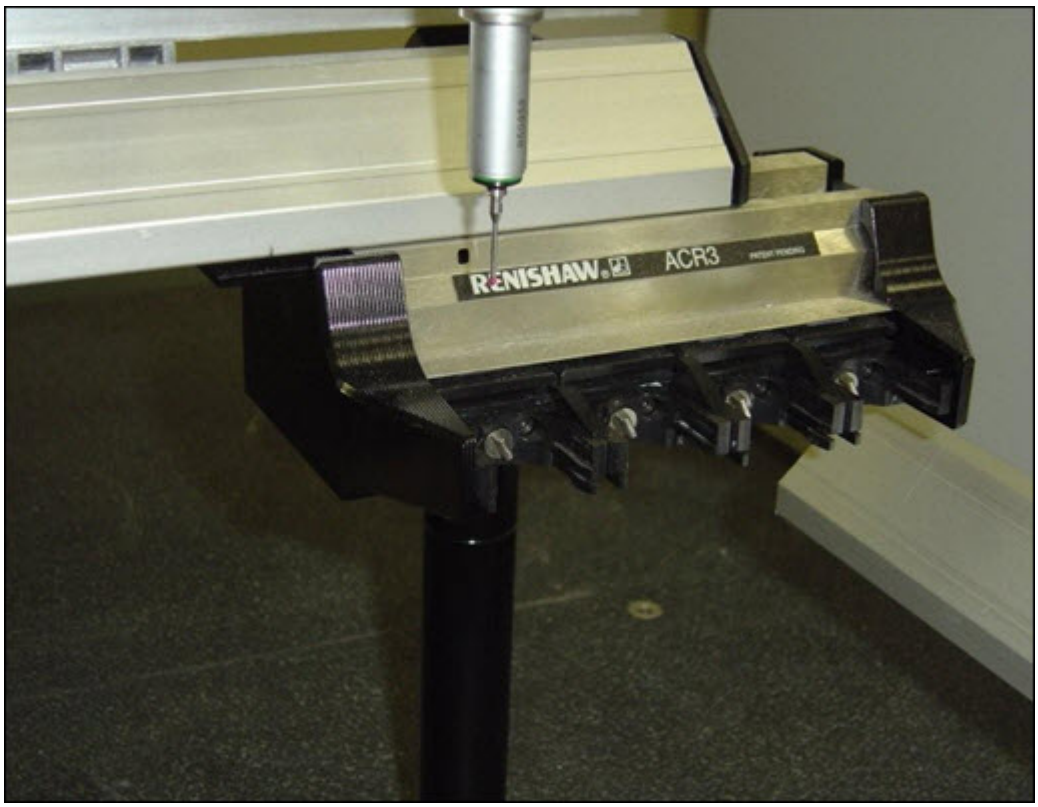
To continue with the calibration process for the ACR3 Probe Changer, you need to reposition the ACR3 probe changer to the right side of its travel range (the "locked" position.) The keys should now be oriented at approximately the 11 o'clock position.

The prompt for the locked position is:



PC-DMIS prompt to move the ACR3 to the locked position.

1. Manually slide the ACR3 probe changer to the right side of its travel range by pushing it along the FCR rail, as shown below:



Example showing the ACR3 probe changer at the right side of its travel range.

2. When the probe changer is correctly positioned, click **OK** to continue the calibration process.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

In the next step, you move the probe head into port 4.

Step 11 - Move the Probe Head into Port 4

For the ACR3 Probe Changer, you must follow additional procedures after the DCC measurement is done. This process includes steps to visually align the probe changer through a sight hole.

The prompt for moving the probe head is:

PC-DMIS Message
Move the ACR3 so that port 4 is directly behind the autojoint on the probe head. Slowly move the probe head directly back into the docked position in port 4. Try to move only along the one axis if possible.

When in position, Click OK

PC-DMIS prompt to move the probe head into port 4.

The prompt for aligning the probe changer through a sight hole is:

PC-DMIS Message

Slowly move the probe head along the direction of + or - ACR3 travel as needed until the alignment circle is centered in the alignment window above port 1.

For more information about the alignment circle and alignment window please refer to the ACR3 manual.

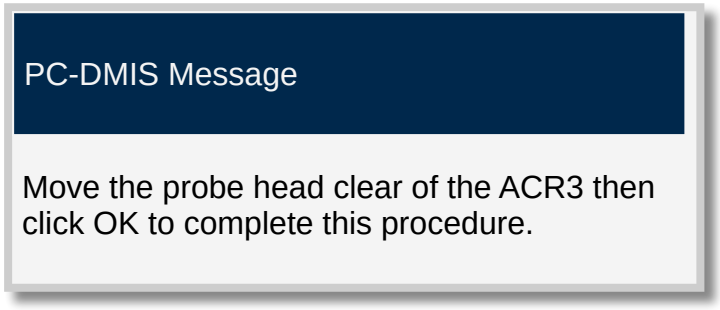
When in position, click OK

PC-DMIS prompt to align the probe changer.

1. Use your machine's jog box to slowly move the probe changer to the left or right until the square sight hole aligns with the round hole underneath, as shown below:

Example showing the visual alignment of the probe changer.

2. Once this is done, click **OK** at the last prompt, as shown below:



PC-DMIS prompt to move the probe head clear of the ACR3.

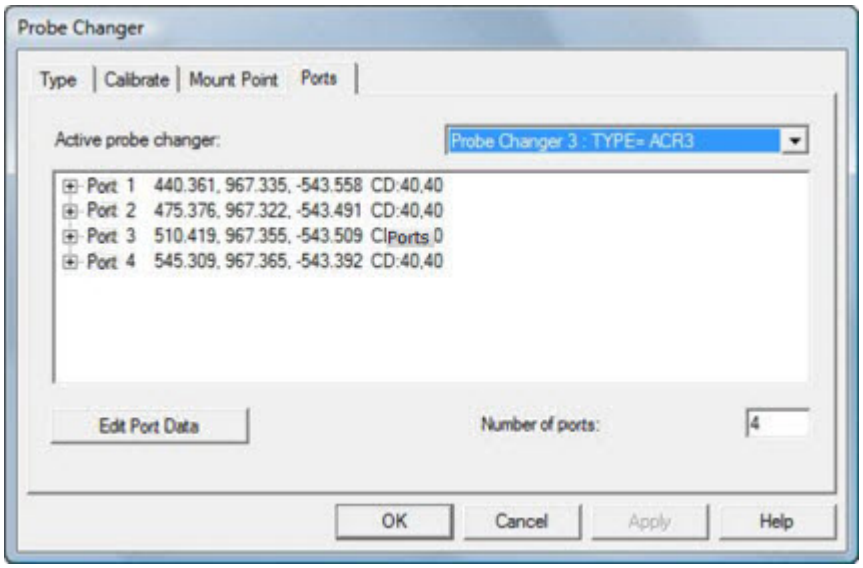
The ACR3 calibration is now complete.

3. Move the probe out and away from the probe changer's port 4 position. Return the probe changer to its right-most side of travel.

In the next step, you review the calibration results.

Step 12 - Review the Calibration Results

1. When the calibration of the ACR3 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Example showing the calibration results on the Ports tab of the Probe Changer dialog box.

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

   - This probe changer MUST BE aligned parallel to either the X or Y axis of the CMM. This means that in one of these axes, there should be little or no change between ports.
   - In the other (X/Y) axis, you should see only the distance between the ports (roughly 35 mm).

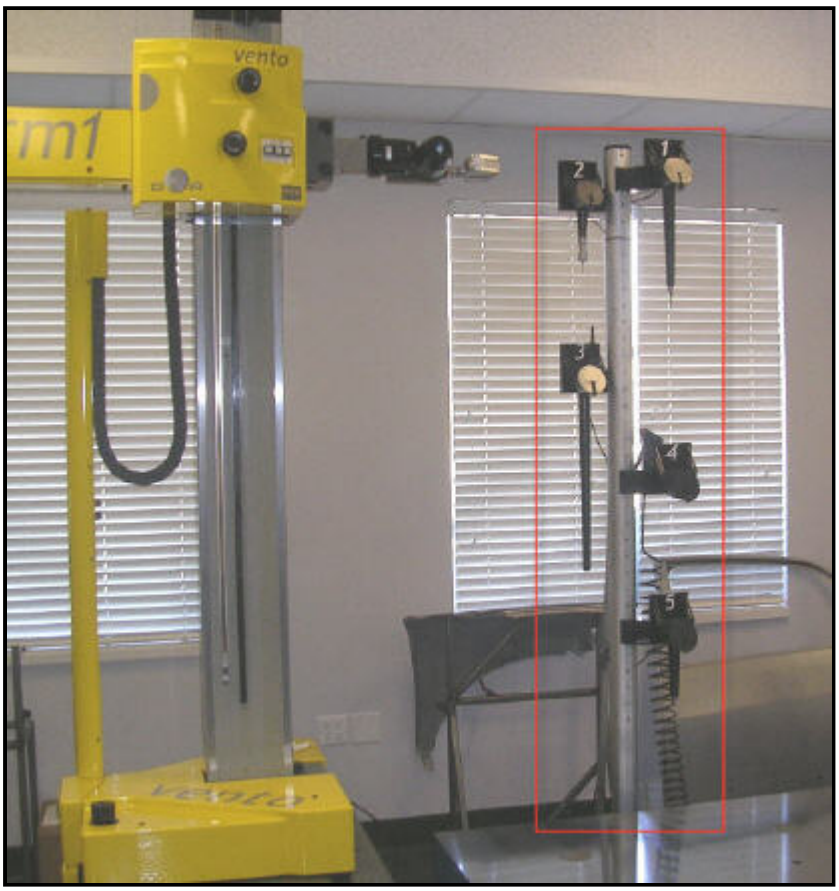
- The Z values should be nearly identical, since the ports are all at the same height.

The results during measurement routine execution are:

1. During measurement routine execution, probe entities that are added to each port are automatically picked up from that location whenever PC-DMIS executes a LOADPROBE command for that probe.
2. The probe body moves to the mount point and then into the port to drop off the current probe, which pushes the lid backwards.
3. The machine DCC movement slides the rack from the right to the left side of travel, turning the key in the process to unlock the probe from the wrist.
4. The probe then moves out from the port and into a different port to load the next probe. At this point, the DCC movement slides from the left to the right to lock this probe onto the wrist.
5. Finally, the probe travels out of the port and to the probe changer’s mount point.
6. From there, it continues to measure.

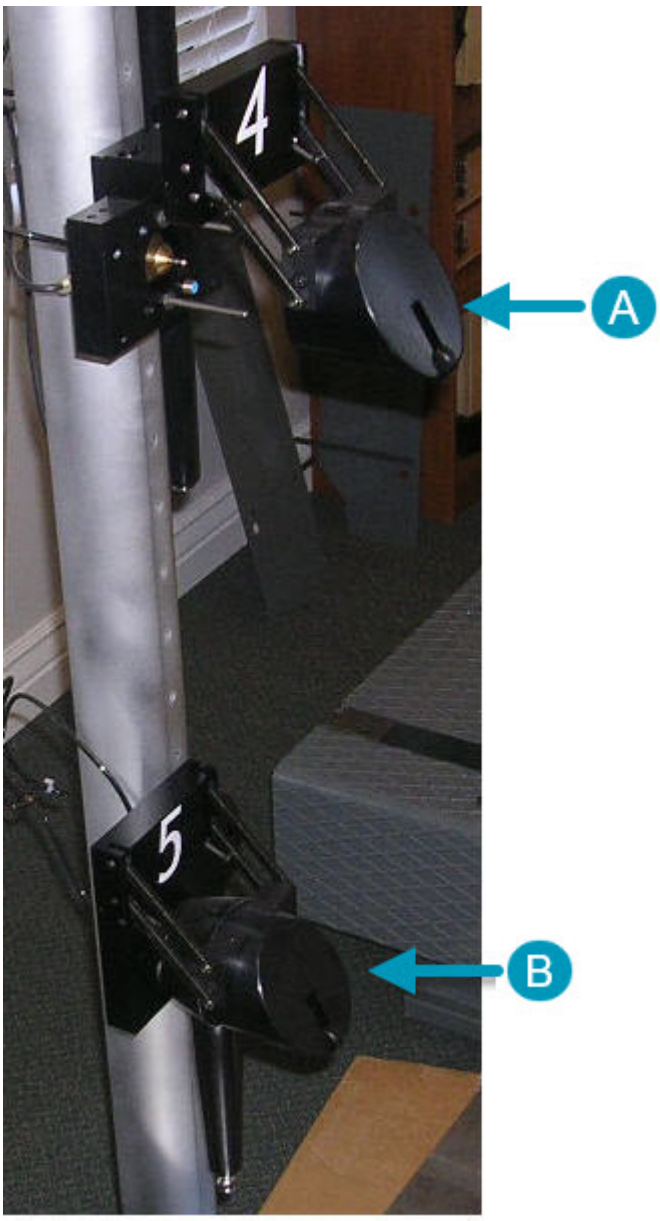
Calibrating the CW43 Probe Changer

This topic describes how to calibrate the CW43L Probe Changer.



Vertical CW43 Probe Changer with Five Ports

The CW43 Probe Changer can hold two different kinds of ports: normal ports and three-axis ports (ports that hold probes that can rotate in three axes).



A *- Port 4 shows a three-axis port.*
B *- Port 5 shows a normal port.*

The CW43L Probe Changer calibration procedure requires you to perform the calibration using a probe with the wrist map in order to properly calculate the port locations. Once you calibrate the probe changer, you can then use it to change probes regardless of whether or not they actually use the wrist map.

Before you start the calibration process, ensure that the installation personnel have mechanically aligned all of the ports so that the front face and the top of the ports are parallel to the movement of the machine's ram. If the ports are not properly aligned in this fashion, problems can occur during a probe change operation.

Step 1 - Select the CW43 Probe Changer

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **CW43L**.
3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Number of probe changers** box, type the number of different types of probe changers. Generally, this is **1**.

   The **Active probe changer** list now reads: **Probe Changer 1 : Type= CW43L**.

Step 2 - Define the Ports

In this step, define the number of ports for your probe changer as well as the hardware held in each port.

1. Select the **Ports** tab.
2. In the **Number of ports** box, specify the number of ports used on your probe changer.
3. Click **Apply**.
4. Expand each port in the list, and define the probe file names that correspond to the hardware (probe or extension) that each port holds. You can do this now or any time after this point.
5. Click **Apply** when you are finished.

Getting Around Three-Axis Ports

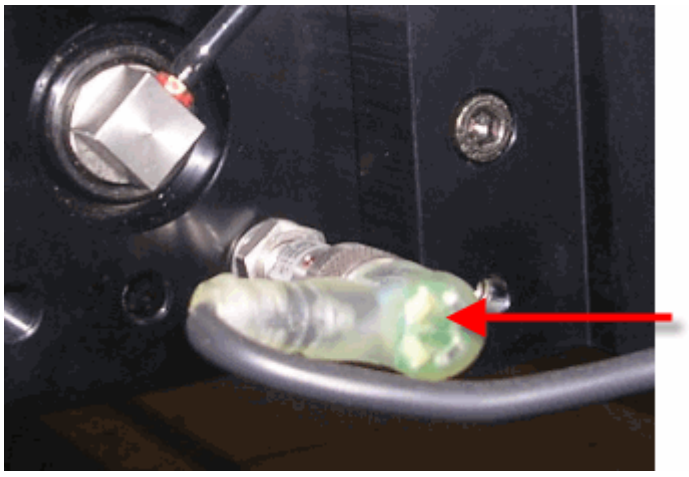
If you have a three-axis port, notice that it sticks out from the rack more than the normal ports. This may cause clearance problems when performing a probe change operation. To help avoid crashing into these ports, we recommend that you install this type of port on the ends of the probe changer: either on the top or the bottom on a vertical probe changer or on the left or right sides of a horizontal probe changer. If you cannot do this due to space constraints in your environment, modify the default clearance distance that the probe uses when moving to and from the different ports. You can do this by accessing the PC-DMIS Settings Editor and modifying the `TCRackClearance` or `TCRack3AxisClearance` entry in the next step.

- `TCRackClearance` - This entry is the clearance distance in front of a normal port. It has a default value of 150 mm.
- `TCRack3AxisClearance` - This entry is the clearance distance in front of a three-axis port. It has a default value of 300 mm.

Step 3 - Define the Three-Axis Port and Change Clearance Distances

You only need to perform this step if you have a three-axis port (a port capable of storing a three-axis probe). If not, go to the next step. You can perform this step later, but it is recommended that you do it now. This step essentially tells PC-DMIS which port is the three-axis port and whether or not you intend to perform load or unload operations automatically for that port.

1. Click **OK** to close the **Probe Changer** dialog box.
2. Exit the PC-DMIS application.
3. Launch the PC-DMIS Settings Editor.
4. Navigate to the **Leitz** section.
5. Select the `CW43LThirdAxisTCSlot` or `CW43LThirdAxisTCVerticalSlot` entry according to the port type mounted on the tool changer. The `CW43LThirdAxisTCSlot` entry is used for the old port type, where the third axis lies horizontally. The `CW43LThirdAxisTCVerticalSlot` entry is used for the new port type, where the third axis lies vertically. By default, these entries have a value of -1, which means that a three-axis port is not available on the probe changer.
    - If you plan to *manually* load and unload the three-axis probe from the port yourself, you should set this value to **0**.
    - If you plan to have PC-DMIS *automatically* load and unload the probe, you should set this value to the port number that holds the three-axis probe.
6. Ensure that the `CW43LTTest3AxisSlotTC` entry is set to **True**. There are two LED lights in a magnetic relay located on the back of a three-axis port: a green light and an amber light. The green light means that the port relay has power. The amber light is only illuminated if the three-axis joint is physically located in the port. A **True** value for this entry checks for the amber light and is an indication to the CMM controller that it is safe to apply the 24 volt DC motor power to the three-axis joint.



Solenoid Connection with Green and Amber LED Lights

> During a probe changer operation, NEVER attempt to *manually change* the three-axis probe while the green LED light on the top of the three axis joint is illuminated. An illuminated green LED light signifies that motor power (+24V DC) is present. If a probe change is done while there is motor power, it could cause a voltage spike that would damage the third axis motor. The same is possible for other voltage signals (+5V DC, +12V DC, and so on) needed for items that may attach to the third axis (NC100 video probe, and so forth). This applies only when the joint is connected to the probe head.

7. If needed, you can also modify the clearance distances for the `TCRackClearance` and `TCRack3AxisClearance` entries. You should only need to do this if your three-axis port could not be placed on the ends of the rack.
8. If you need to change the third-axis angle position when the probe is inside the port, set the desired value for the `CW43LTThirdAxisSlotAngle` entry. The default value is -1. If the value is -1, the angle value is the default value (according to the port type). You can set a value in these ranges:
   - 0 to 180
   - 0 to -180
9. If needed, modify the port cover lift direction in the `CW43LThirdAxisTCLeftToRightLift` entry. To determine the value, stand in front of the probe changer port and check the direction.
   - If the port cover lift direction is from left to right, set the value to **True** (this is the default value).
   - If the port cover lift direction is from right to left, set the value to **False**.

> PC-DMIS uses the value that you set for the `CW43LThirdAxisTCLeftToRightLift` entry only when the value for the `CW43LThirdAxisTCVerticalSlot` entry is not set to **-1**.

10. Save any changes and then close PC-DMIS Settings Editor.
11. Restart PC-DMIS and reload your measurement routine.
12. Select **Edit | Preferences | Probe Changer** to open the **Probe Changer** dialog box.

Step 4 - Prepare for Calibration

This step defines the probe file and tip angle to use during the calibration process.

1. Select the **Calibrate** tab.

2. In the **Active probe file** list, select the probe to use.

3. In the **Active tip** list, select the tip angle to use. The tip angle you use depends on your specific machine. This tip angle is used throughout the entire calibration process.

Step 5 - Begin Calibration

In this step, you need to decide whether to calibrate a single port in the probe changer or calibrate all of the ports, and then start the calibration procedure.

1. Decide the type of calibration to perform: either **Single port calibration** or **Full calibration**.

    - If you selected **Single port calibration**, the **Probe changer port** list becomes available for selection. In the **Probe changer** list, select the single probe to calibrate.

    - If you selected **Full calibration**, PC-DMIS calibrates all the available ports on your probe changer, starting with the first port and moving sequentially through all the ports.

2. Click the **Calibrate** button. PC-DMIS prompts if it is OK to rotate to the previously defined probe angle.

3. Click **OK**. The probe rotates to the defined angle, and a message box appears prompting you to take three hits.

Step 6 - Calibrate by Taking Three Hits

In this step, the probe is manually moved to take three hits, one on three different faces of each port being calibrated. For a full calibration, hits on the first defined port are taken and then move sequentially to the second port, and so on until the last. For a single port calibration, only hits on the one port being calibrated are made.

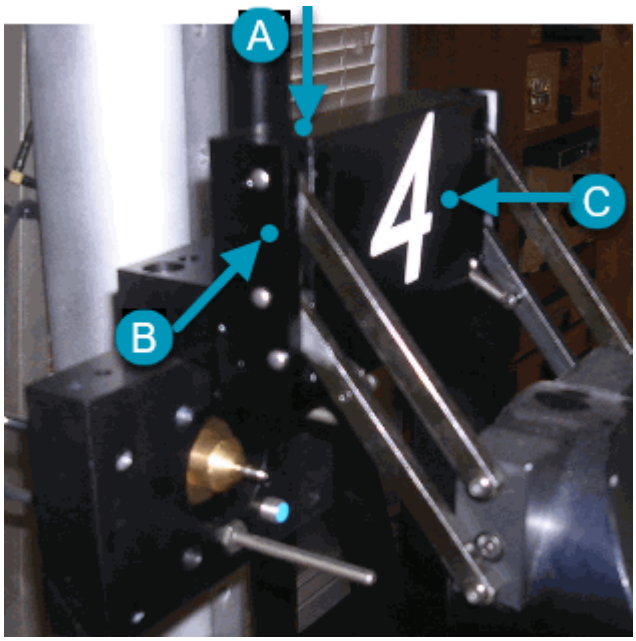
For a vertical probe changer:

- Take the first hit anywhere on the *top* of the top of the port. Press **Done**.

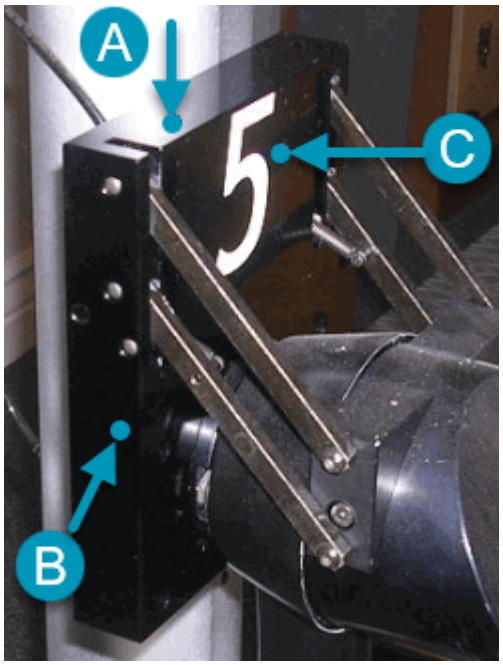
- Take the second hit anywhere on the *side* of the port *nearest the ram* of the machine. Press **Done**.

- Take the third hit anywhere on the *front face* of the port. (If this is the three-axis port, take the hit on the face to which the hinges of the port cover are attached.) Press **Done**.

Examples of taking three hits on a vertical probe changer. The following two pictures provide examples of the surfaces where the three hits should be taken for both a three-axis probe port and a normal probe port.



Three-axis probe port on a vertical rack



Normal probe port on a vertical rack

A - First hit on the top surface
B - Second hit on the side closest to the ram (depending on where your CMM ram is located, this could be on the other side)
C - Third hit on the front face

For a horizontal probe changer:

- Take the first hit anywhere on the *top* of the top of the port. Press **Done**.
- Take the second hit anywhere on *either side* of the port. Press **Done**.

- Take the third hit anywhere on the *front face* of the port. (If this is the 3 axis port, take the hit on the face to which the hinges of the port cover are attached.) Press **Done**.

Step 7 - Define a Mount Point

In this step, you define a safe location and probe head angle in front of the rack where the CMM is able to reach all the ports. This is called the "mount point".

The mount point location is NOT the same thing as the distance in front of the probe changer as defined by the `TCRackClearance` and `TCRack3AxisClearance` entries.

1. Select the **Mount Point** tab.
2. In the **Probe head wrist angle** boxes, use the **A angle** and **B angle** boxes to define the probe head angle to use when the probe head is at the mount point location.
3. Use the jog box to manually move the probe to where you want your mount point, and then click the **Read Machine** button. PC-DMIS pulls in the XYZ location from the CMM.
4. Click **Apply** and then **OK**.

You are now done with the calibration of the CW43 Probe Changer. You can now perform probe changes.

Remember, during a probe changer operation, NEVER attempt to manually change a three axis probe while the green LED light on the top of the three axis joint is lit. This applies only when the joint is connected to the probe head.

Calibrating the HR-MP (used with TM or THD) Probe Changer

This section describes how to calibrate the HR-MP (used with TM or THD) Probe Changer. The process discussed here is specific to the HR-MP (used with TM or THD) Probe Changer.

You can calibrate the probe changer in the following ways::

- When you calibrate the probe changer for the first time, you need to take manual hits.
- When the probe changer is removed and mounted at the same location, you can select to calibrate the probe changer in DCC Mode without taking manual hits.
- When the probe changer is removed and relocated on the CMM table, you can follow the same steps when you calibrate for the first time.

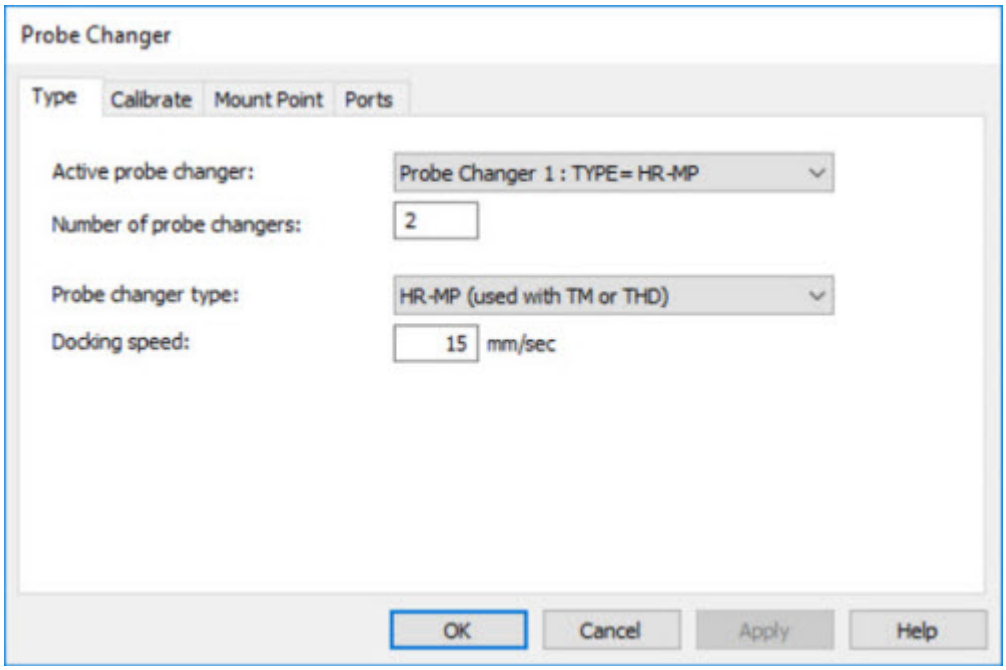
- The calibration of a single port is available in DCC Mode or Manual Mode.

To calibrate your probe changer:

Step 1 - Select the HR-MP (used with TM or THD) Probe Changer

To select the HR-MP (used with TM or THD) Probe Changer, follow these steps:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **HR-MP (used with TM or THD)**.



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** list, select the list item that represents the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.
6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

> To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

> For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

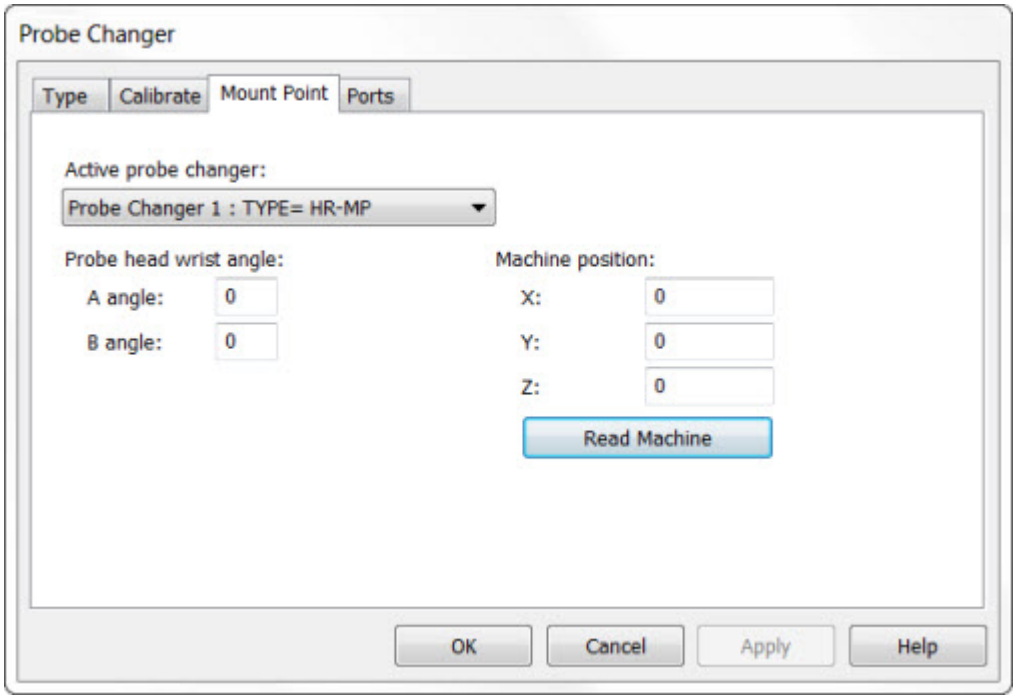
Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).

2. In the **Active probe changer** list, select **TYPE=HR-MP**:



Probe Changer Dialog Box - Mount Point Tab

3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use a calibrated probe rotation that ensures that the probe has the ability to move in and out of the probe rack during the necessary steps of the probe rack calibration procedure.

4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

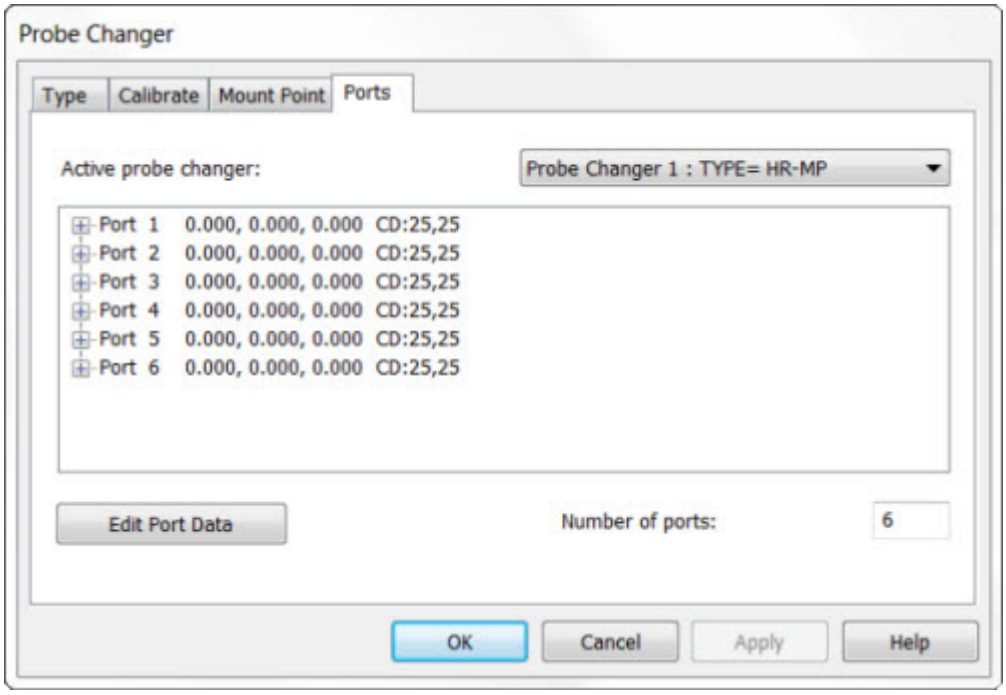
In the next step, you define the ports.

Step 3 - Define the Ports

The final step prior to rack calibration is to define the ports on the rack that you are calibrating.

To define the ports for your HR-MP (used with TM or THD) Probe Changer, follow these steps:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).
2. In the **Active probe changer** list, select **TYPE=HR-MP**:



Probe Changer Dialog Box - Ports Tab

3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

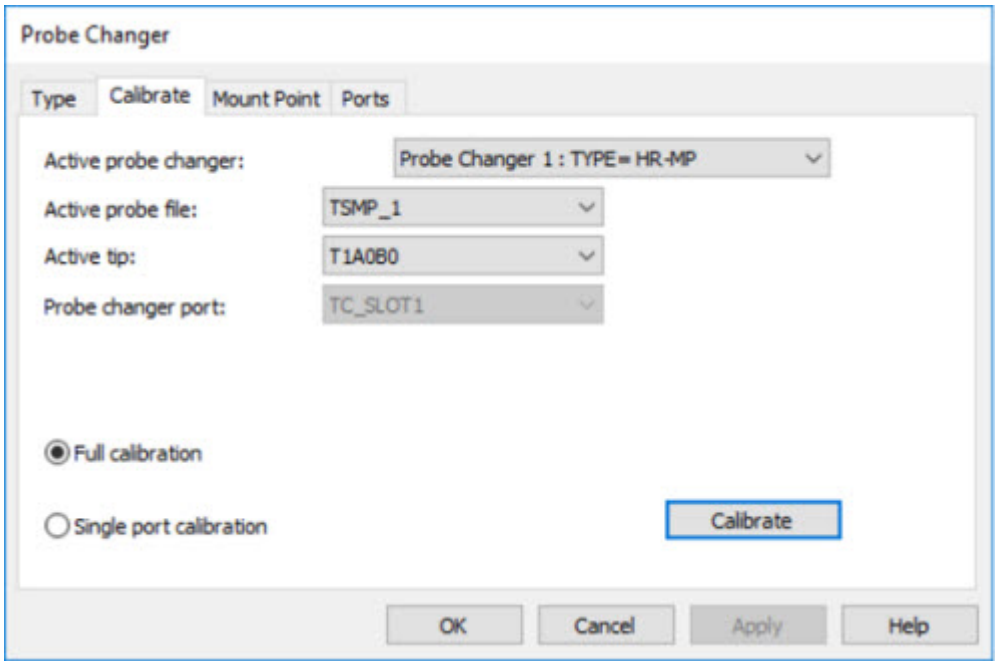
Step 4 - Prepare for Calibration

This step starts the calibration process for the HR-MP (used with TM or THD) Probe Changer.

To start the calibration process, follow these steps:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=HR-MP**:

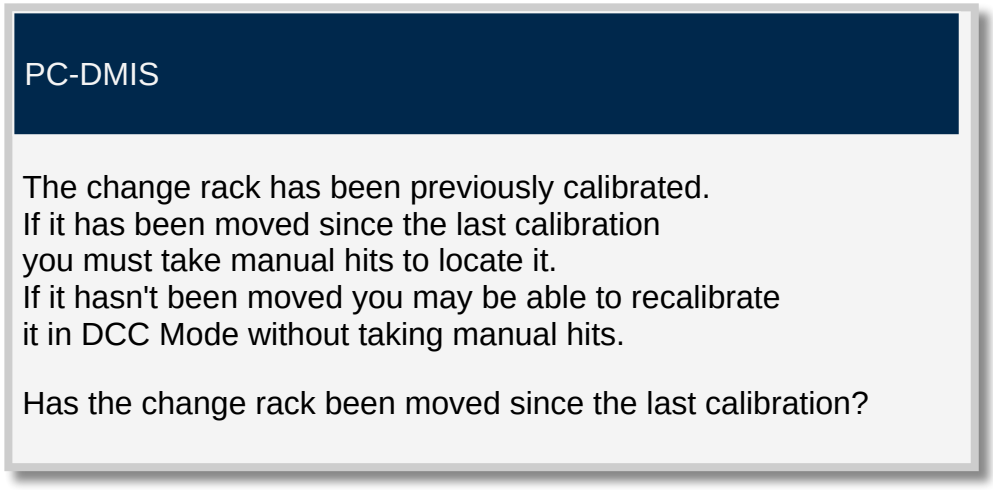


Probe Changer Dialog Box - Calibrate Tab

3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what is to be used for the rack calibration, select the proper probe.

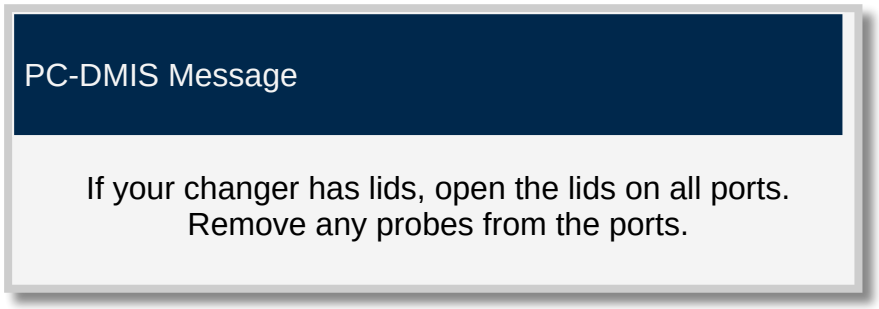
4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not what is to be used for the rack calibration, select the tip ID to be used for calibration. This is usually **T1A0B0**.

5. Click **Calibrate**. The following message box appears:



6. If you are calibrating the probe changer for the first time after installation, or if you want to move the probe changer to a new location, click **Yes**. If you have removed and mounted the probe changer at the same location, click **No**. If you click **Yes**, PC-DMIS prompts you to take manual hits. If you click **No**, the probe changer calibration starts in DCC Mode.

7. If you click **Yes**, the following message appears:



8. Open all lids. To open the lid or lids, push them all the way back.

9. With the lid or lids open, remove all of the modules or styli by sliding them forward and out of the ports.

10. Empty the ports of all probes.

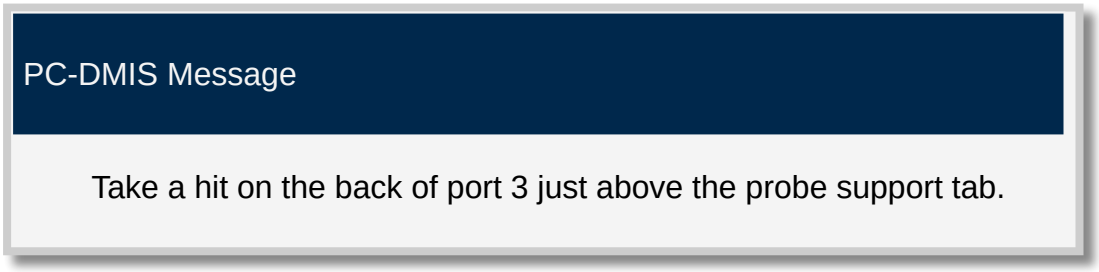
11. When you are ready to take the first measurement, click **OK**.

In the next step, you take the manual hits.

Step 5 - Take the Manual Hits

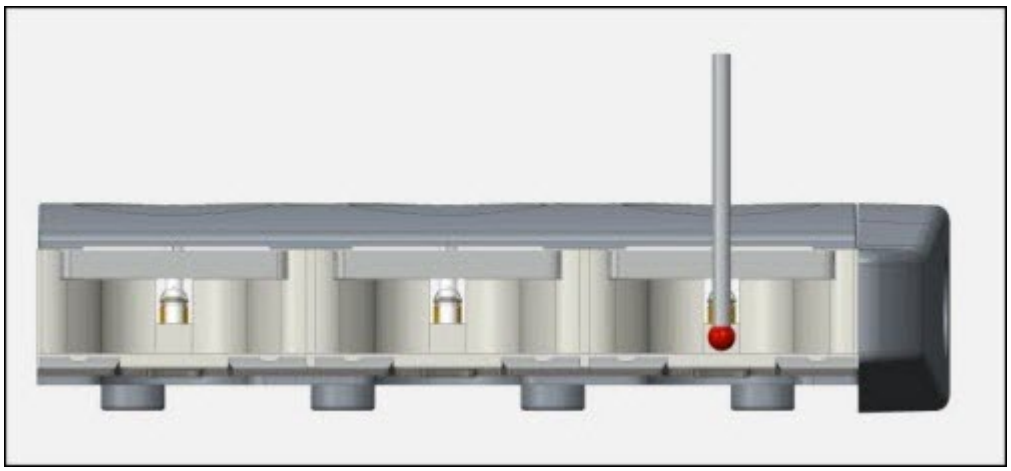
PC-DMIS moves you through the process of measuring four manual hits through a series of message boxes. You will take the first hit for the HR-MP (used with TM or THD) Probe Changer on the back surface of the last port (for example, port 3).

The following message prompts you to open all the lids and take the first hit:

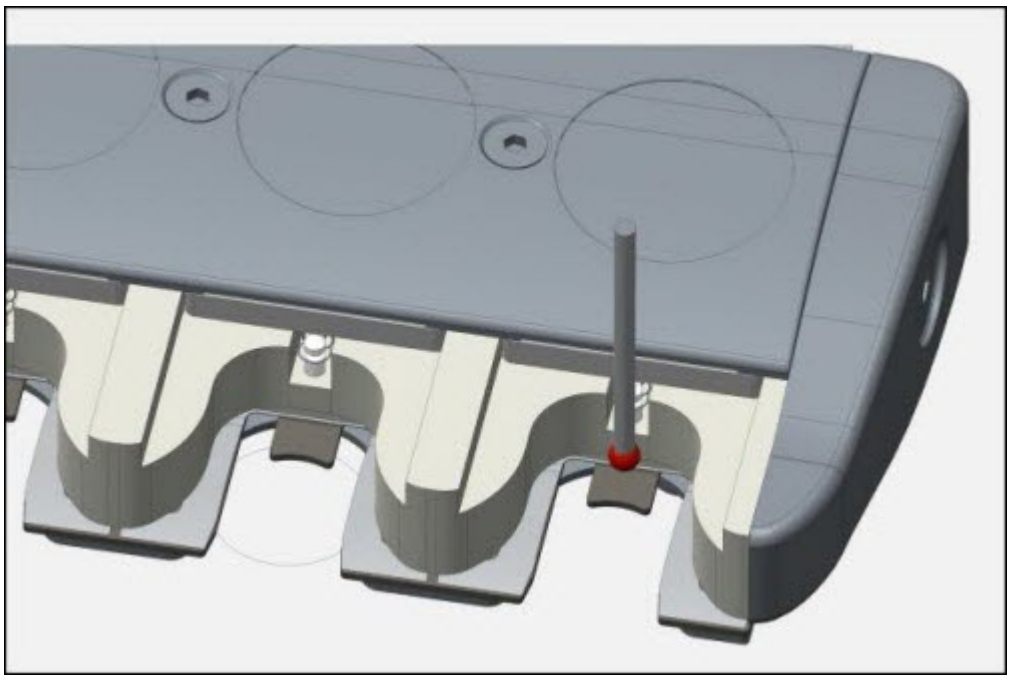


1. When you are ready to take the first measurement, click **OK**.

2. Use your machine's jog box to take the first manual hit on the back surface of port 3 and just above the probe support tab, as shown below:



View of the first hit on the back surface of port 3



View of the first hit on the back surface of port 3 and above the probe support tab

3. Repeat the step 1 and 2 as prompted for all the ports.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

In the next step, you take the manual hit on the top of the probe support tab.

Step 6 - Take the Manual Hit on Top of the Probe Support Tab

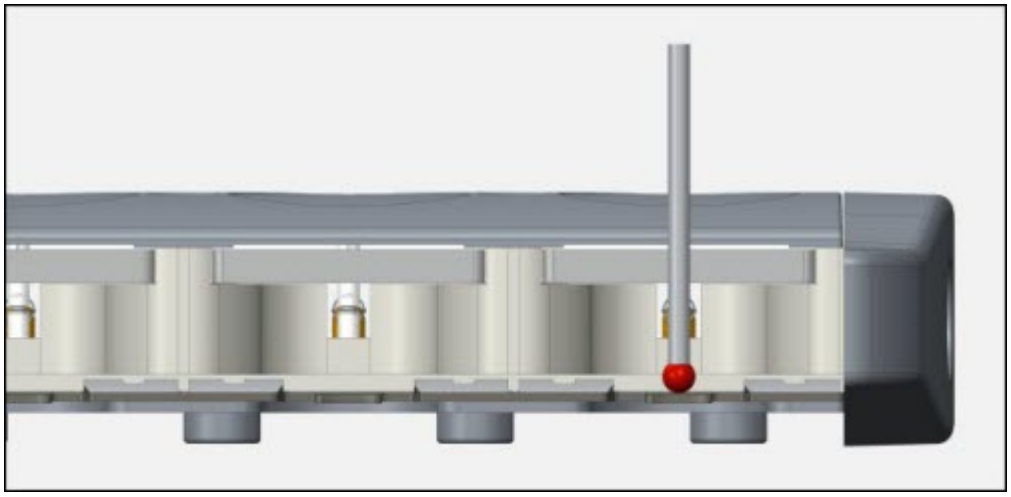
When the following message box appears, you will take the manual hit for the HR-MP (used with TM or THD) Probe Changer on the top of the probe support tab in port 1.

PC-DMIS Message

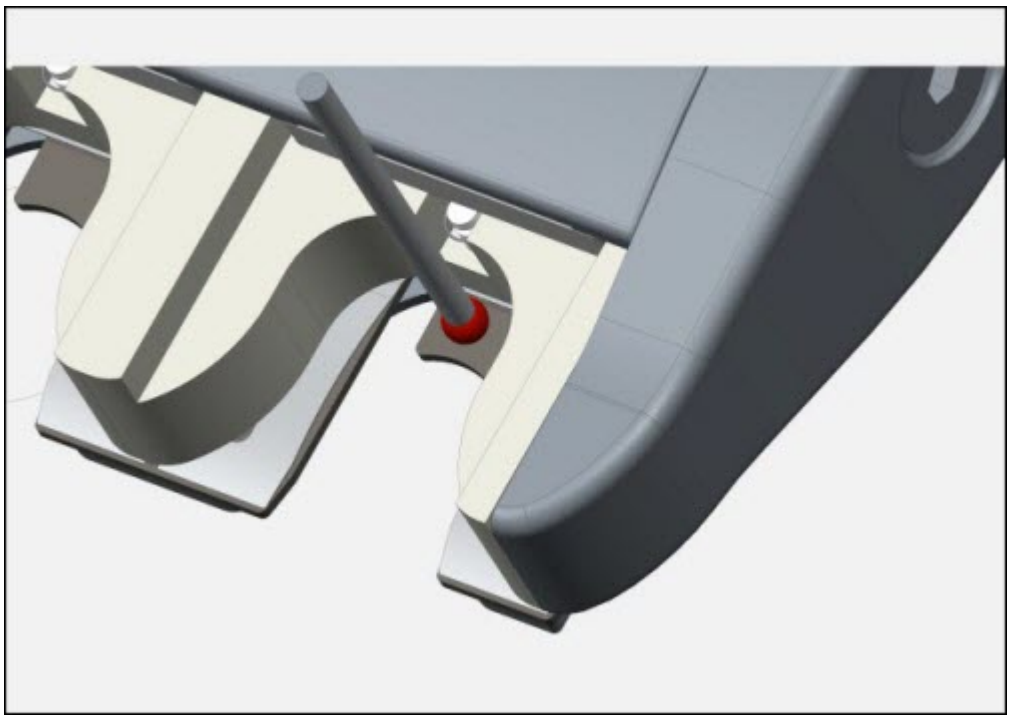
Take a hit on top of the probe support tab in the back of port 1.

After completing the hit, DCC motion will begin
for the rest of the measurement.

1. When you are ready to take the hit, click **OK**.
2. Use your machine's jog box to measure the hit on the top surface of the probe support tab in the back of the port:



View of the fourth hit on the top surface of the probe support tab



View of the fourth hit on the top surface of the probe support tab in the back of the port

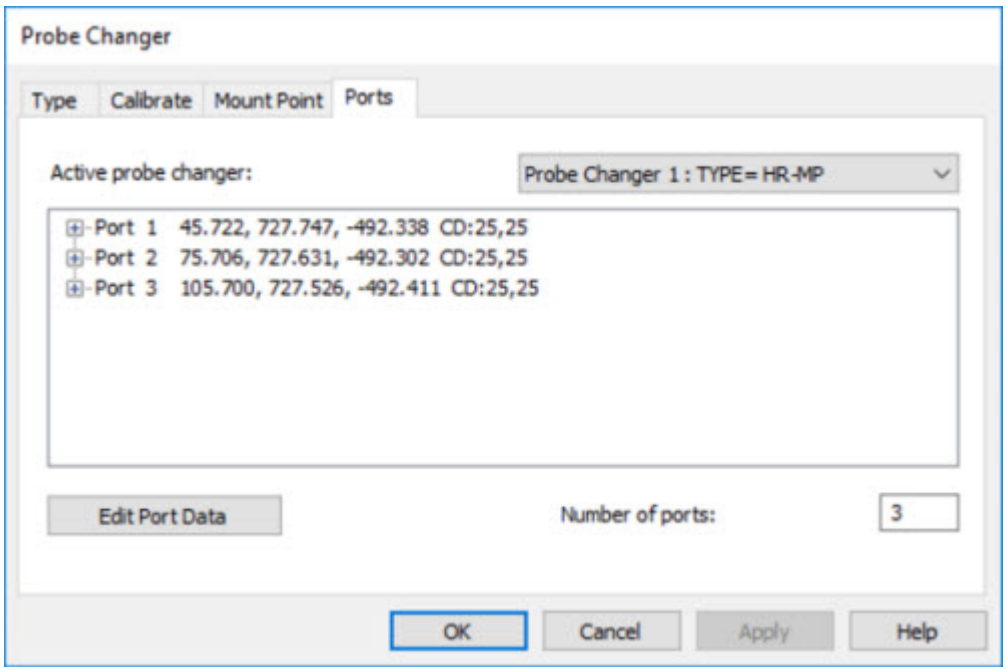
> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

3. Click **OK**. The system goes into DCC Mode and measures a series of hits required to accurately locate and orient the rack.

In the next step, you review the calibration results.

Step 7 - Review the Calibration Results

1. When the calibration of the HR-MP (used with TM or THD) Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Ports tab of Probe Changer dialog box showing the calibration results.

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.
    - This rack is not necessarily aligned parallel to any axis of the CMM.
    - The Z values should be nearly identical since the ports are all at the same height.

The results during measurement routine execution are:

1. The probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the "unload" port (the port that used to contain the currently-used probe entity) to drop off the current probe. The current "puck", a conical piece of hardware that connects to the bottom of the probe body, is held in place by the rack while the probe body lifts away to detach.
3. From there, the probe moves over to the next "load" position. The magnetic connection automatically engages to load the new module.
4. The probe then moves back to the rack's mount point.
5. From there, it continues to measure.

Next, you can choose to recalibrate all ports in DCC Mode, or recalibrate a single port.

Recalibrate the Ports

After you complete a full calibration of the HR-MP (used with TM or THD) Probe Changer, you can choose to recalibrate all ports in DCC Mode, or recalibrate a single port.

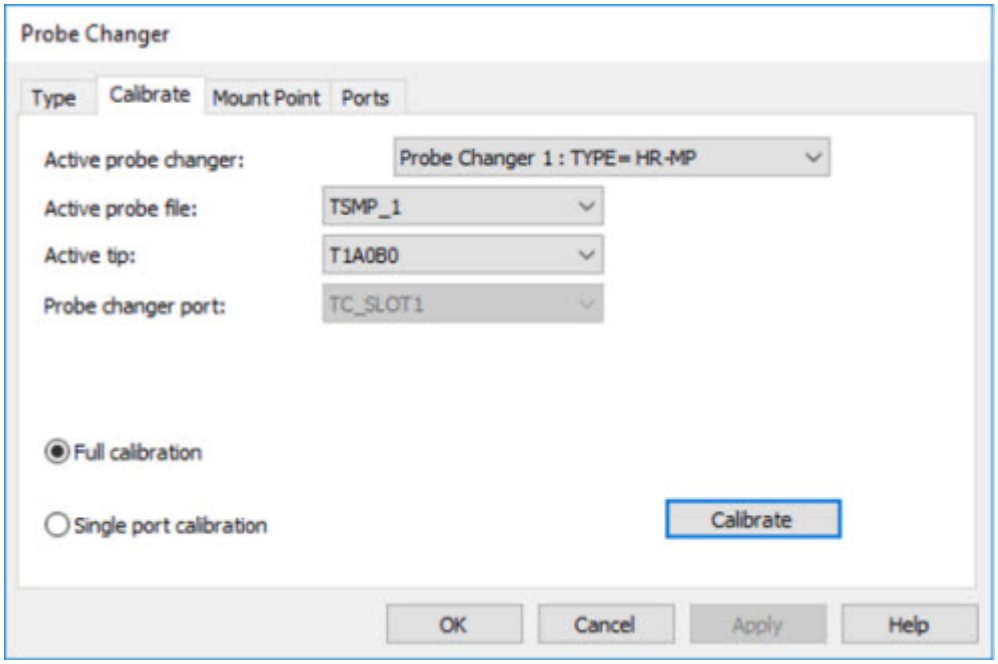
WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Recalibrating All Ports in DCC Mode

If you have removed and mounted the probe changer at the same location, you will be able to recalibrate in DCC Mode.

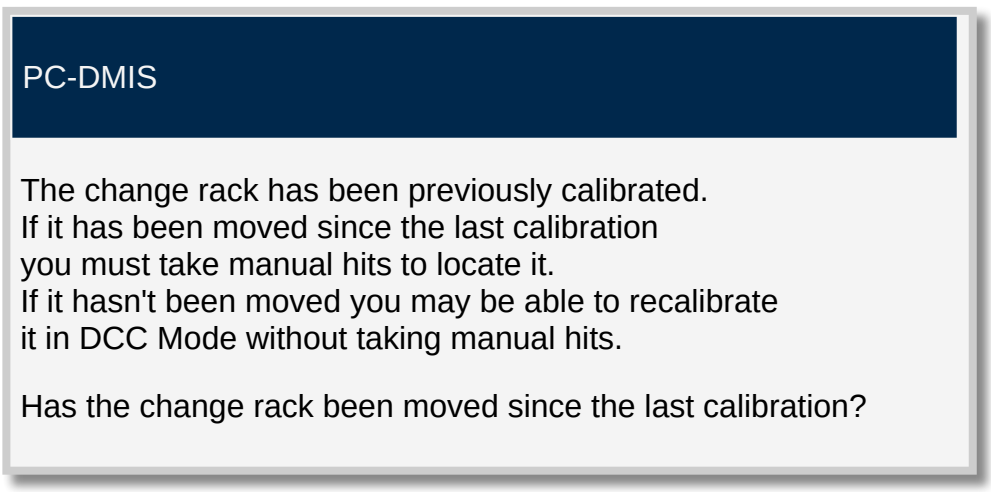
To recalibrate all ports in DCC Mode, follow these steps:

1. Select the **Full calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). (If you did not complete a full calibration, the option is not available.)

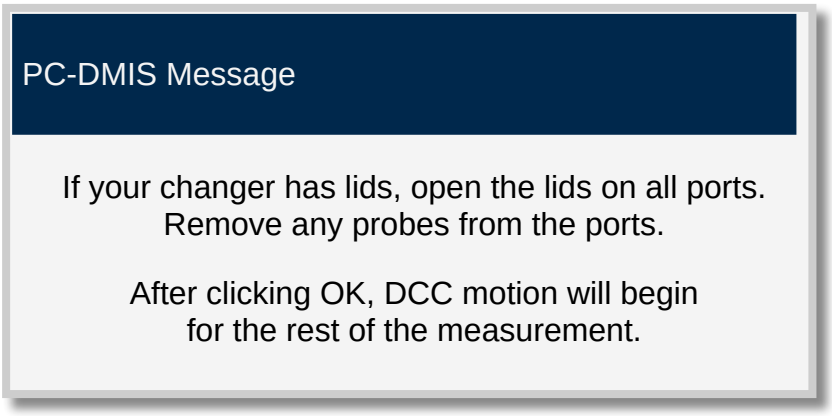


Probe Changer Dialog Box - Calibrate Tab

2. To begin the calibration, click **Calibrate**. A message prompts you if the change rack has been moved since the last calibration:



3. Click **No**. The following message appears:

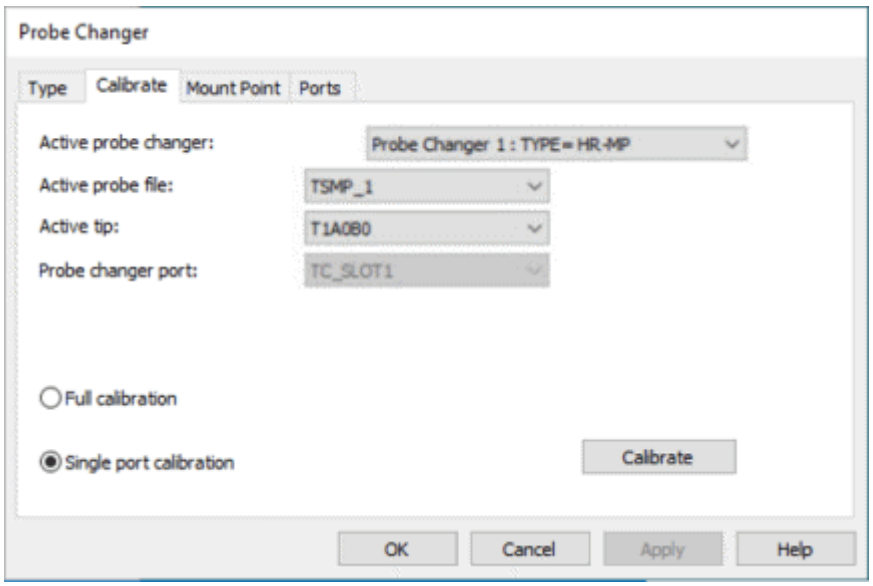


4. Open all lids. To open the lid or lids, push them all the way back.
5. With the lid or lids open, remove all of the modules or styli by sliding them forward and out of the ports.
6. Empty the ports of all probes.
7. Click **OK**. The system goes into DCC Mode and measures a series of hits required to accurately locate and orient the rack.

Recalibrating a Single Port

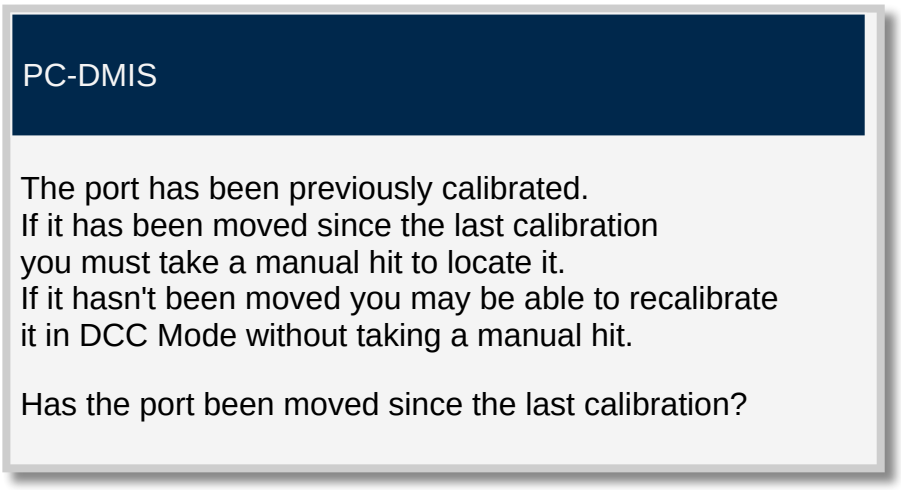
To recalibrate a single port, follow these steps:

1. Select the **Single port calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). (If you did not complete a full calibration, the option is not available.)
2. When you select the **Single port calibration** option, the **Probe changer port** list becomes available. Select the port that you want to calibrate. For example:

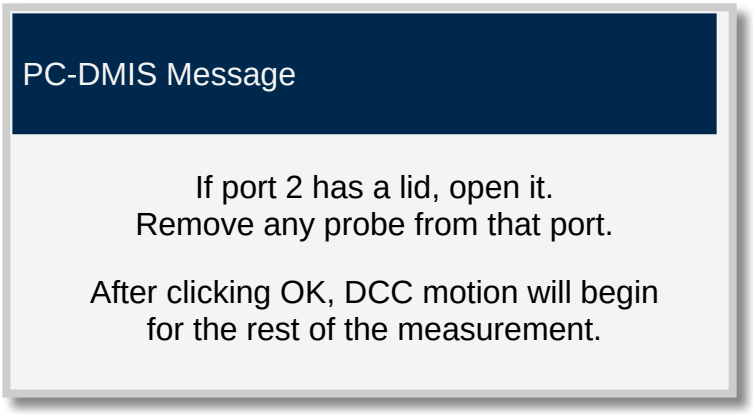


Probe Changer Dialog Box - Calibrate Tab

3. To begin the calibration, click **Calibrate**. A message prompts you to take a hit on the top left surface of the port that you selected:



4. Click **No**. The following message appears:



   a. Open the lid. To open the lid, push it all the way back.

   b. With the lid open, remove the module or styli by sliding it forward and out of the port.

   c. Empty the port of its probe.

   d. Click **OK**. The system goes into DCC Mode and measures the hit.

Calibrating the HR-MS or HR-X1 Probe Changer

This section describes how to calibrate the HR-MS or HR-X1 Probe Changer. The process discussed here is specific to the HR-MS and HR-X1 Probe Changer.

You can calibrate the probe changer in the following ways:

- When you calibrate the probe changer for the first time, you need to take manual hits.

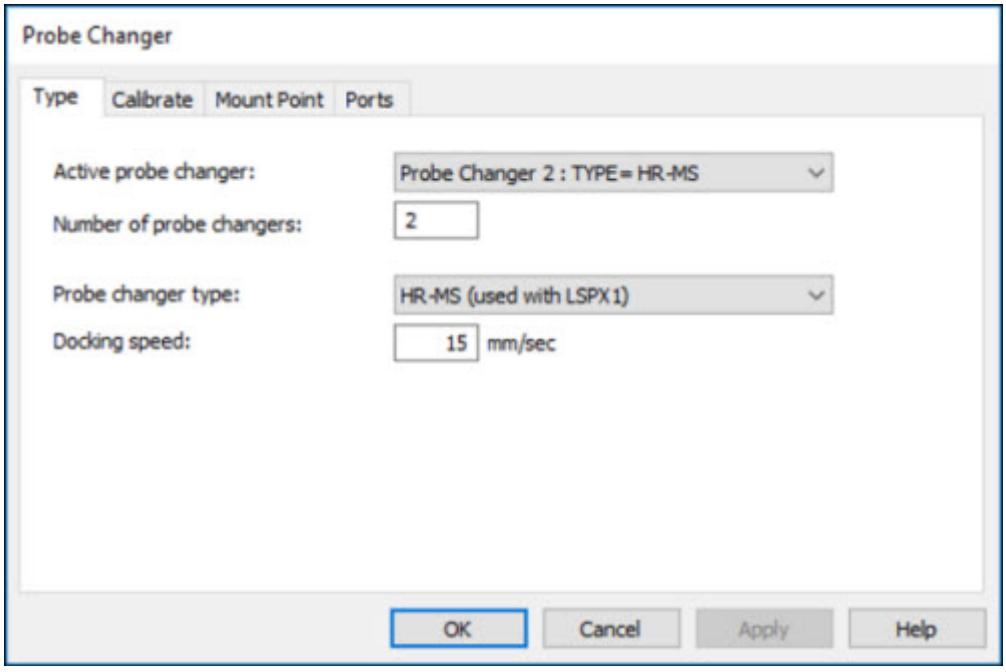
- When the probe changer is removed and mounted at the same location, you can select to calibrate the probe changer in DCC Mode without taking manual hits.
- When the probe changer is removed and relocated on the CMM table, you can follow the same steps when you calibrate for the first time.
- The calibration of a single port is available in DCC Mode or Manual Mode.

To calibrate your probe changer:

Step 1 - Select the HR-MS or HR-X1 Probe Changer

To select the HR-MS or HR-X1 Probe Changer, follow these steps:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **HR-MS** or **HR-X1**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** list, select the list item that represents the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.
6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

> To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

> For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

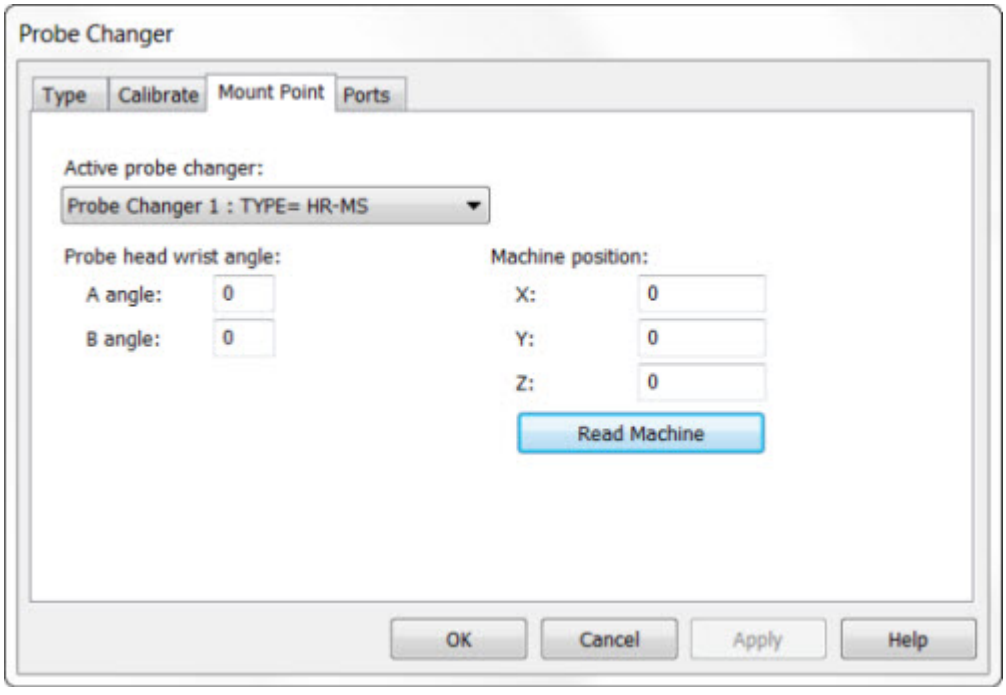
Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).

2. Depending on which probe changer you are calibrating, in the **Active probe changer** list, select **TYPE=HR-MS** or **TYPE=HR-X1**:



Probe Changer Dialog Box - Mount Point Tab

3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use a calibrated probe rotation that ensures that the probe can move in and out of the probe rack during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

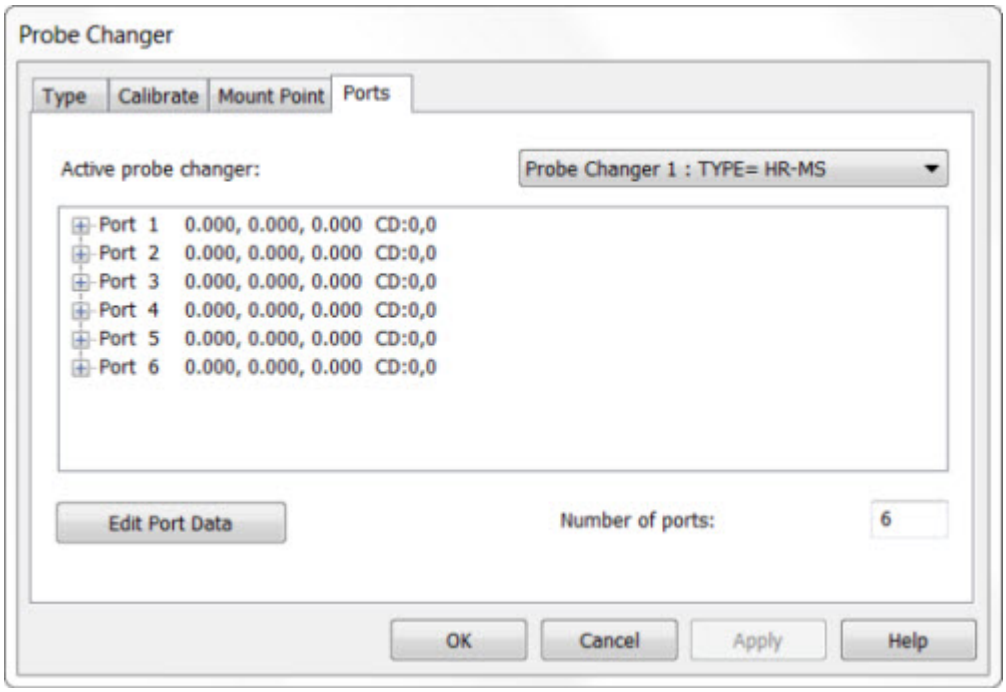
In the next step, you define the ports.

Step 3 - Define the Ports

The final step prior to rack calibration is to define the ports on the rack that you are calibrating.

To define the ports for your HR-MS or HR-X1 Probe Changer, follow these steps:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).
2. In the **Active probe changer** list, select **TYPE=HR-MS** or **TYPE=HR-X1**:



Probe Changer Dialog Box - Ports Tab

3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

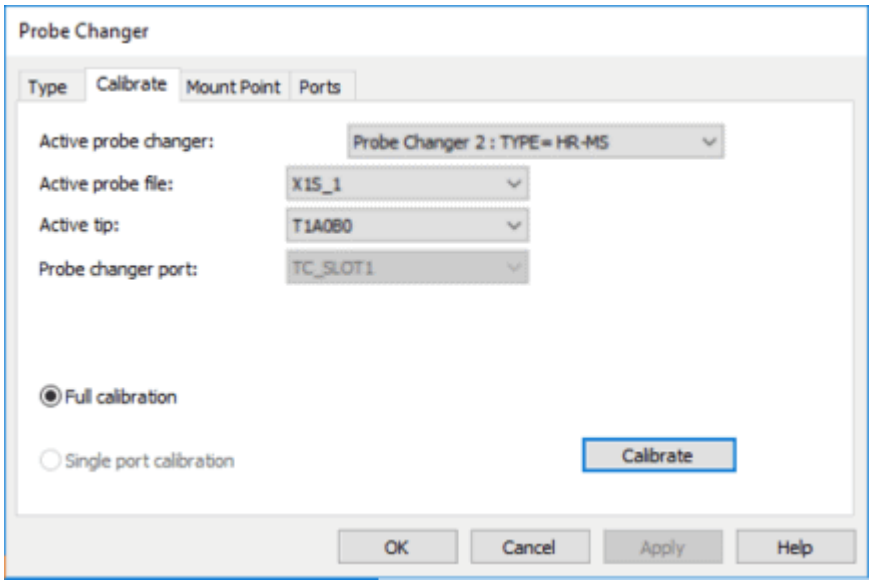
Step 4 - Prepare for Calibration

This step starts the calibration process for the HR-MS or HR-X1 Probe Changer.

To start the calibration process, follow these steps:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).

2. Depending on which probe changer you are calibrating, in the **Active probe changer** list, select **Probe Changer 1: TYPE=HR-MS** or **TYPE=HR-X1**:



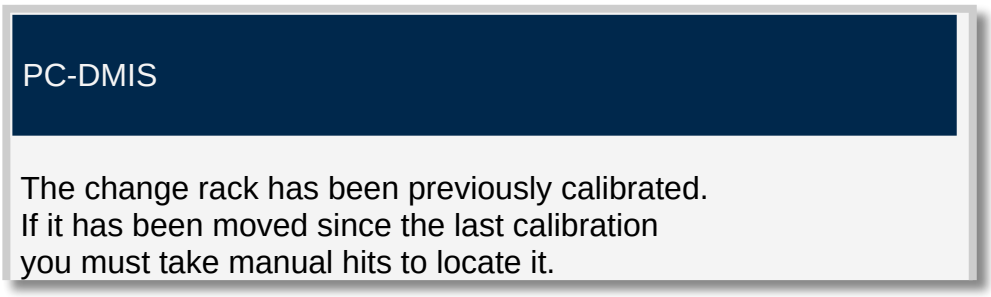
Probe Changer Dialog Box - Calibrate Tab

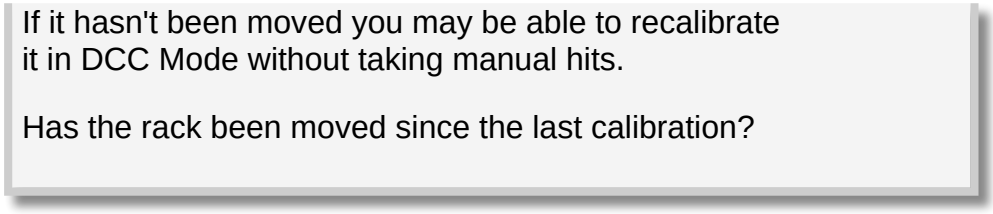
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not the correct probe for the rack calibration, select it from the list.

4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not the correct tip for the rack calibration, select the correct tip ID from the list. This is usually the tip corresponding to the angles defined on the **Mount Point** tab in Step 2.

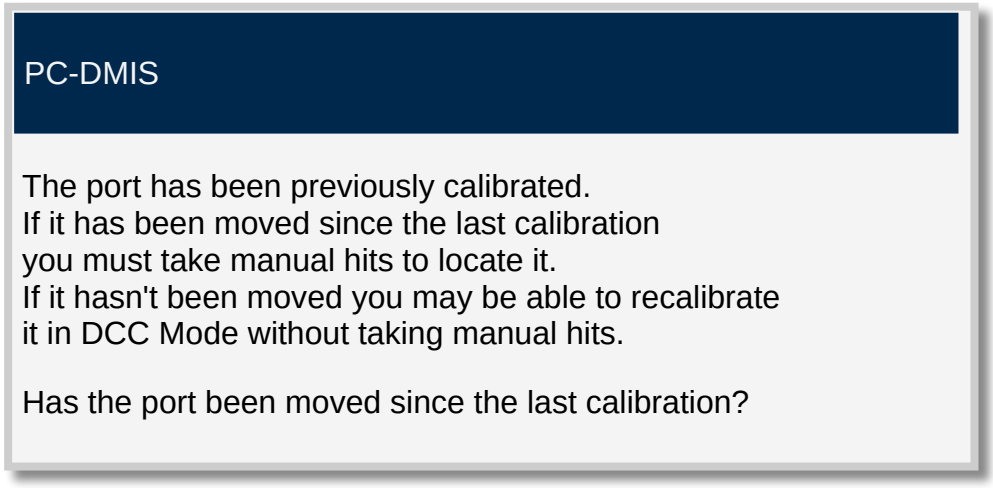
5. Click **Calibrate**. For a full calibration using manual hits, PC-DMIS shows one of the following messages:

   For full calibration:

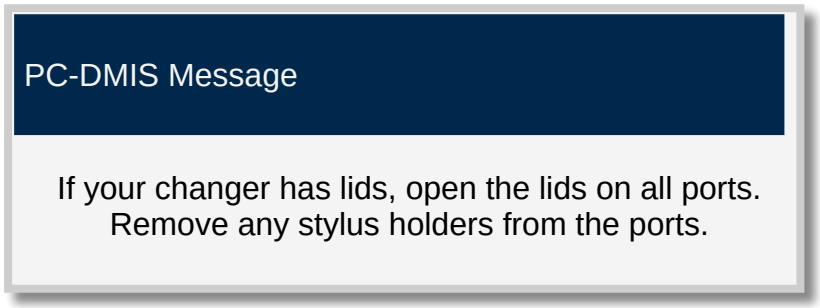




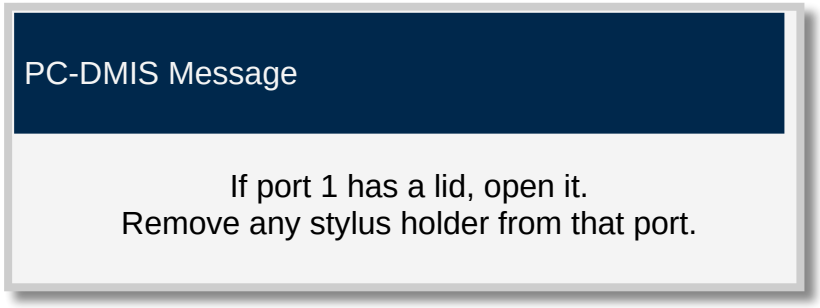
For single-port calibration:



If you click **Yes** on this message, PC-DMIS shows the following message for a full calibration:



If this is a single port calibration, PC-DMIS shows a similar message:



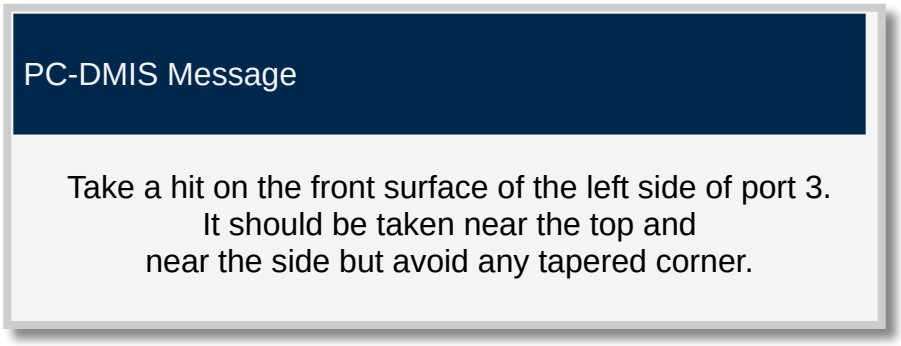
6. Remove the requested modules or styli by sliding them forward and out of the stylus holder.
7. When you are ready to take the first measurement, click **OK**.

In the next step, you take the first manual hit.

Step 5 - Take the Manual Hits

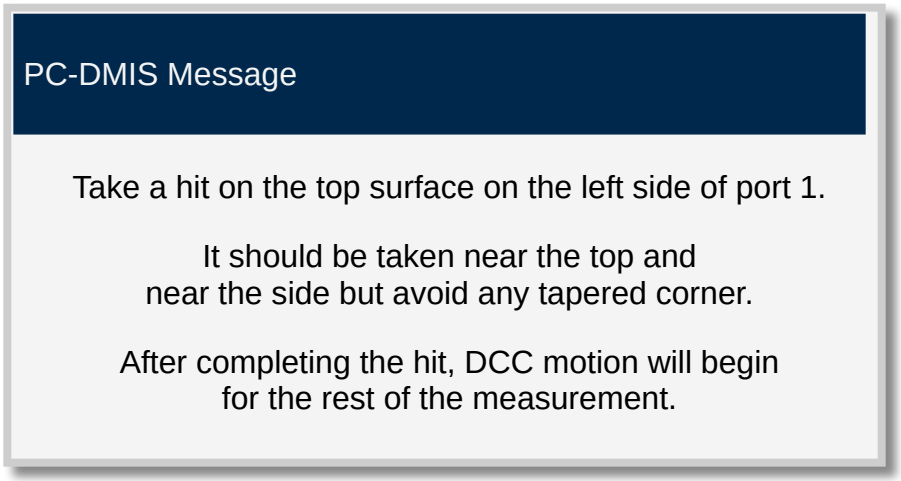
PC-DMIS moves you through the process of measuring the manual hits through a series of message boxes. The number of hits depends on how many ports the probe changer has. For the full calibration, the software requires a single manual hit on the front surface on the left side of each port starting with the last port and ending with the first port. When you complete the manual hits on the front surface of the left side of each port, the software prompts for a final hit on the top of port 1.

If your probe changer has three ports, the prompt for the first hit is:



1. When you are ready to begin, click **OK**.
2. Use your machine's jog box to take each manual hit in the location that the software prompts you.

When you complete all the front surface, left side hits, PC-DMIS prompts you one last time to take a top hit on the left side of the first port:

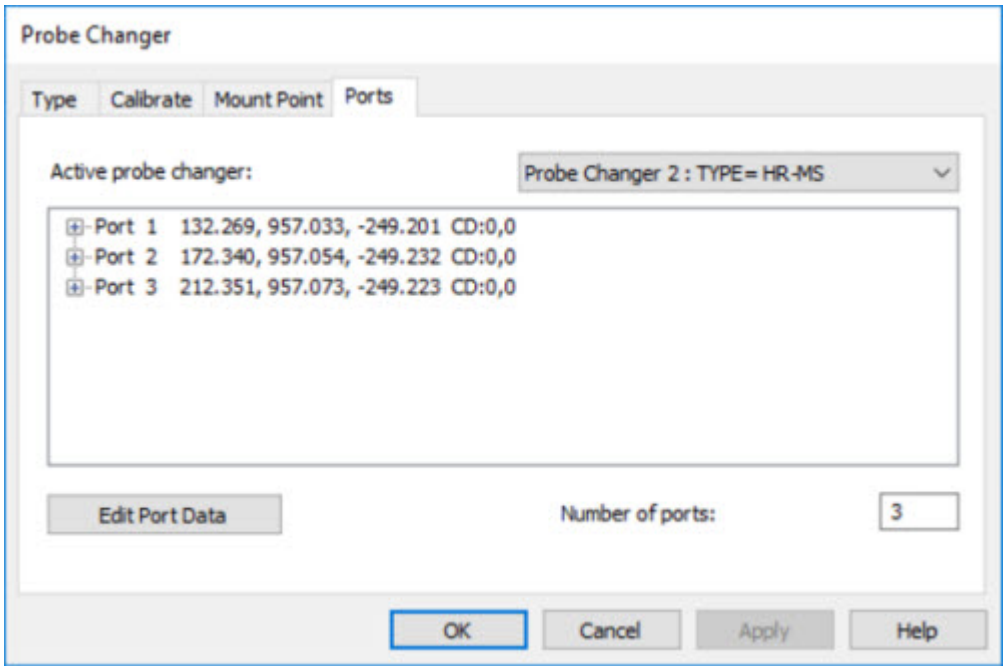


WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

In the next step, you review the calibration results.

Step 6 - Review the Calibration Results

1. When the calibration is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

   - This rack is not necessarily aligned parallel to any axis of the CMM.
   - The Z values should be nearly identical since the ports are all at the same height.

Recalibrate the Ports

After you complete a full calibration of the HR-MS or HR-X1 Probe Changer, you can choose to recalibrate all ports in DCC Mode or recalibrate a single port.

Recalibrating All Ports in DCC Mode

Recalibrating a Single Port

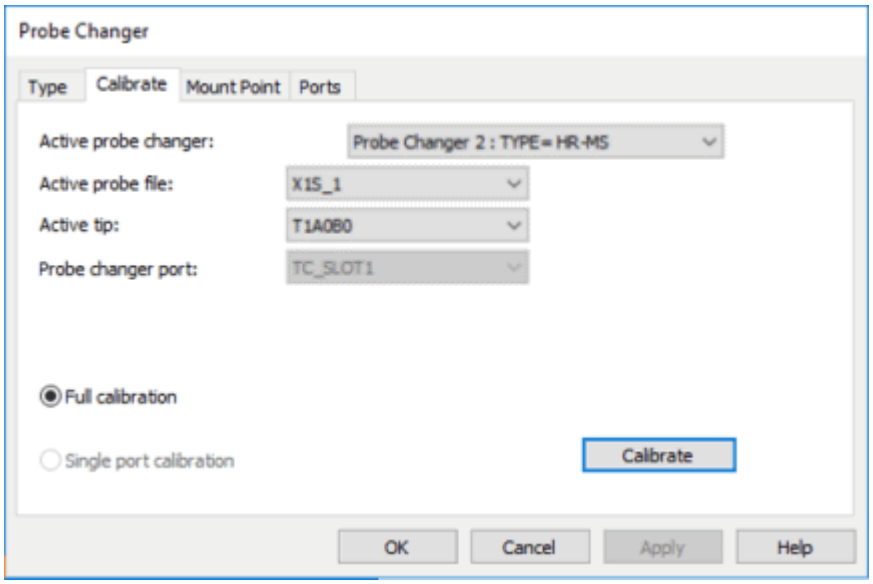
WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Recalibrating All Ports in DCC Mode

If you have removed and mounted the probe changer at the same location, you will be able to recalibrate in DCC Mode.

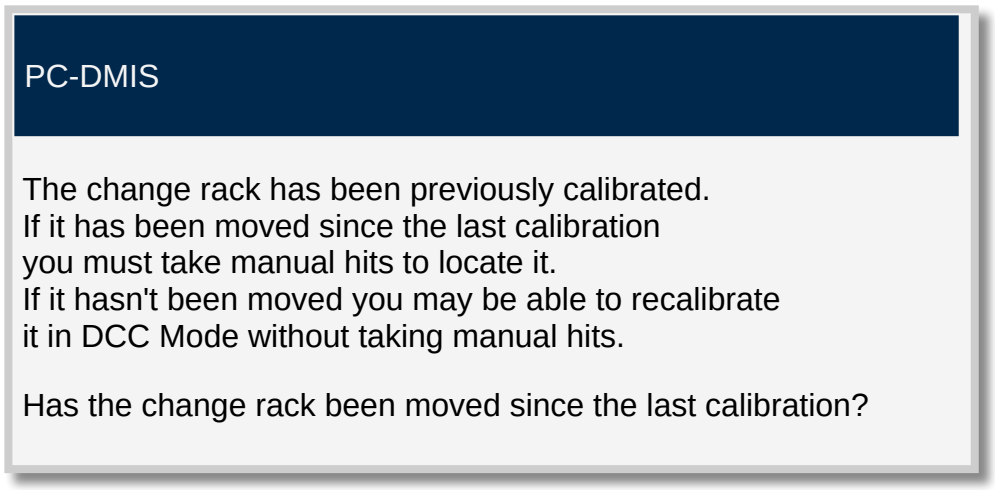
To recalibrate all ports in DCC Mode, follow these steps:

1. Select the **Full calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). (If you did not complete a full calibration, the option is not available.)

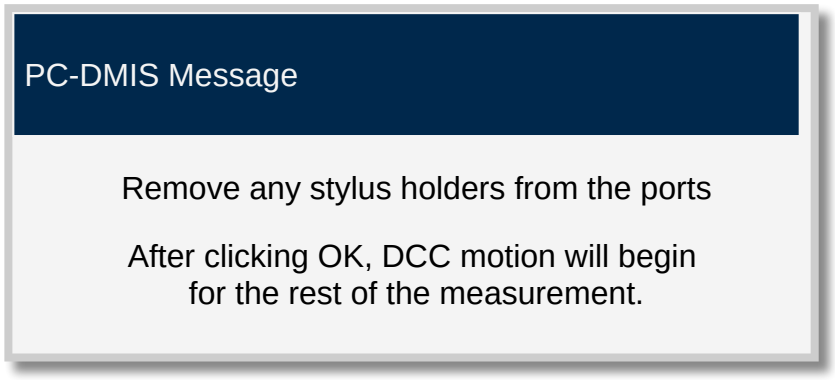


Probe Changer Dialog Box - Calibrate Tab

2. To begin the calibration, click **Calibrate**. A message prompts you if the change rack has been moved since the last calibration:



3. Click **No**. The following message appears:

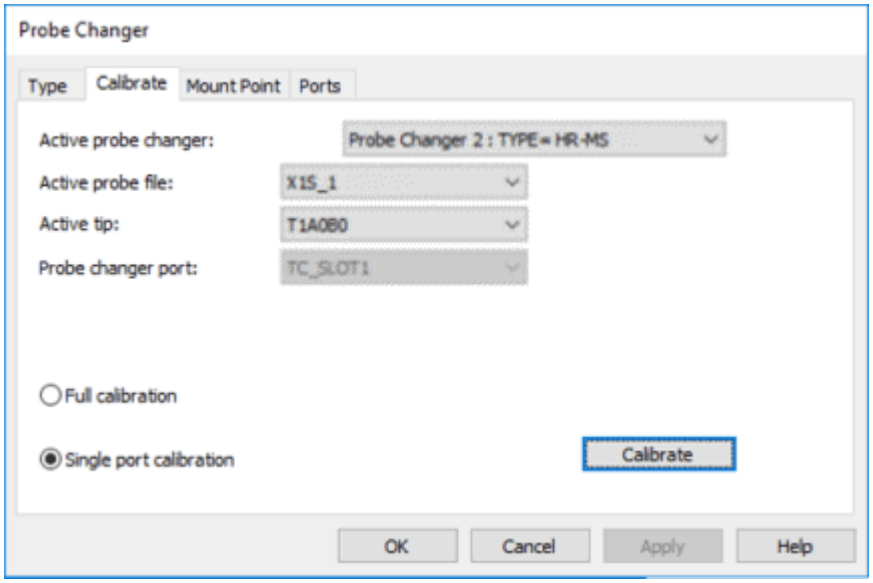


4. Open all lids. To open the lid or lids, push them all the way back.
5. With the lid or lids open, remove the requested modules or styli by sliding them forward and out of the stylus holder.
6. Click **OK**. The system goes into DCC Mode and measures a series of hits required to accurately locate and orient the rack.

Recalibrating a Single Port

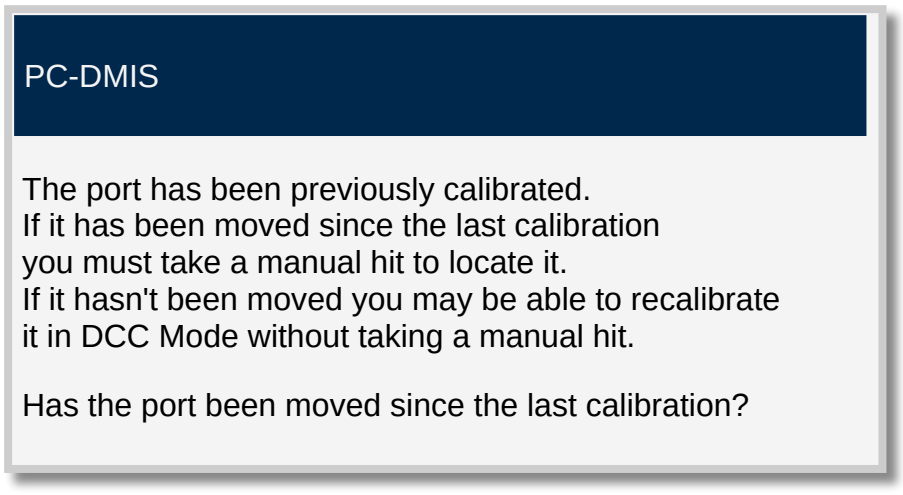
To recalibrate a single port, follow these steps:

1. Select the **Single port calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). (If you did not complete a full calibration, the option is not available.)
2. When you select the **Single port calibration** option, the **Probe changer port** list becomes available. Select the port that you want to calibrate. For example:

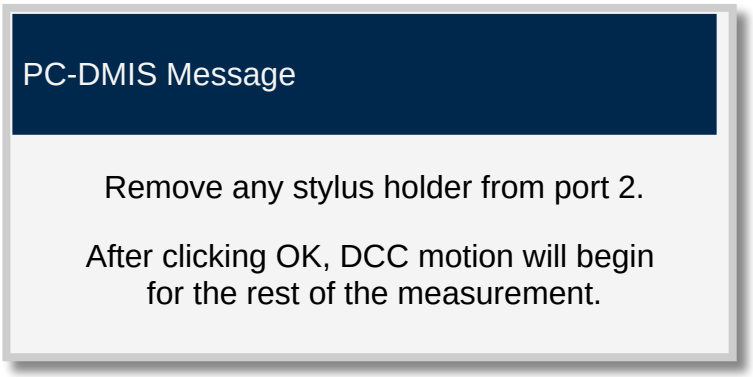


Probe Changer Dialog Box - Calibrate Tab

3. To begin the calibration, click **Calibrate**. A message prompts you to take a hit on the top left surface of the port that you selected:



4. Click **No**. The following message appears:



5. Open the lid. To open the lid, push it all the way back.
6. With the lid or lids open, remove the requested modules or styli by sliding them forward and out of the stylus holder.
7. Click **OK**. The system goes into DCC Mode to complete the measurement.

Defining the I++ Client Probe Changer

You can assign all of the probes that are available for DCC tool changes by the I++ server to the one port that is available for the I++ Client Probe Changer. When loading one of those probes, a request to load it is sent to the I++ server. The I++ server performs the actual change.

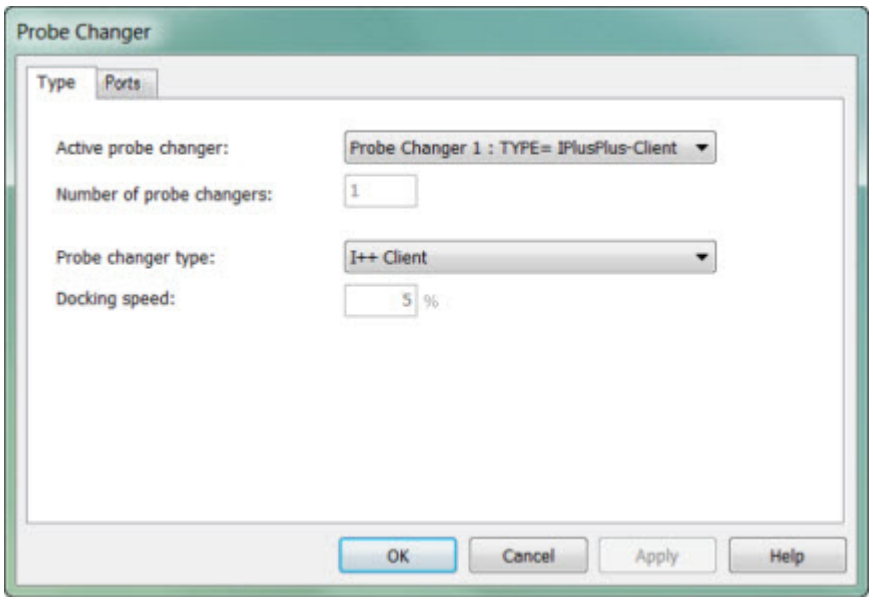
The I++ Client Probe Changer cannot have multiples, and you cannot combine it with other types.

To define the I++ Client Probe Changer:

Step 1 - Select the I++ Client Probe Changer

To select the I++ Client Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **I++ Client**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** box, select **TYPE=IPlusPlus-Client**.
5. Ensure that the value of "1" appears in the **Number of probe changers** box.
6. Note that you cannot change the value that appears in the **Docking speed** box.
7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

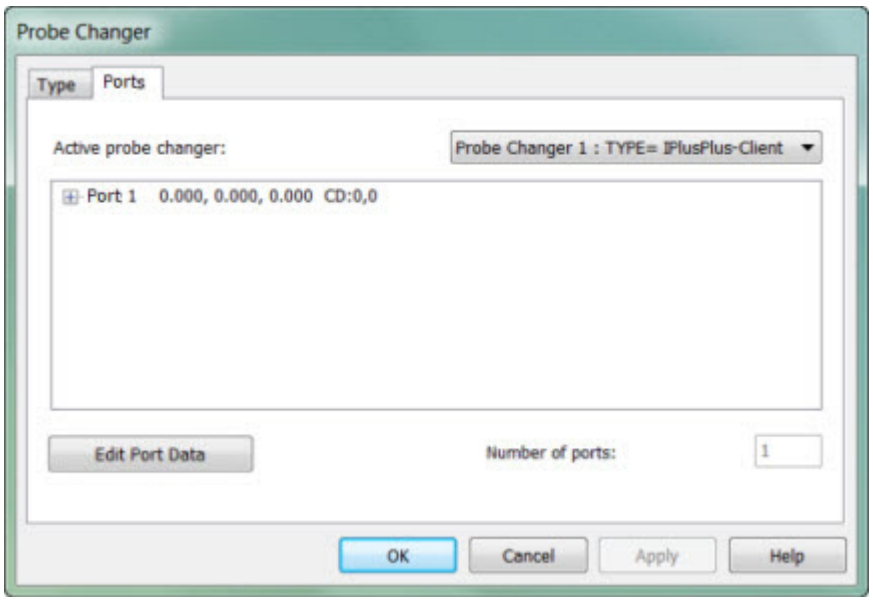
In the next step, you assign the probes to the port.

Step 2 - Assign the Probes to the Port

The **Ports** tab for the I++ Client Probe Changer has only one port. You can assign probes to this port, or remove them. You cannot edit the port's data or change the number of ports.

To assign the probes:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

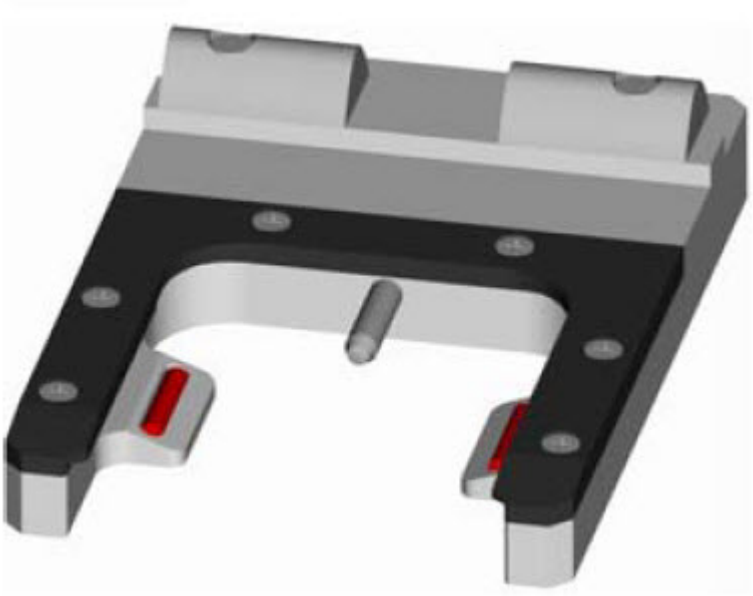
2. Expand the list beside port 1 to open the list of assigned probes.
3. Add the probes that are available for changes by the I++ server. If the I++ server configuration changes, you should update this list by adding any newly available probes and removing any that are no longer available.
4. To save your changes, click **Apply**.

Calibrating the LSPX3 / HR-X (HR-X3-P) Probe Changer

The HR-X3-P and HR-X3-P-SF style ports have considerably different physical constructions and have different dimensional offsets. You cannot mix-and-match these port styles.

The calibration process for the LSPX3 / HR-X (HR-X3-P) probe changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the LSPX3 / HR-X (HR-X3-P) probe changer.

No inserts or extensions are used for any of the port positions.



Leitz LSPX3 / HR-X (HR-X3-P) probe changer

You must mount the rack in a horizontal orientation that your sensor can match.

The LSPX3 / HR-X (HR-X3-P) probe changer DOES NOT need to be mounted on the machine table parallel to either the X or Y axis. However, it must be oriented at an angle which matches an available wrist angle for loading and unloading. For mounting instructions, consult the documentation that came with your probe changer.

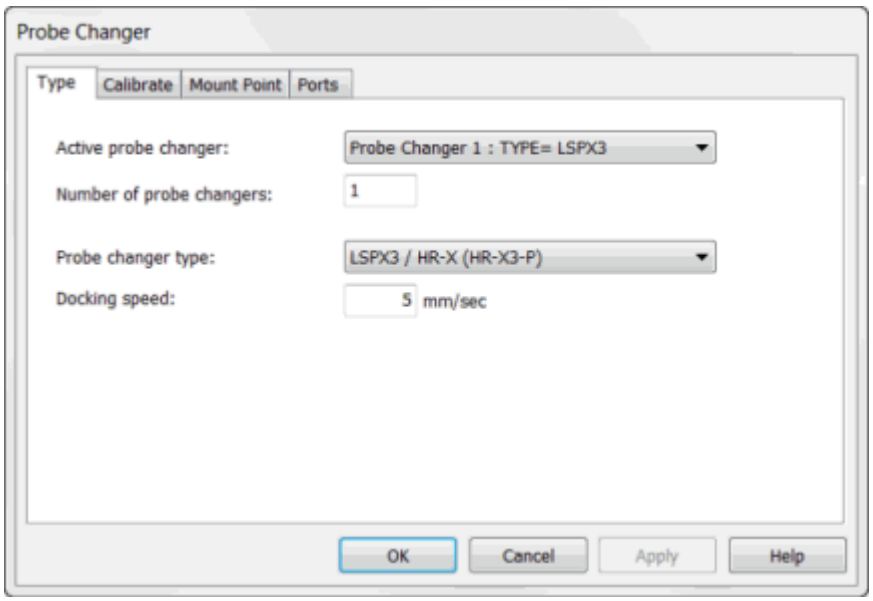
If you use X3T style probes, you need to define the A/B wrist angles that are appropriate to the orientation of the probe changer. This is not required for X3C style probes. For X3C style probes, the rigid mounting of the probe head to the CMM Ram constrains the physical orientation of the probe changer.

To calibrate your probe changer:

Step 1 - Select the LSPX3 / HR-X (HR-X3-P) Probe Changer

To select the LSPX3 / HR-X (HR-X3-P) probe changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **LSPX3 / HR-X (HR-X3-P)**.



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the list item that represents the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer : TYPE= None**.

5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

   For details on controller configuration settings, see the documentation that came with your controller.

Next Step

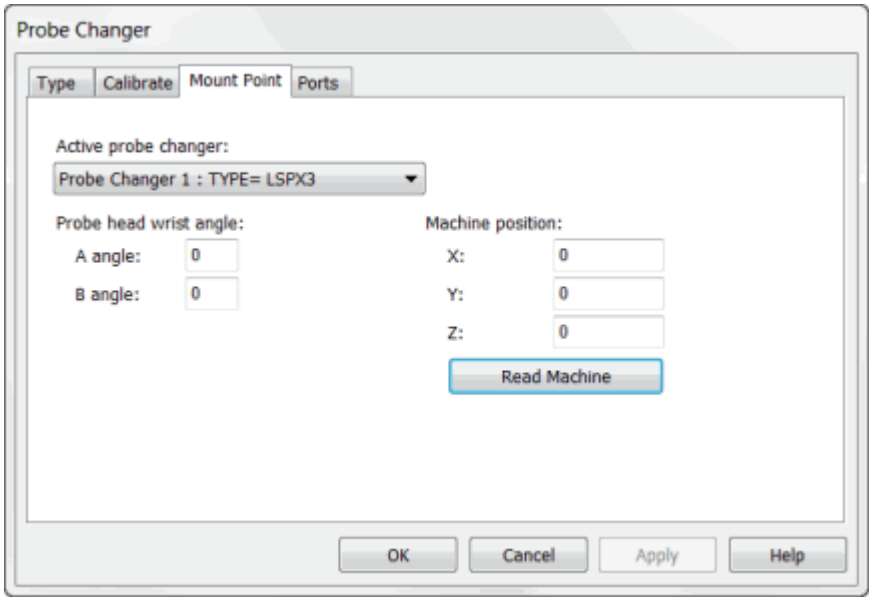
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point

The mount point for the LSPX3 / HR-X (HR-X3-P) probe changer is the location in front of the probe changer that the machine moves to before it picks up or drops off a probe. This should be a location that avoids collisions with the probe changer or part.

To define the mount point for your probe changer:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=LSPX3**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0. You need to use a calibrated probe rotation that ensures that the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

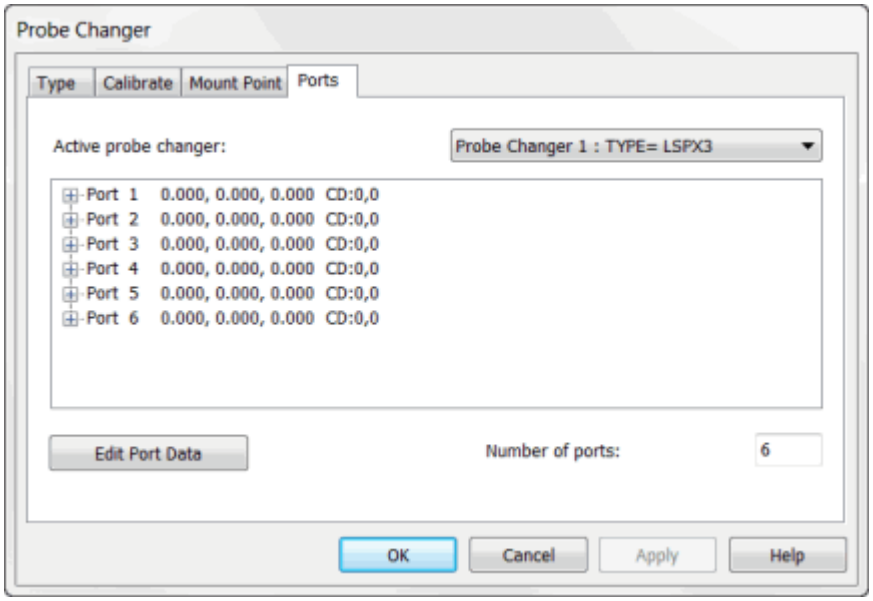
In the next step, you define the number of ports.

Step 3 - Define the Number of Ports

The final step prior to LSPX3 / HR-X (HR-X3-P) probe changer calibration is to define the number of ports.

To define the number of ports:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Ports Tab

2. In the **Active probe changer** list, select **Probe Changer : TYPE= LSPX3**.
3. In the **Number of ports** box, type the number of ports that the probe changer has.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

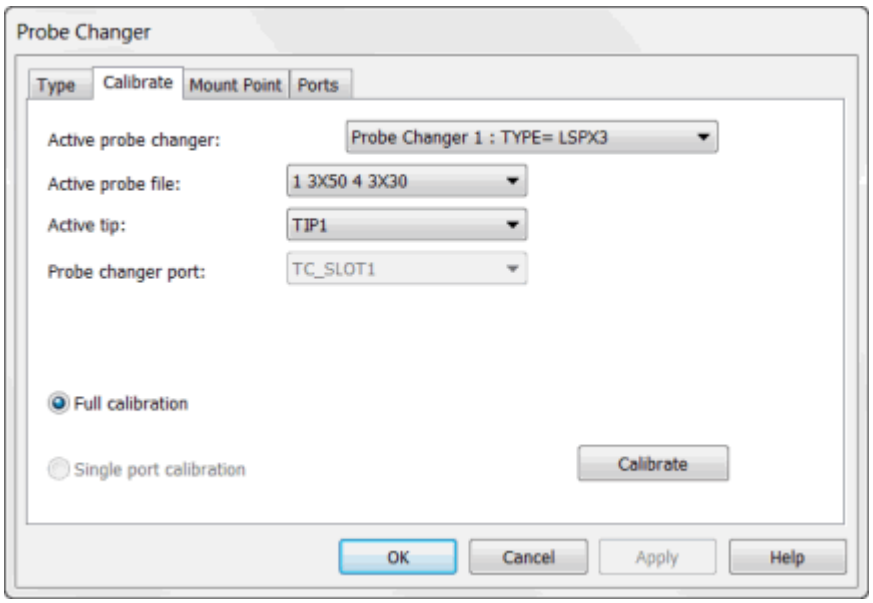
Step 4 - Prepare for Calibration

This step starts the calibration process for the LSPX3 / HR-X (HR-X3-P) probe changer.

You may use a straight-down 5 x 50 mm stylus tip.

To start the calibration process:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer : TYPE= LSPX3**.
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what you will use for the probe changer calibration, select the proper probe.
4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not what you will use for the probe changer calibration, select the tip ID to use for calibration. This is usually **TIP1**.
5. Empty the ports of all probes.
6. Click **Calibrate**.
7. When you are ready to take the first measurement, click **OK**.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

When you start the calibration procedure for the LSPX3 / HR-X (HR-X3-P) probe changer, PC-DMIS shows you a sequence of message boxes that step you through a series of manual hits. You first take one hit on the overhang of the collet, and then three hits on each port. These hits fully define the orientation of the probe changer in the machine's XY plane. You do not need to align the probe changer to any single axis, as the manual hits identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

You take the first hit on top of the clamp for port 1, using the overhang of the collet near the probe pin. The prompt for the first hit on port 1 is:

PC-DMIS Message

Please take a hit on top of the clamp for port 1 using the overhang of the collet near the probe pin

PC-DMIS prompt to take the first manual hit on port 1.

You must take a hit on the top of the first port, using the overhang of the collet. The image below shows the overhang of the collet:

Location of first hit for port 1 on overhang of collet

1. When you are ready to take the first measurement, click **OK**.
2. Use your machine's jog box to take the first manual hit as instructed by PC-DMIS.

In the next step, you define the location of the first port.

Step 6 - Define the Location of the First Port

In this step for the LSPX3 / HR-X (HR-X3-P) probe changer, you define the location of the first port. This message appears:

PC-DMIS Message

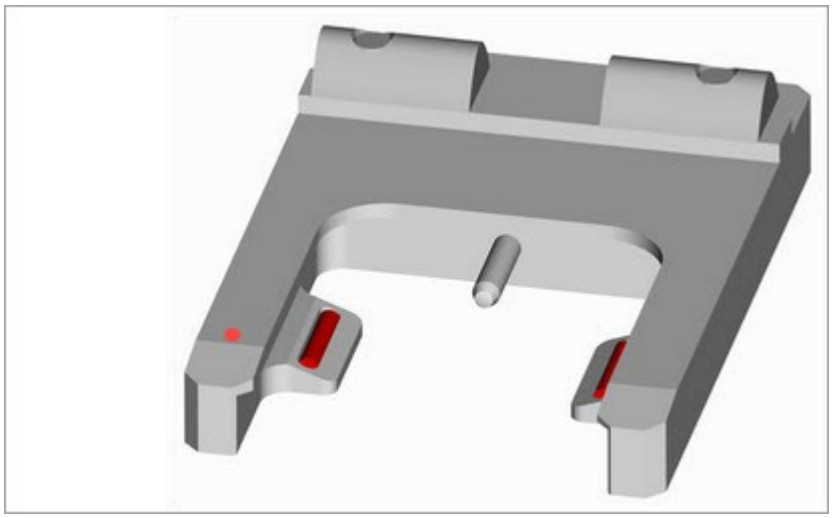
From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the top of the port, then one hit on the external left edge of the port and finally one hit on the external front edge of the port for port 1

PC-DMIS prompt to define the location of the first port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

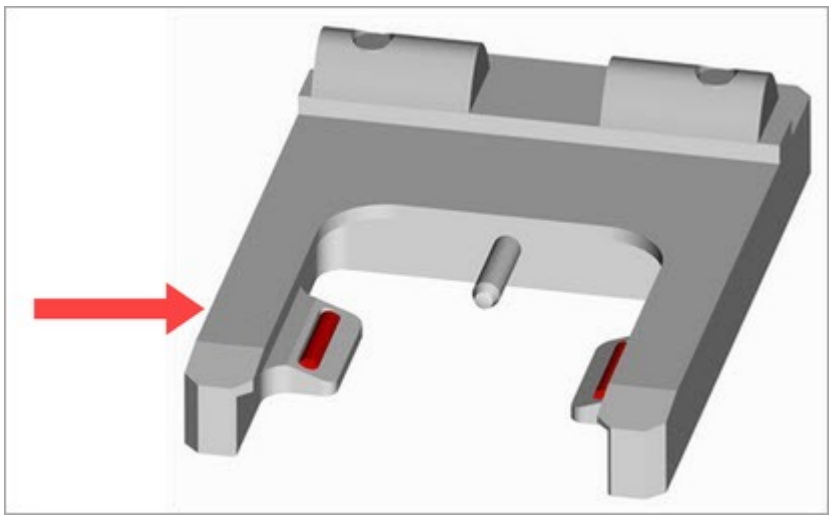
You must take three hits:

1. Take the first hit on the top of the port. The image below shows the location:



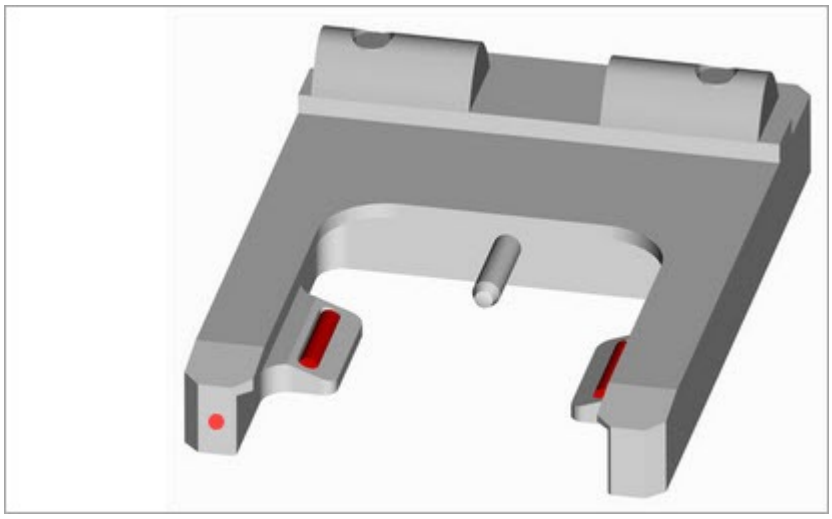
Example showing the first hit on the top of the port.

2. Take the second hit on the external left edge of the port. This face is turned away from you in the image below, so the arrow shows where it is:



Example showing the second hit on the external left edge of the port.

3. Take the third hit on the external front edge of the port. The image below shows the location:



Example showing the third hit on the external front edge of the port.

In the next step, you define the location of the second port and all subsequent ports.

Step 7 - Define the Location of the Second Port

In this step for the LSPX3 / HR-X (HR-X3-P) probe changer, you define the location of the second port and all subsequent ports that are available on your probe changer. This message appears:

PC-DMIS Message

From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the top of the port, then one hit on the external left edge of the port and finally one

hit on the external front edge of the port for port 2

PC-DMIS prompt to define the location of the second port.

To define the locations of the second port and all subsequent ports, follow the procedure described in Step 6 - Define the Location of the First Port. PC-DMIS prompts you for each port in turn, and these hits establish the location of every port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

You take all of these hits in manual mode with your machine's jog box.

In the next step, you review the calibration results.

Step 8 - Review the Calibration Results

1. When the calibration of the LSPX3 / HR-X (HR-X3-P) probe changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port.

2. When you view the results, look for the following.

   - This probe changer is not necessarily aligned parallel to any axis of the CMM.
   - The Z values should be nearly identical since the ports are all at the same height.

   Any significant deviations from these expectations may be due to a bad hit.

The results during measurement routine execution are:

1. The probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.

2. The probe body moves to the mount point and then into the "unload" port (the port that used to contain the currently used probe entity) to drop off the current probe. The current "puck", a conical piece of hardware that connects to the bottom of the probe body, is held in place by the probe changer while the probe body lifts away to detach.

3. From there, the probe moves over to the next "load" position. The magnetic connection automatically engages to load the new module.

4. The probe then moves back to the probe changer's mount point and from there, it continues to measure.

In the next step, you can choose to recalibrate individual ports.

Step 9 - Recalibrate Individual Ports

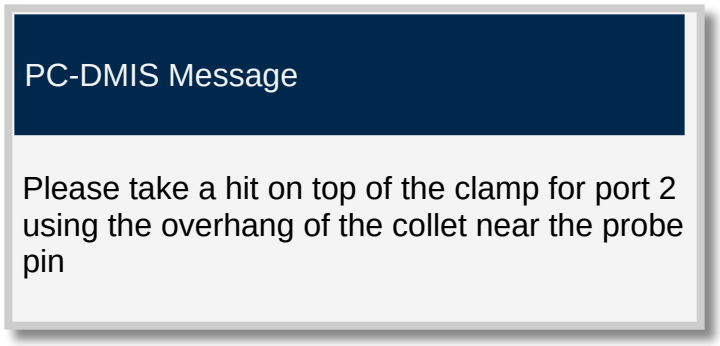
After you complete a full calibration of the LSPX3 / HR-X (HR-X3-P) probe changer, if you suspect there's an issue with the calibration results of a port, you can choose to recalibrate individual ports.

1. Select the **Single port calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This option is only available after you complete a full calibration.

2. When you select the **Single port calibration** option, the **Probe changer port** list becomes available. Select the port that you want to calibrate.

3. Remove any probe or stylus holder from the port that you selected.

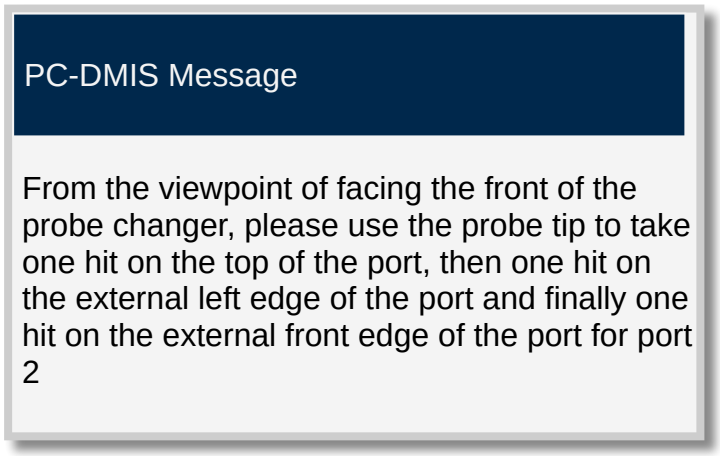
4. To begin the calibration, click **Calibrate**. PC-DMIS prompts you to take a hit on the overhang of the collet for the port that you selected. For example, for port 2:



PC-DMIS prompt to take a hit on the overhang of the collet for port 2.

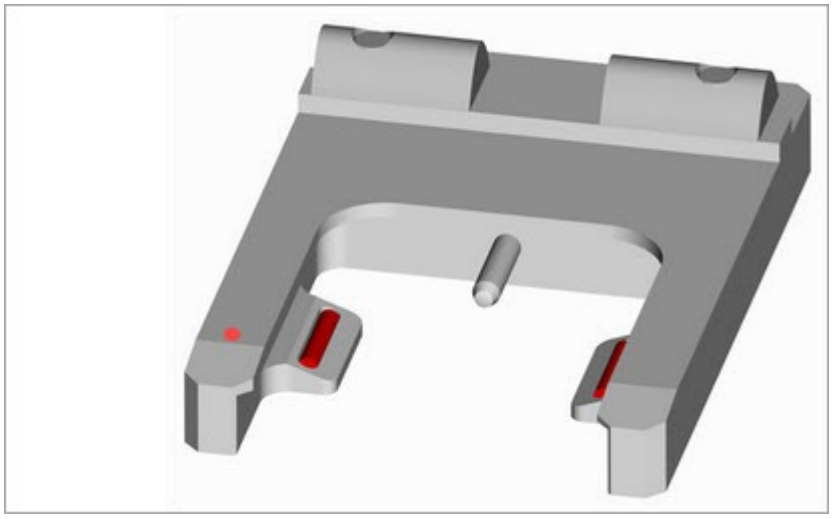
Location of the hit on the overhang of the collet.

5. PC-DMIS then prompts you to take three hits on the port that you selected. For example, for port 2:



PC-DMIS prompt to define the location of port 2.

6. Click **OK**, and then take the three hits in the order that PC-DMIS prompts for them, as described in Step 6 - Define the Location of the First Port: one hit on the top of the port, one hit on the external left edge, and one hit on the external front edge.



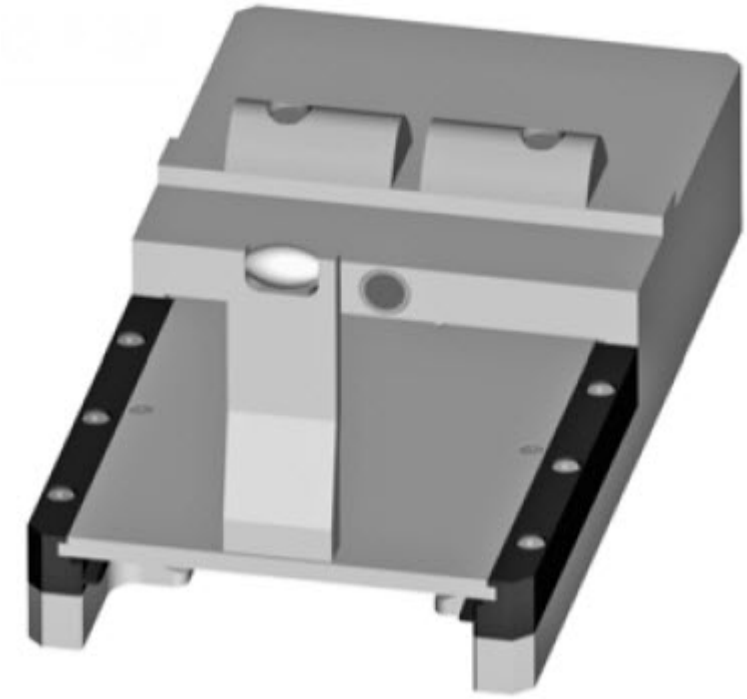
Example showing the first hit on the top of the port.

Calibrating the LSPX3SF / HR-X (HR-X3-P-SF) Probe Changer

The HR-X3-P and HR-X3-P-SF style ports have considerably different physical constructions and have different dimensional offsets. You cannot mix-and-match these port styles.

The calibration process for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the LSPX3SF / HR-X (HR-X3-P-SF) probe changer.

No inserts or extensions are used for any of the port positions.



Leitz LSPX3SF / HR-X (HR-X3-P-SF) probe changer

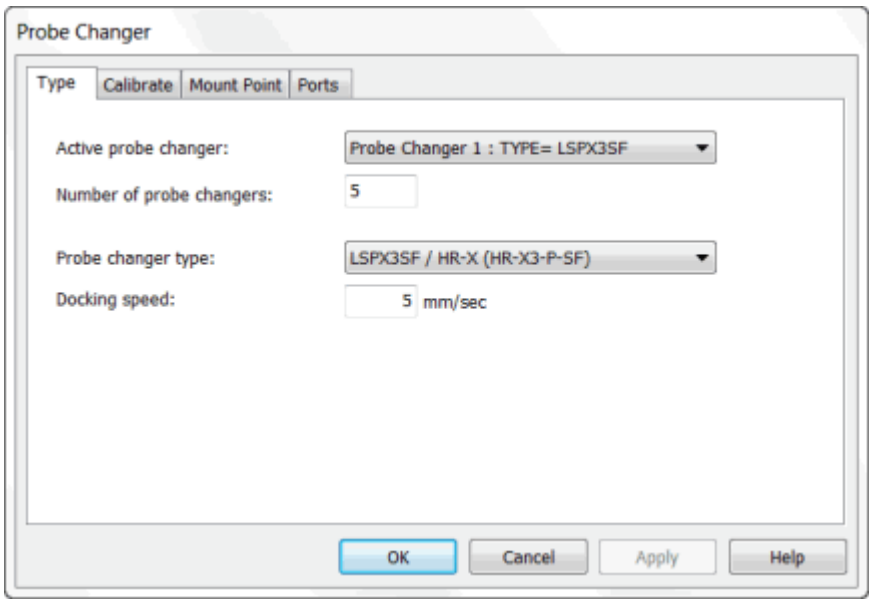
> **!** The LSPX3SF / HR-X (HR-X3-P-SF) stylus changer DOES NOT need to be mounted on the machine table parallel to either the X or Y axis. However, it must be oriented at an angle which matches an available wrist angle for loading and unloading. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the LSPX3SF / HR-X (HR-X3-P-SF) Probe Changer

To select the LSPX3SF / HR-X (HR-X3-P-SF) probe changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **LSPX3SF / HR-X (HR-X3-P-SF)**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the list item that represents the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer : TYPE= None**.

5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

Next Step

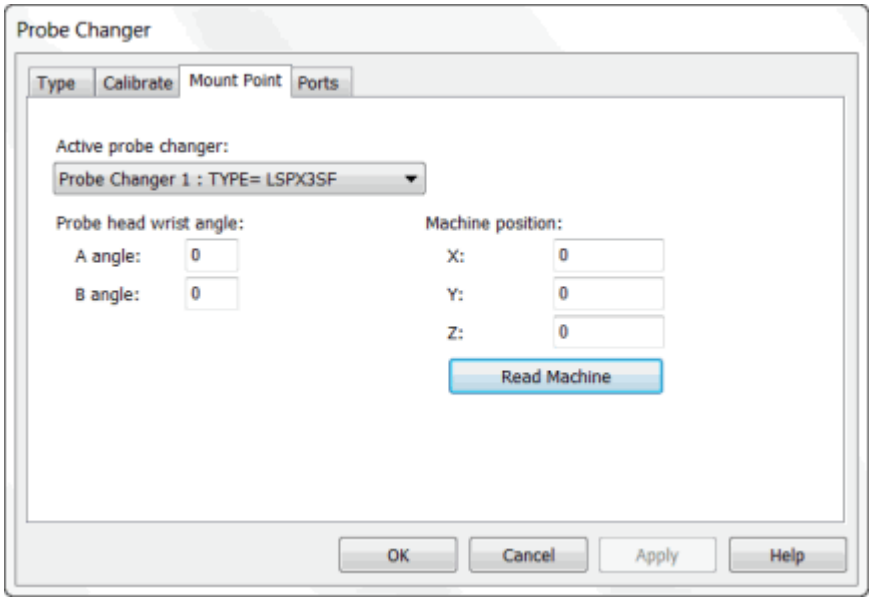
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point

The mount point for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer is the location in front of the probe changer that the machine moves to before it picks up or drops off a probe. This should be a location that avoids collisions with the probe changer or part.

To define the mount point for your probe changer:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=LSPX3SF**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0. You need to use a calibrated probe rotation that ensures that the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

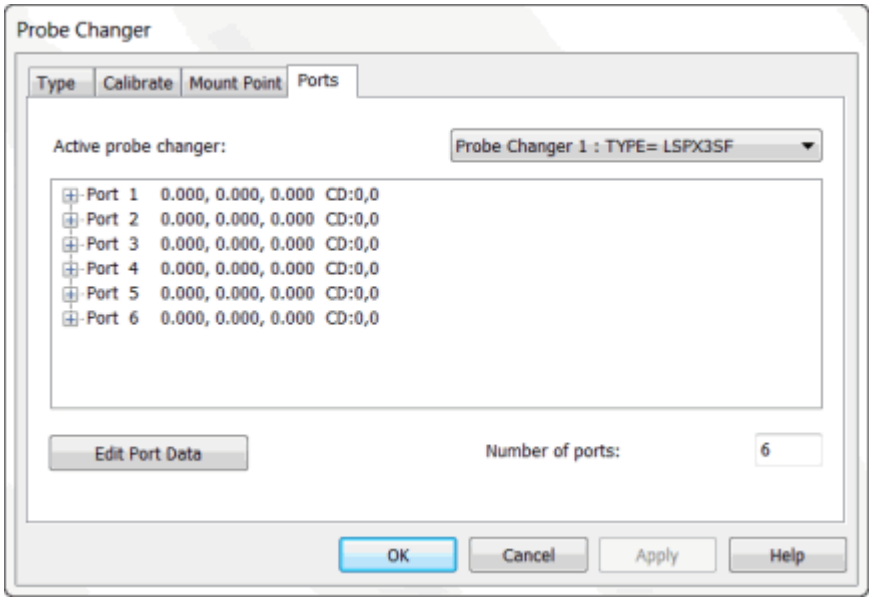
In the next step, you define the number of ports.

Step 3 - Define the Number of Ports

The final step prior to probe changer calibration is to define the number of ports.

To define the number of ports:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

2. In the **Active probe changer** list, select **Probe Changer : TYPE= LSPX3SF.**
3. In the **Number of ports** box, type the number of ports that the probe changer has.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

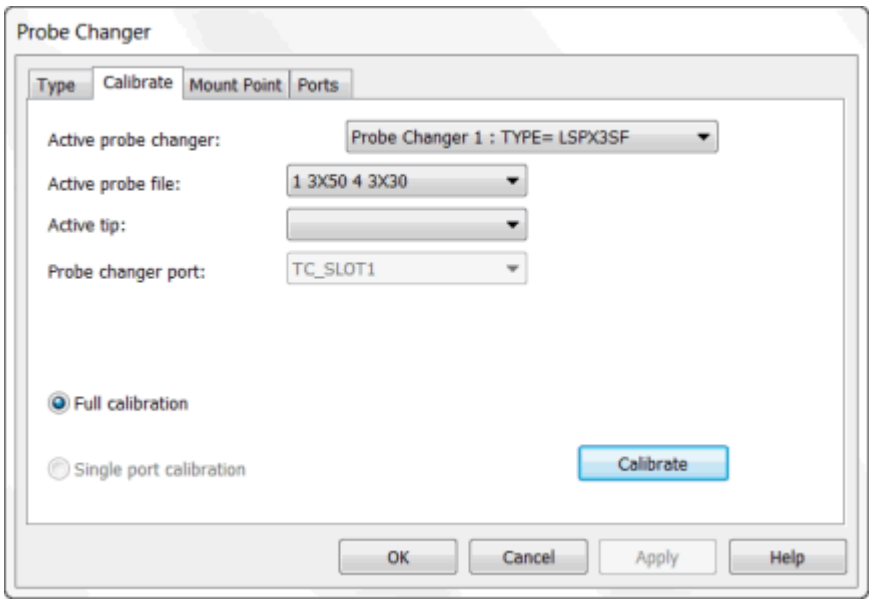
Step 4 - Prepare for Calibration

This step starts the calibration process for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer.

You may use a straight-down 5 x 50 mm stylus tip.

To start the calibration process:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer : TYPE= LSPX3SF**.
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what is to be used for the probe changer calibration, select the proper probe.
4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not what you will use for the probe changer calibration, select the tip ID to use for calibration. This is usually **TIP1**.
5. Empty the ports of all probes.
6. Click **Calibrate**.
7. When you are ready to take the first measurement, click **OK**.

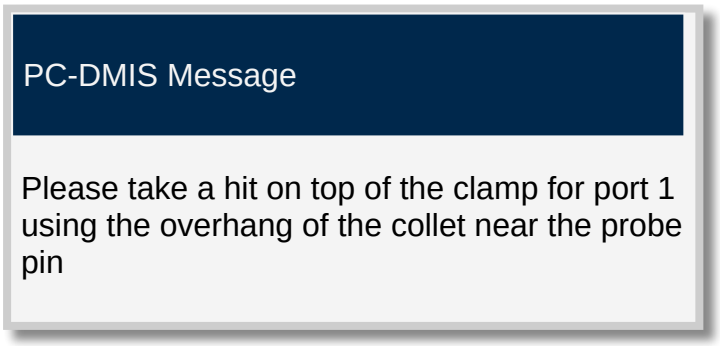
In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

When you start the calibration procedure for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer, PC-DMIS shows you a sequence of message boxes that step you through a series of manual hits. You first take one hit on the overhang of the collet, and then three hits on each port. These hits fully define the orientation of the probe changer in the machine's XY plane. You do not need to align the probe changer to any single axis, as the manual hits identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

> **!** On the SF style ports, you need to slide the port covers back out of the way prior to beginning the alignment hits.

You take the first hit on top of the clamp for port 1, using the overhang of the collet near the probe pin. The prompt for the first hit on port 1 is:



PC-DMIS prompt to take the first manual hit on port 1.

You must take a hit on the top of the first port, using the overhang of the collet. The image below shows the overhang of the collet:

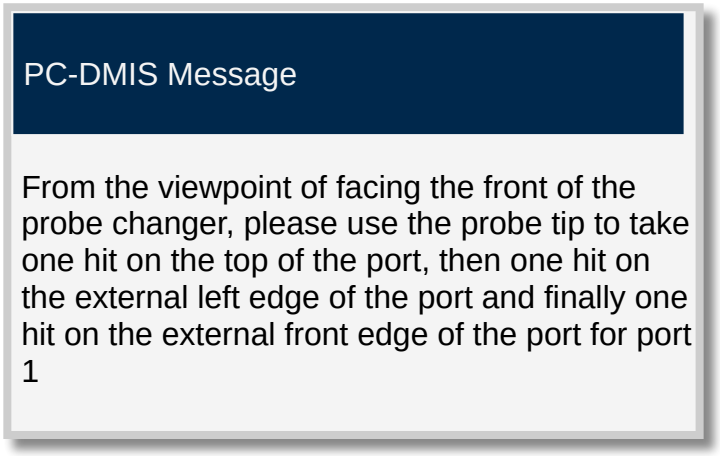
Location of first hit for port 1 on overhang of collet

1. When you are ready to take the first measurement, click **OK**.
2. Use your machine's jog box to take the first manual hit as instructed by PC-DMIS.

In the next step, you define the location of the first port.

Step 6 - Define the Location of the First Port

In this step for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer, you define the location of the first port. This message appears:



PC-DMIS prompt to define the location of the first port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

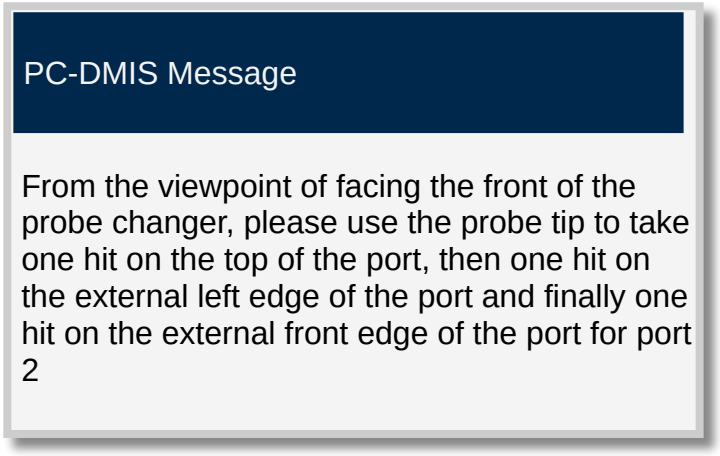
You must take three hits:

1. Take the first hit on the top of the port.
2. Take the second hit on the external left edge of the port, on the outer face of the left side.
3. Take the third hit on the external front edge of the port.

In the next step, you define the location of the second port and all subsequent ports.

Step 7 - Define the Location of the Second Port

In this step for the LSPX3SF / HR-X (HR-X3-P-SF) probe changer, you define the location of the second port and all subsequent ports that are available on your probe changer. This message appears:



PC-DMIS prompt to define the location of the second port.

To define the locations of the second port and all subsequent ports, follow the procedure described in Step 6 - Define the Location of the First Port. PC-DMIS prompts you for each port in turn, and these hits establish the location of every port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

You take all of these hits in manual mode with your machine's jog box.

In the next step, you review the calibration results.

Step 8 - Review the Calibration Results

1. When the calibration of the LSPX3SF / HR-X (HR-X3-P-SF) probe changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port.

2. When you view the results, look for the following.

    - This probe changer is not necessarily aligned parallel to any axis of the CMM.
    - The Z values should be nearly identical since the ports are all at the same height.

      Any significant deviations from these expectations may be due to a bad hit.

The results during measurement routine execution are:

1. The probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.

2. The probe body moves to the mount point and then into the "unload" port (the port that used to contain the currently used probe entity) to drop off the current probe. The current "puck", a conical piece of hardware that connects to the bottom of the probe body, is held in place by the probe changer while the probe body lifts away to detach.

3. From there, the probe moves over to the next "load" position. The magnetic connection automatically engages to load the new module.

4. The probe then moves back to the probe changer’s mount point.

5. From there, it continues to measure.

In the next step, you can choose to recalibrate individual ports.

Step 9 - Recalibrate Individual Ports

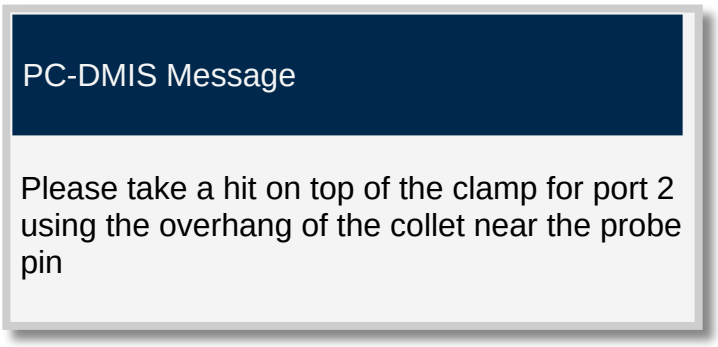
After you complete a full calibration of the LSPX3SF / HR-X (HR-X3-P-SF) probe changer, if you suspect there's an issue with the calibration results of a port, you can choose to recalibrate individual ports.

1. Select the **Single port calibration** option on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). (If you did not complete a full calibration, the option is not available.)

2. When you select the **Single port calibration** option, the **Probe changer port** list becomes available. Select the port that you want to calibrate.

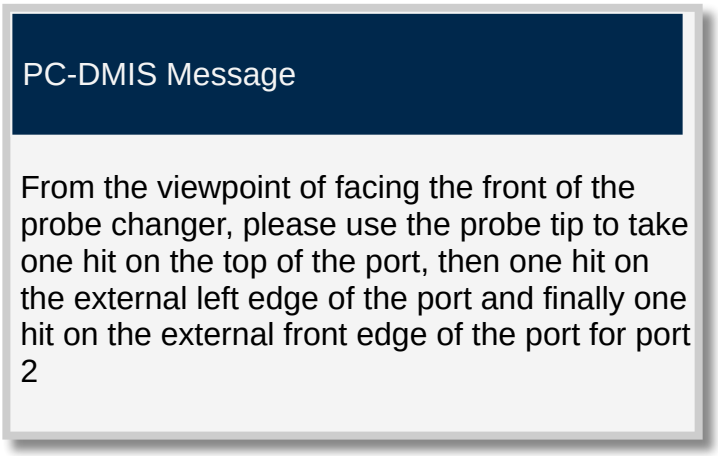
3. Remove any probe or stylus holder from the port that you selected.

4. To begin the calibration, click **Calibrate**. PC-DMIS prompts you to take a hit on the overhang of the collet for the port that you selected. For example, for port 2:



PC-DMIS prompt to take a hit on the overhang of the collet for port 2.

5. PC-DMIS then prompts you to take three hits on the port that you selected. For example, for port 2:



PC-DMIS prompt to define the location of port 2.

6. Click **OK**, and then take the three hits in the order that PC-DMIS prompts for them, as described in Step 6 - Define the Location of the First Port: one hit on the top of the port, one hit on the external left edge, and one hit on the external front edge.

Calibrating the LSPX5 / HR-XS Probe Changer

The calibration process for the LSPX5 / HR-XS probe changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the LSPX5 / HR-XS Probe Changer.

This probe changer MUST BE mounted on the machine table parallel to either the X or Y axis for successful calibration. For mounting instructions, consult the documentation that came with your probe changer.

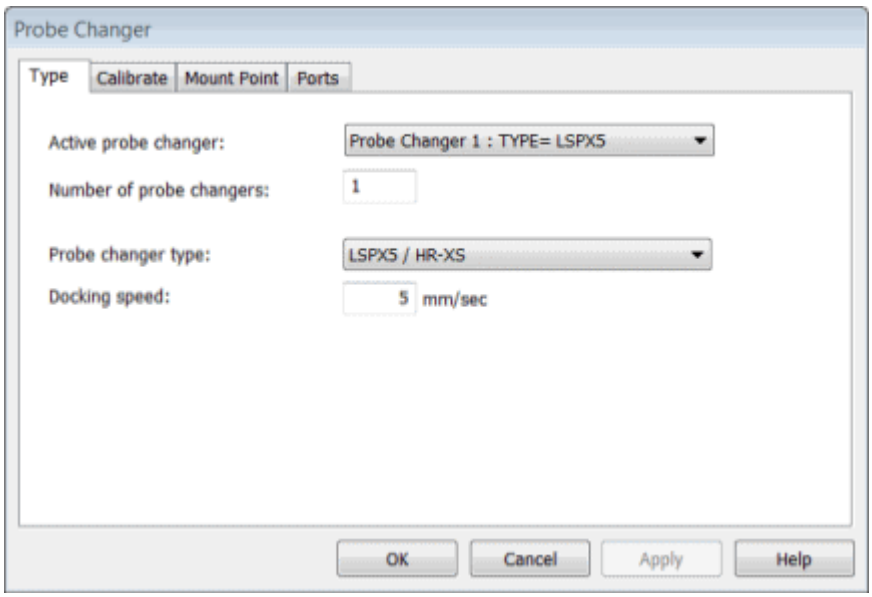
LSPX5 / HR-XS Probe Changer

To calibrate your probe changer:

Step 1 - Select the LSPX5 / HR-XS Probe Changer

To select the LSPX5 / HR-XS Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **LSPX5 / HR-XS**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the list item that represents the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.

5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   > To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

> For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the mount point.

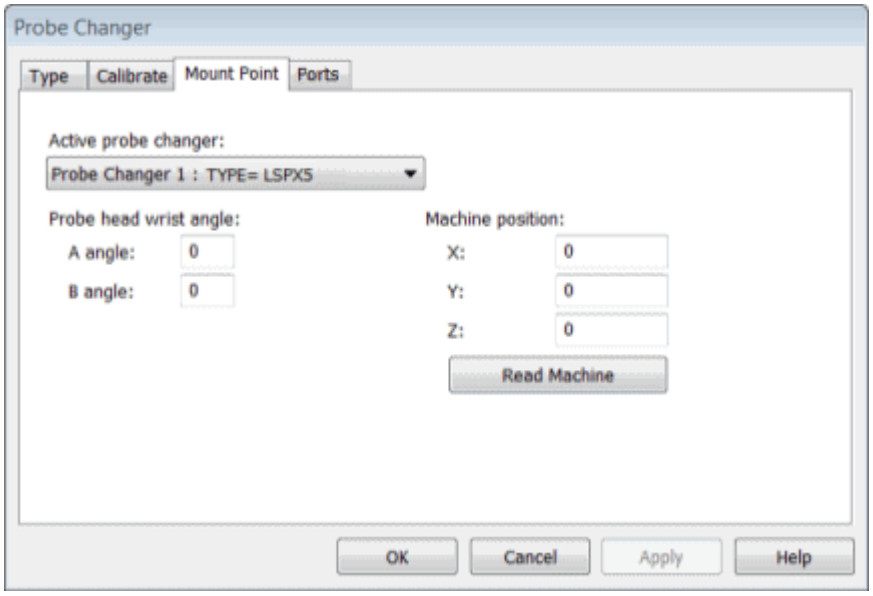
Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that

avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=LSPX5**.
3. Alter the probe head wrist angle for both **A angle** and **B angle** to 0. (This is a fixed head and cannot be mounted on a wrist.)
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

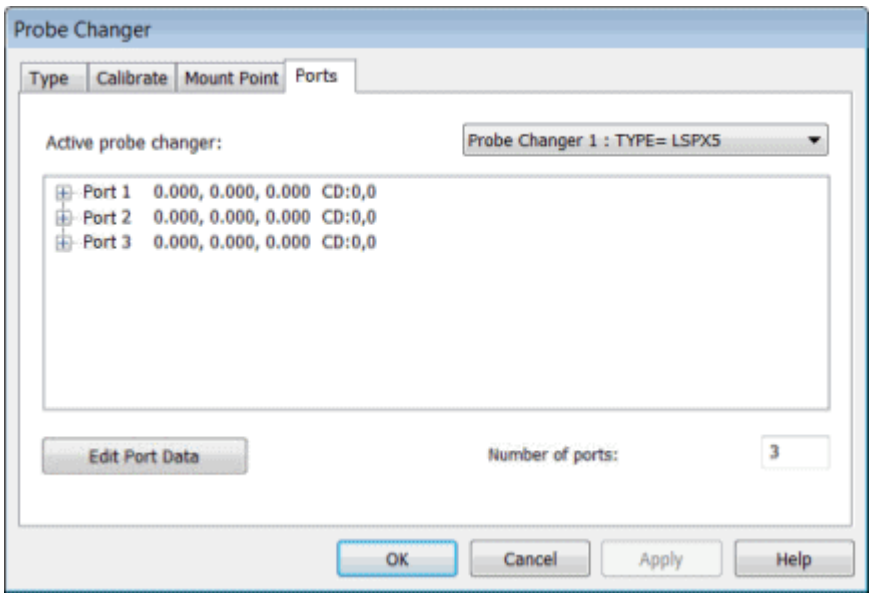
In the next step, you define the ports.

Step 3 - Define the Ports

The final step prior to rack calibration of the LSPX5 / HR-XS Probe Changer is to define the ports on the rack that you are calibrating.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

2. In the **Active probe changer** list, select **TYPE=LSPX5**.
3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

Step 4 - Prepare for Calibration

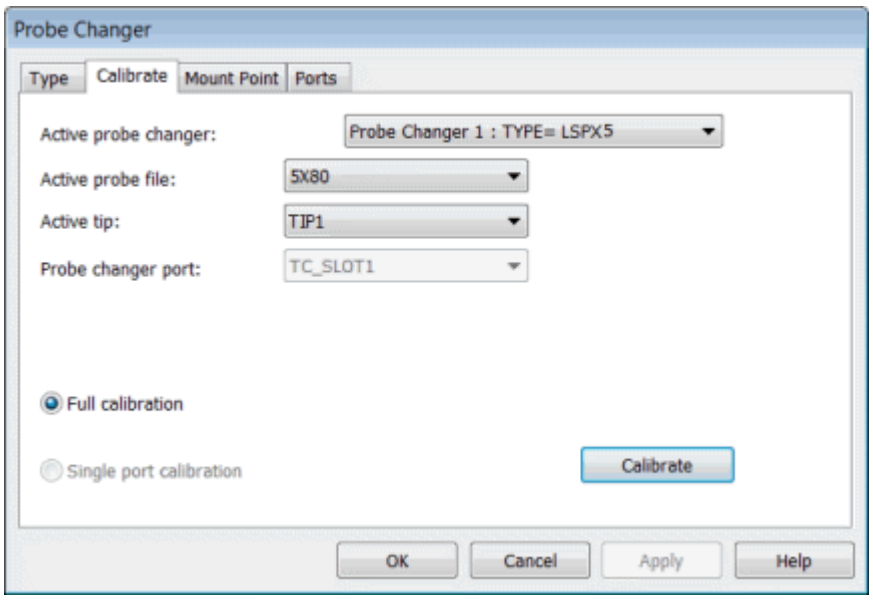
This step starts the calibration process for the LSPX5 / HR-XS probe changer.

The calibration process does not need any specific probe tip. However, a sturdy probe tip is recommended. Typically, a 5 x 60 mm tip is used.

Calibrate with a straight tip that points down with no cubes or any other joints.

To begin the calibration process, do the following:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



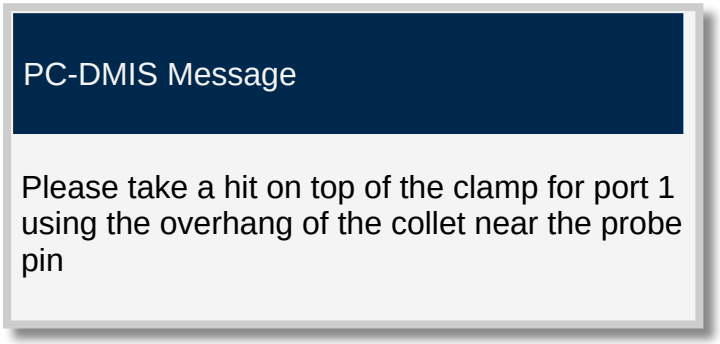
Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=LSPX5**.
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what is to be used for the rack calibration, select the proper probe.
4. The entry in the **Active tip** list defaults to the current measurement routine setting. If this is not what is to be used for the rack calibration, select the tip ID to be used for calibration. This is usually **TIP1**.
5. To begin, click **Calibrate**.

In the next step, you take a point with the over-hang of the collet.

Step 5 - Take a Point with the Overhang of the Collet

For the LSPX5 / HR-XS Probe Changer, the prompt to take a point on the overhang of the collet is:

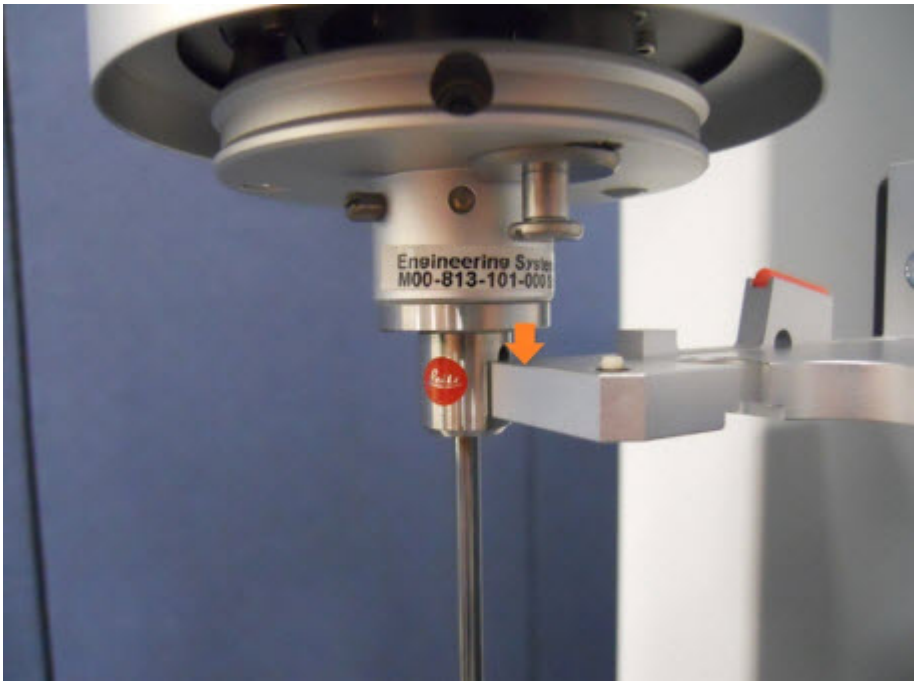


PC-DMIS prompt to take a hit on top of the clamp for port 1.

You must take a hit on the top of the first port, using the overhang of the collet. The image below shows the overhang of the collet:

Overhang of the collet

The image below shows how to take a point with the overhang:

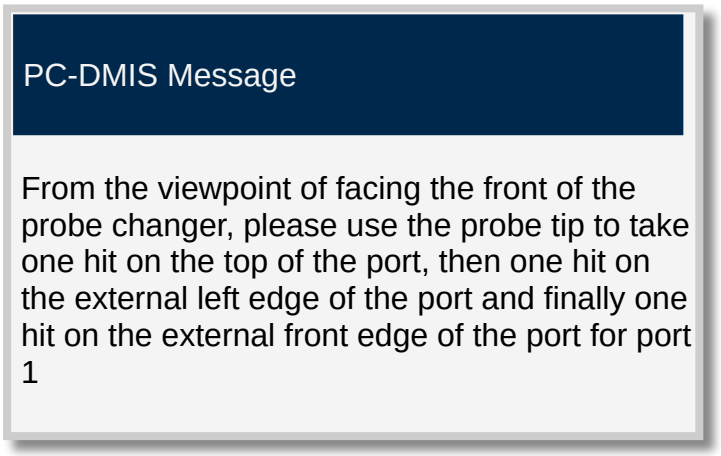


One point with the overhang of the collet

In the next step, you define the location of the first port.

Step 6 - Define the Location of the First Port

In this step for the LSPX5 / HR-XS Probe Changer, you define the location of the first port. This message appears:

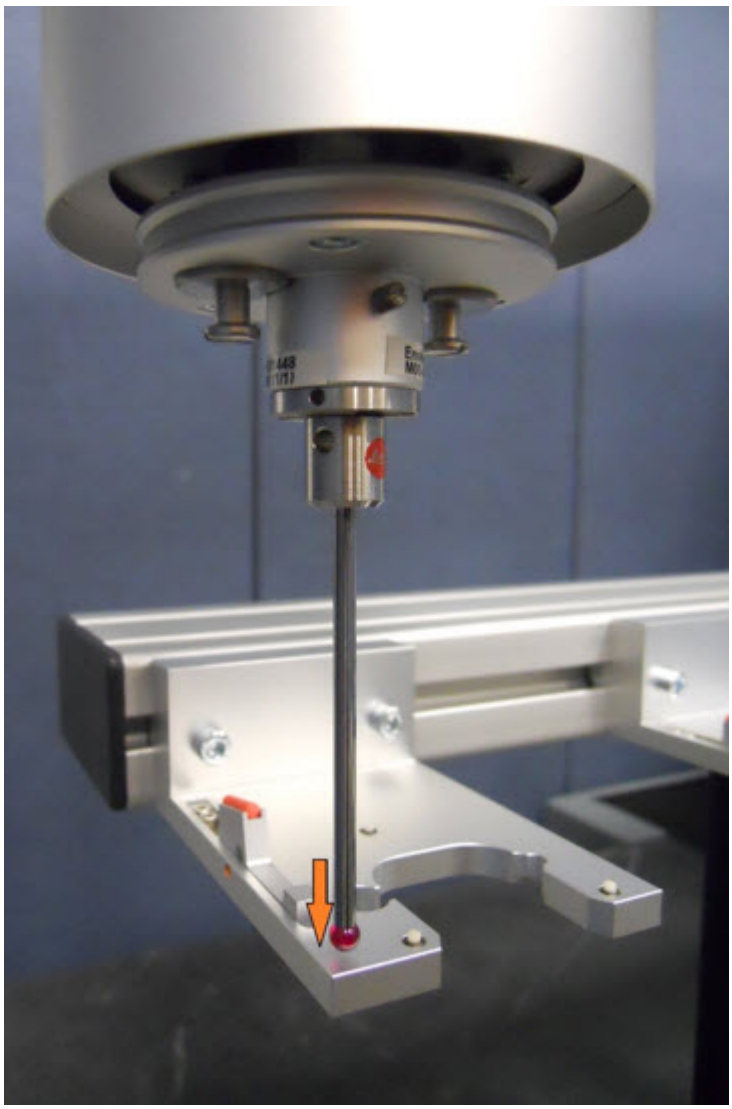


PC-DMIS prompt to define the location of the first port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

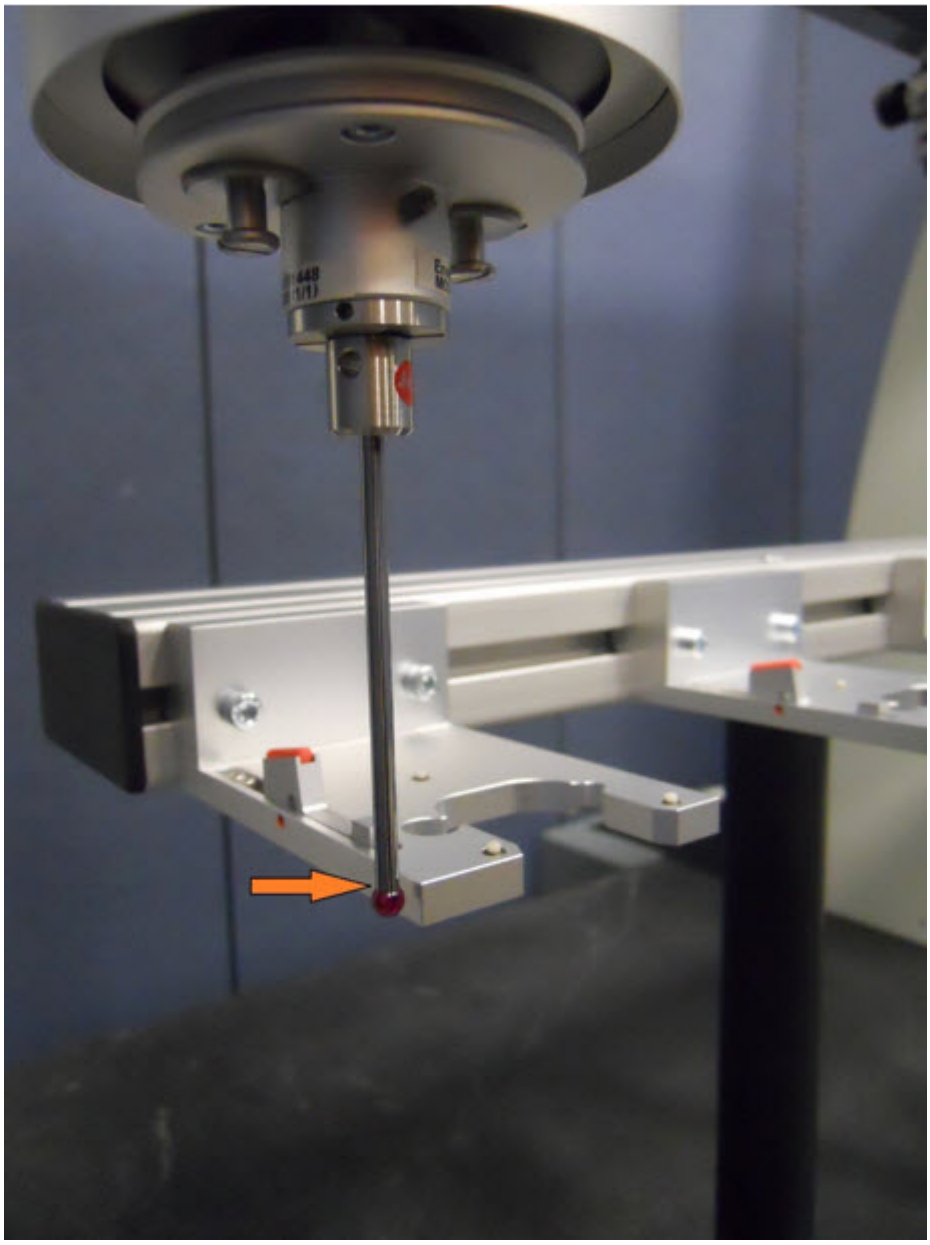
You must take three hits:

1. Take the first hit on the top of the port to define its location. The image below shows the location:



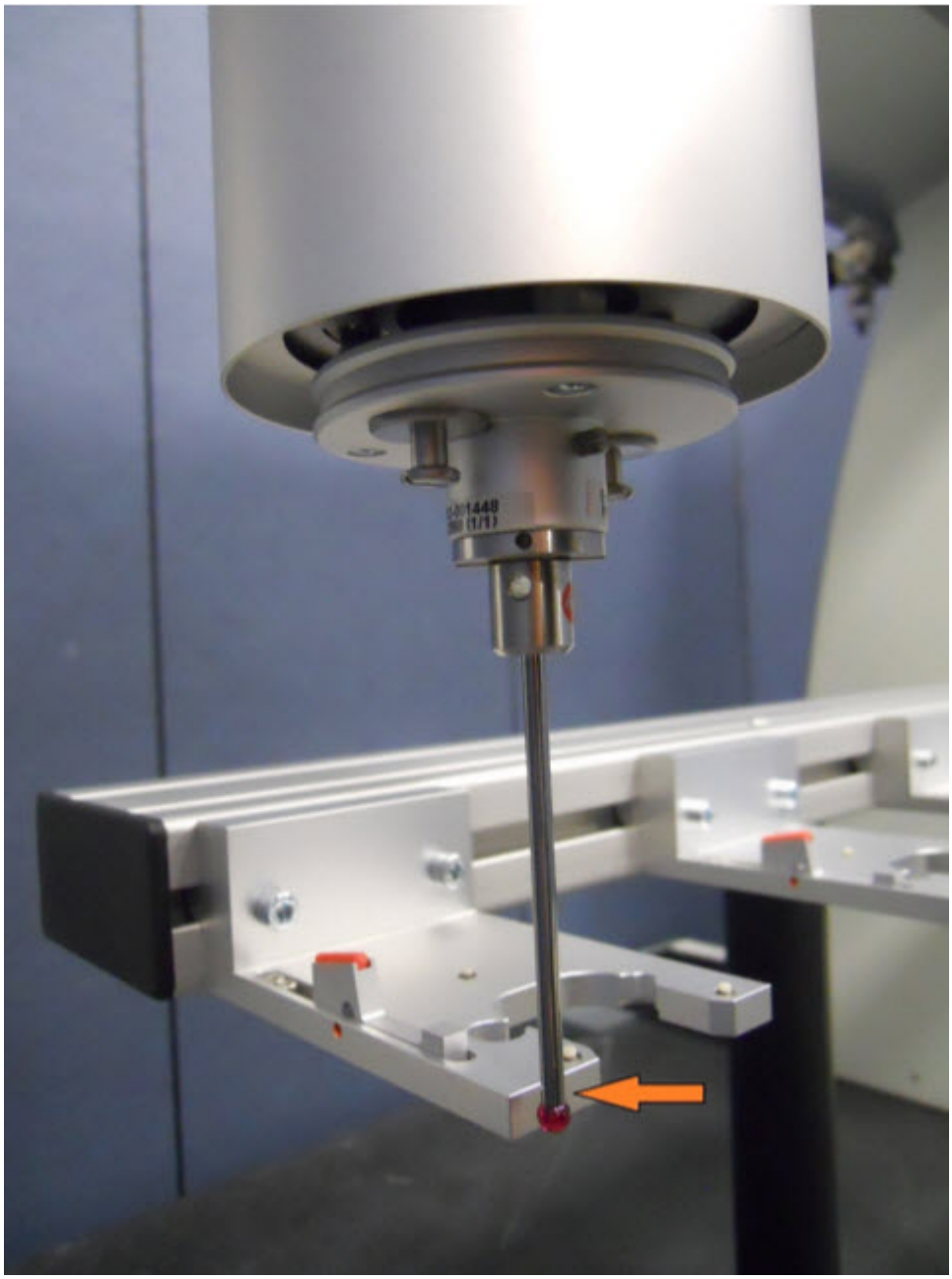
Example showing the first hit on top of the port.

2. Take the second hit on the left of the port to define its location. The image below shows the location:



Example showing the second hit on the left side of the port.

3. Take the third hit on the front of the port to define its location. The image below shows the location:



Example showing the third hit in front of the port.

In the next step, you define the location of the second port and all subsequent ports.

Step 7 - Define the Location of the Second Port

In this step for the LSPX5 / HR-XS Probe Changer, you define the location of the second port and all subsequent ports that are available on your probe changer. This message appears:

PC-DMIS Message

From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the top of the port, then one hit on

the external left edge of the port and finally one hit on the external front edge of the port for port 2

PC-DMIS prompt to define the location of the second port.

To take the hits, click **OK**. To abandon the calibration, click **Cancel**.

To define the locations of the second port and all subsequent ports, follow the procedure described in Step 6 - Define the Location of the First Port.

You have now completed the probe changer calibration process.

Calibrating the SCP600 Probe Changer

These topics describe how to define and calibrate the SP600 Probe Changer.

Inserts or extensions are not used for any of the port positions.

A Renishaw SCP600 Stylus Change Rack

The SCP600 Probe Changer MUST BE mounted on the machine table parallel to either the X or Y axis. The stylus must be greater than 40 mm so the puck and body of the probe clear the stylus wrench used to hold the port open. For mounting instructions, consult the documentation that came with your probe changer.

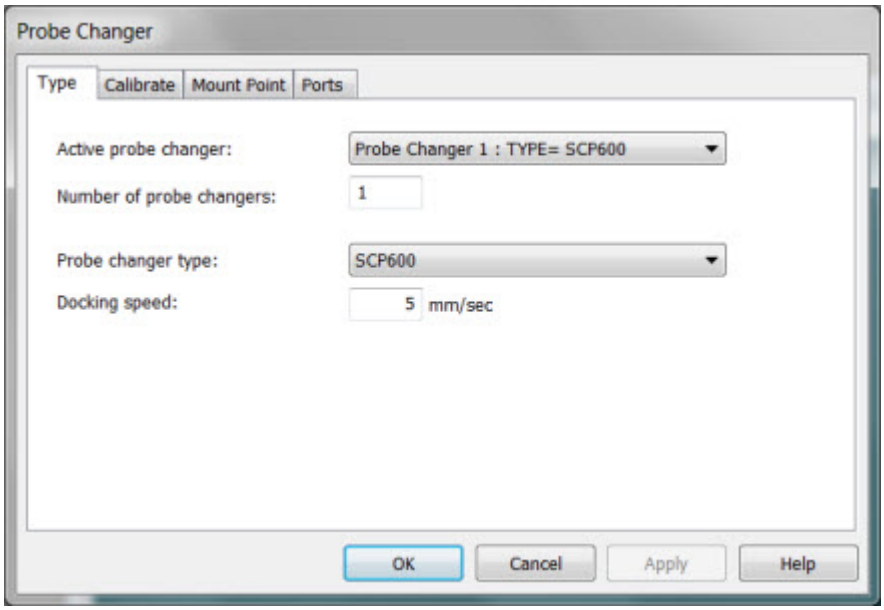
To calibrate your probe changer:

Step 1 - Select the SCP600 Probe Changer

To select the SCP600 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.

2. In the **Probe changer type** list, select **SCP600**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the list item that represents the probe changer to define. If this is the first probe changer, this shows **Probe Changer 1: TYPE=None**.

5. In the **Number of probe changers** box, specify the number of different types of probe changers.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

> To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

> For details on controller configuration settings, see the documentation that came with your controller.

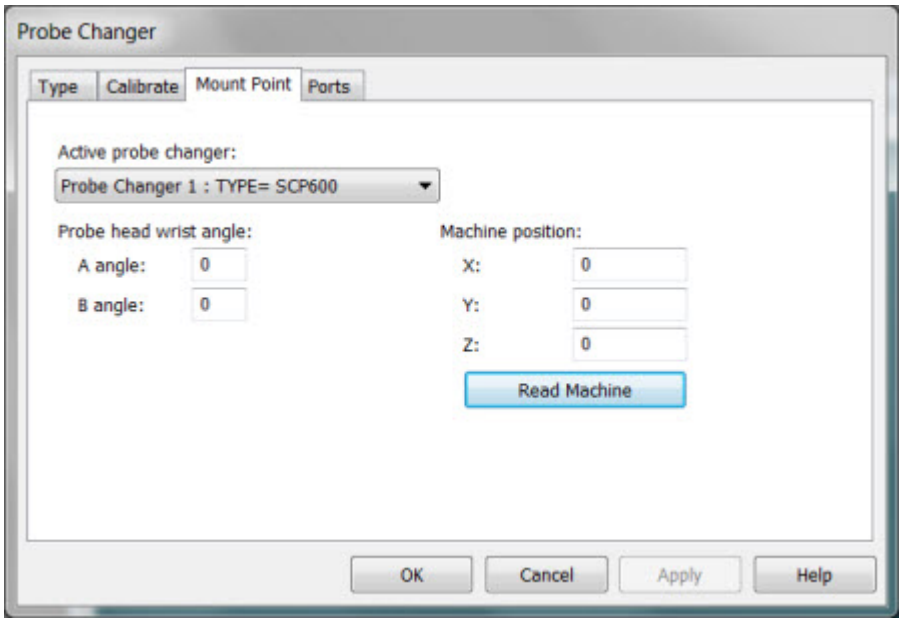
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active Probe Changer** list, select **TYPE=SCP600**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Typically, but not always, these values are both 0 (zero). You need to use a calibrated probe rotation to ensure the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine Position** values with the current position. You may also manually type these values.
6. To save your changes, click **Apply**.

In the next step, you define the ports.

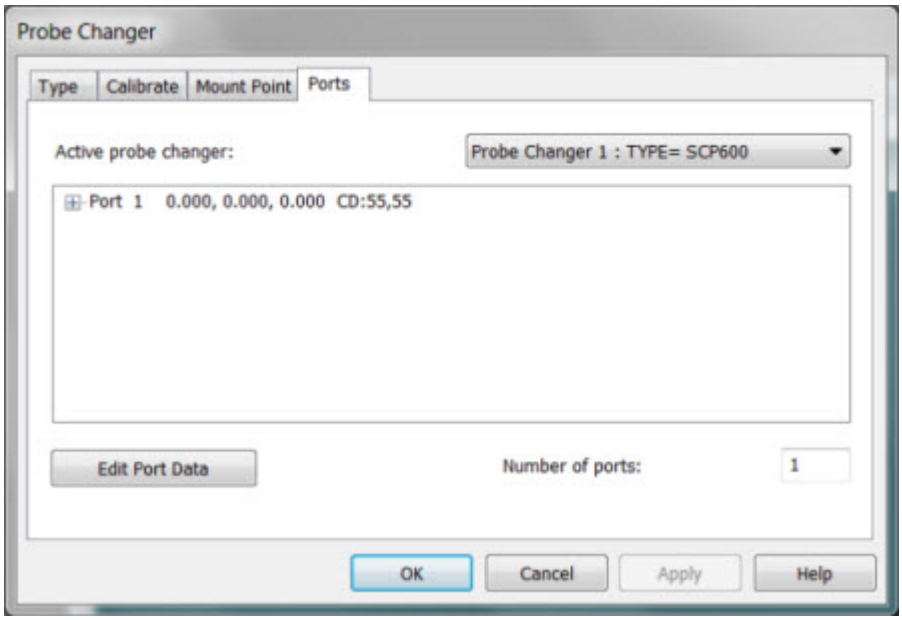
Step 3 - Define the Ports

The SCP600 is a single port rack. You can make use of any quantity of racks acting independently. You can also assemble them to form a multi-port rack of any quantity. This requires that you position them tightly against one another so that there is no space between them.

Regardless of how many rack pieces you use, you must define and calibrate them individually. However you choose to use them, you must first enter the number of ports in the **Ports** tab before calibration begins.

To define the ports for your SCP600 Probe Changer, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

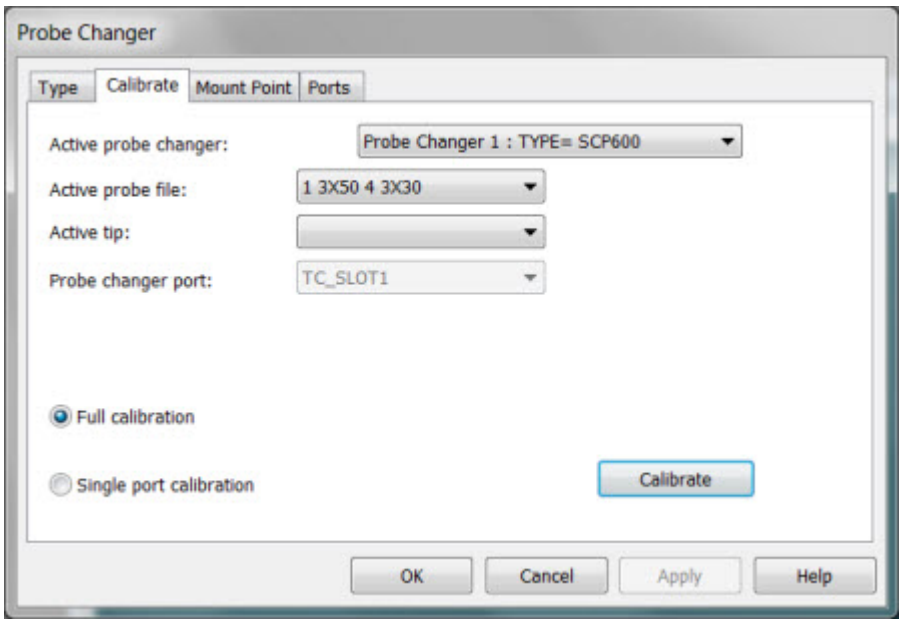
2. In the **Active Probe Changer** list, select **TYPE=SCP600**.
3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

Step 4 - Prepare for Calibration

To begin the calibration process for the SCP600 Probe Changer:

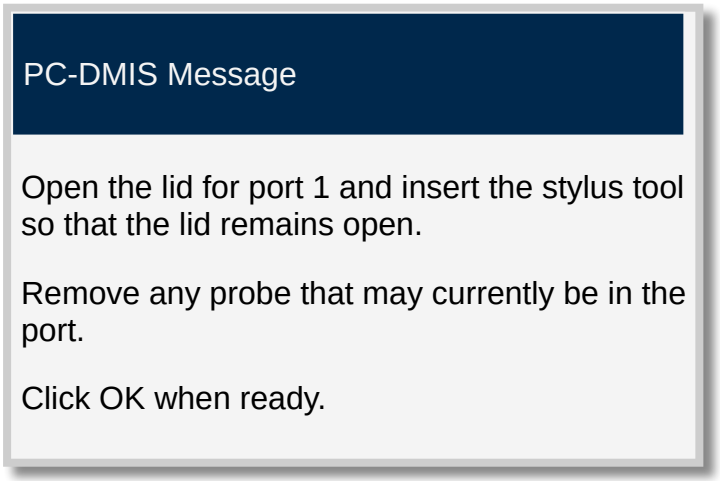
1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



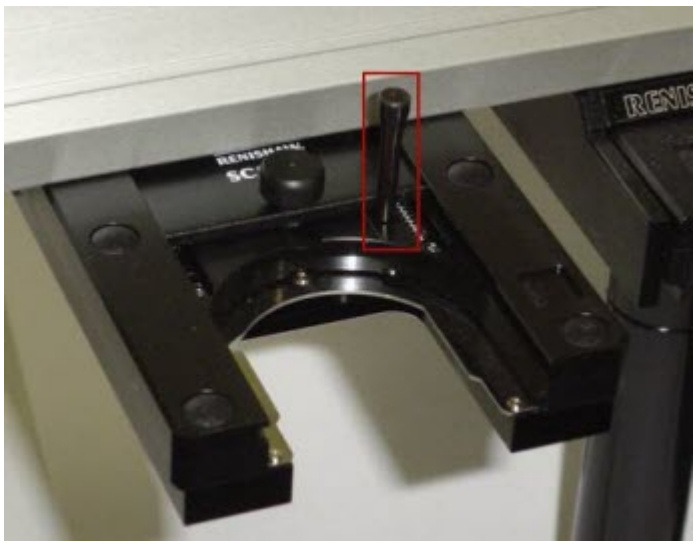
Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=SCP600**.
3. In the **Active tip** list, select the tip ID to use for calibration. This is usually **T1A0B0**.
4. Click the **Calibrate** button.

Once you click **Calibrate**, the following message box appears:



1. To open the lid or lids, push them all the way back and insert a metallic pin (called a stylus wrench) into the hole in the top of the port near the back right corner. This hole is located to the side of the spring on the right when facing the rack.
2. With the lid or lids open, remove any modules or styli by sliding them forward and out of the ports.



Example of the SCP600 probe changer with an empty port held open by the stylus wrench.

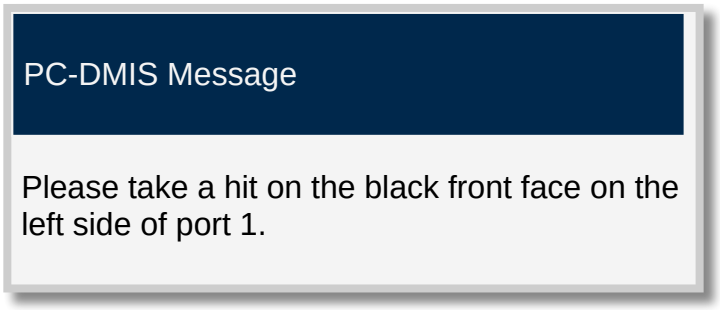
3. When you are ready to take the first measurement, click **OK**.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

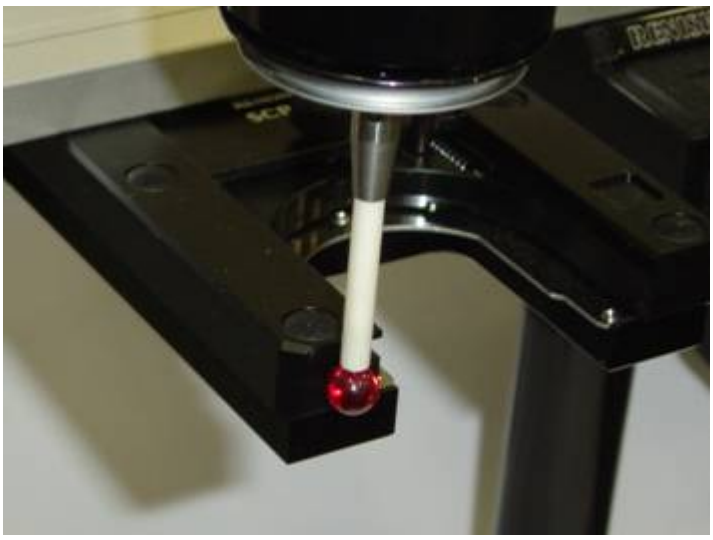
For the SCP600 Probe Changer, a series of message boxes prompts you through the process of measuring two manual hits. Two hits are required to define the location of this rack in the machine's workspace. Follow the prompts and take the needed hits as directed in the instructions for each hit.

You will take the first manual hit in the XY plane on the vertical face in front of the rack on the left side of the port. The prompt is:



PC-DMIS prompt for the first manual hit.

Use your machine's jog box to measure the first hit on the front face of the port, to the left of the opening:



In the next step, you take the second manual hit.

Step 6 - Take the Second Manual Hit

For the SCP600 Probe Changer, you will take the second manual hit in the Z direction, on the horizontal face on top of the rack to the left of the open port.

The prompt for the second hit on the top is:

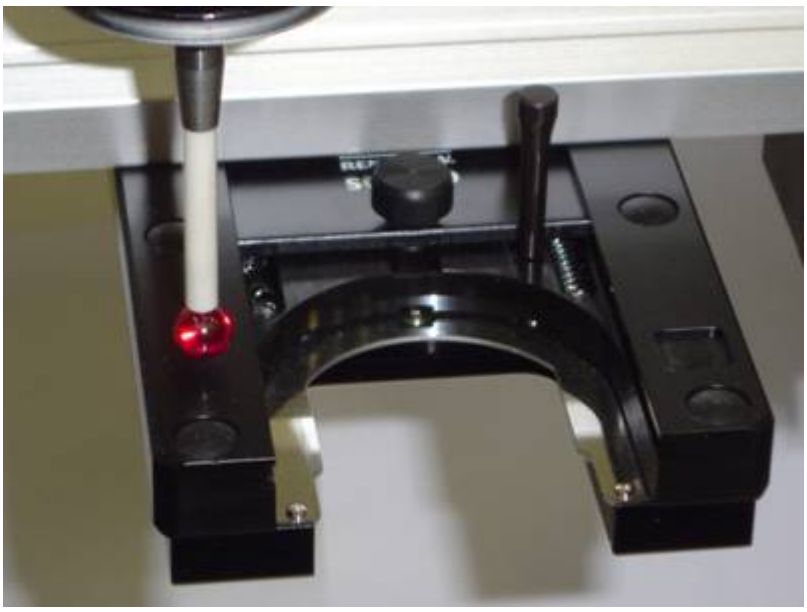
PC-DMIS Message

Please take a hit on the black top face on the left side of port 1.

The hit should be on the left portion and not inside a recessed circle.

After completing this hit, DCC calibration will begin to complete the measurement of this port.

Use your machine's jog box to measure the second hit on the top, flat, plastic surface of the port, left of the opening:

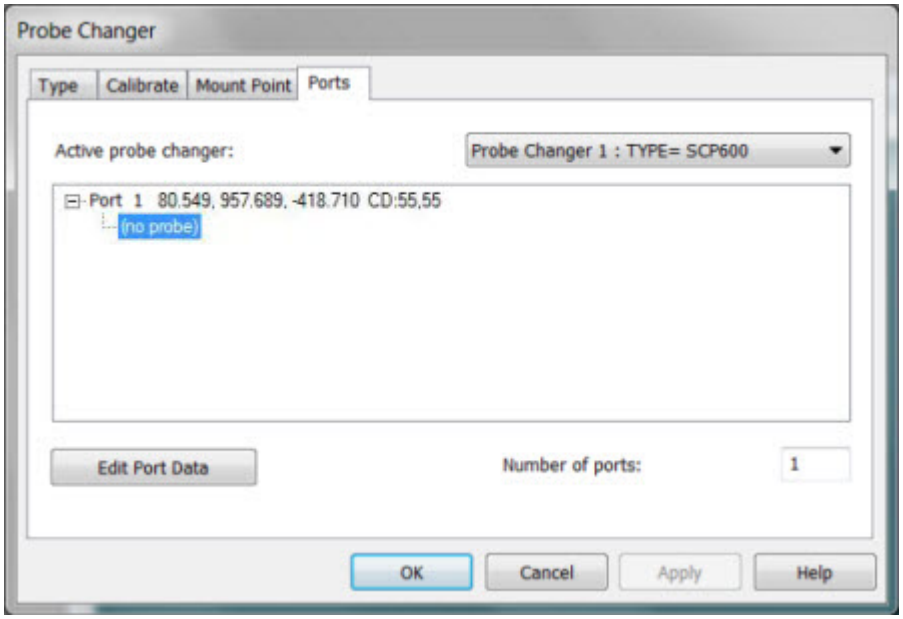


WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

In the next step, you review the calibration results.

Step 7 - Review the Calibration Results

1. When the calibration of the SCP600 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Some things to watch for are the position of the rack and the spacing of the ports. Any significant deviations from these expectations may be due to a bad hit.
    - This rack must be aligned parallel to the X or Y axis of the CMM.
    - When using multiple ports as one rack, the X and Y values should show equal spacing between ports, roughly 85 mm apart.
    - The Z values should be nearly identical, since the ports are all at the same height.

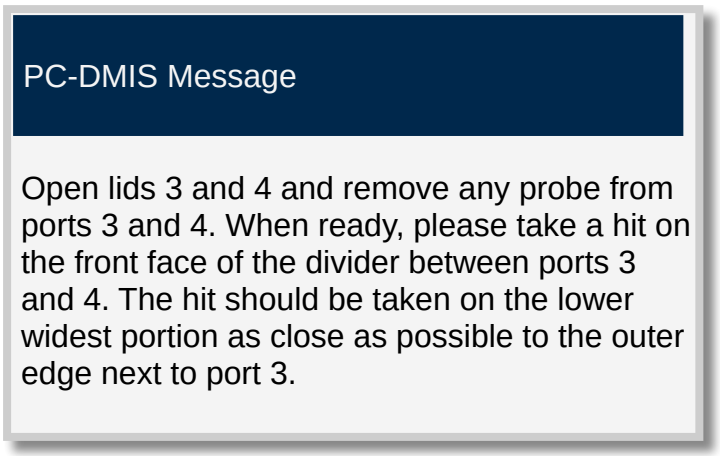
The results during measurement routine execution are:

1. The probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the "unload" port (the port that used to contain the currently-used probe entity), which pushes the lid backwards. The current "puck", a black and conical piece of hardware that connects to the bottom of the probe body, is held in place by the rack while the probe body lifts away to detach.
3. From there, the probe moves over to the next "load" position. The magnetic connection automatically engages to load the new module.
4. It then moves back to the rack’s mount point.
5. From there, it continues to measure.

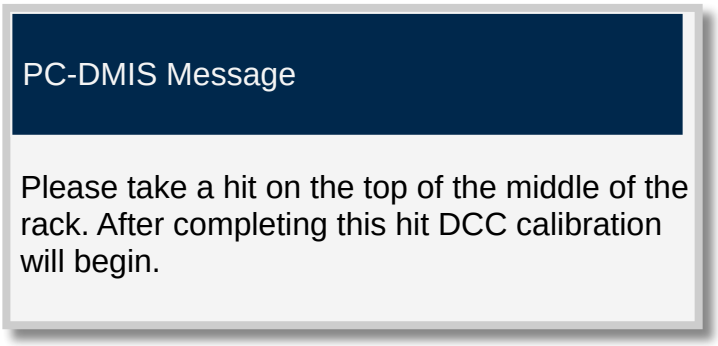
Calibrating the SCR200 Probe Changer

This topic describes how to calibrate the SCR200 Probe Changer.

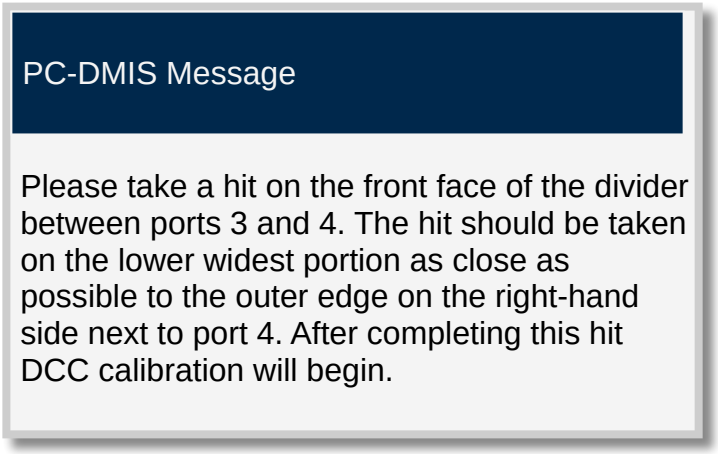
When you click the **Calibrate** button on the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), the first prompt appears:



1. Open lids 3 and 4. Remove any probes from those ports.
2. Click **OK**.
3. Take one hit on the front face of the probe changer's center post. A second prompt appears and asks you to take the second hit:



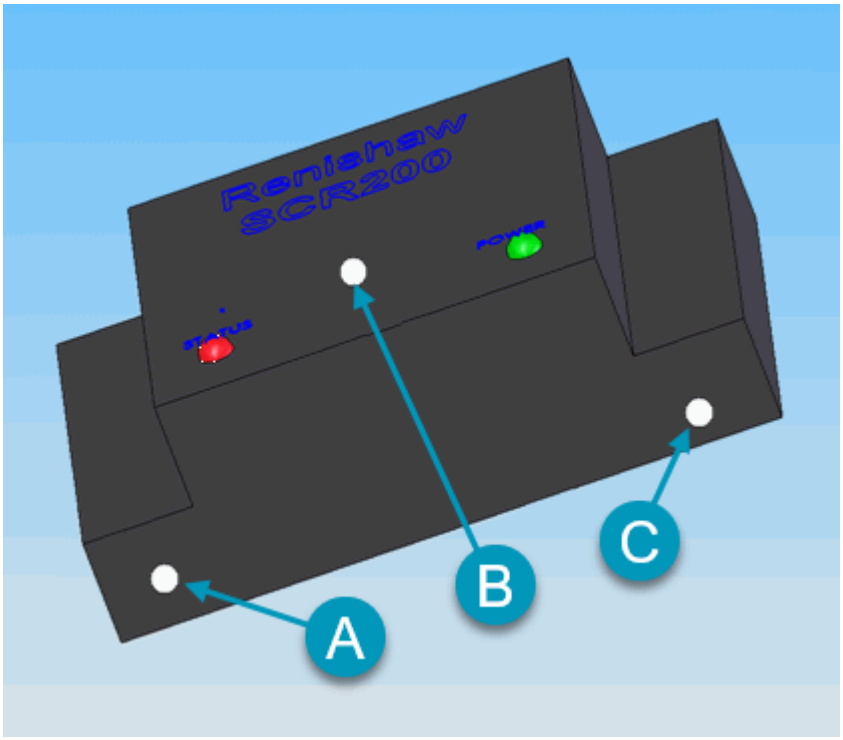
4. Click **OK**.
5. Take the second manual hit on the top of the center post. A third prompt appears and asks you to take the final hit:



6. Click **OK**.
7. Take the third manual hit again on the front face of the center post.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

- You should have taken all three hits similar to shown here:



A - 1st Manual Hit (in front of center post)

B - 2nd Manual Hit (on top of center post)

C - 3rd Manual Hit (in front of center post)

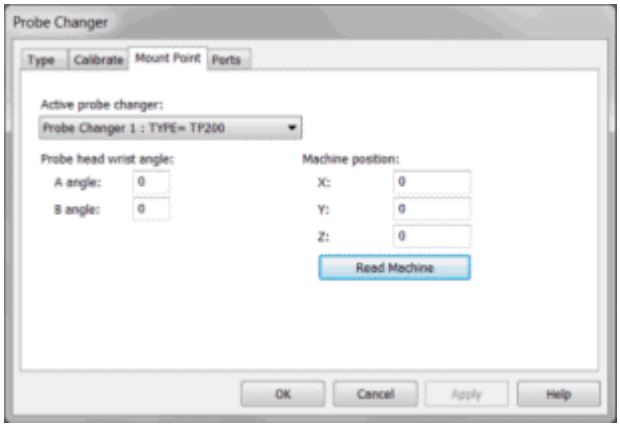
- PC-DMIS takes the remaining hits in DCC Mode:

  1 hit each on the left and right inside sides of port 3.

  1 hit each on the left and right inside sides of port 4.

After the calibration, the **Probe Changer** dialog box appears.

1. Select the **Mount Point** tab:

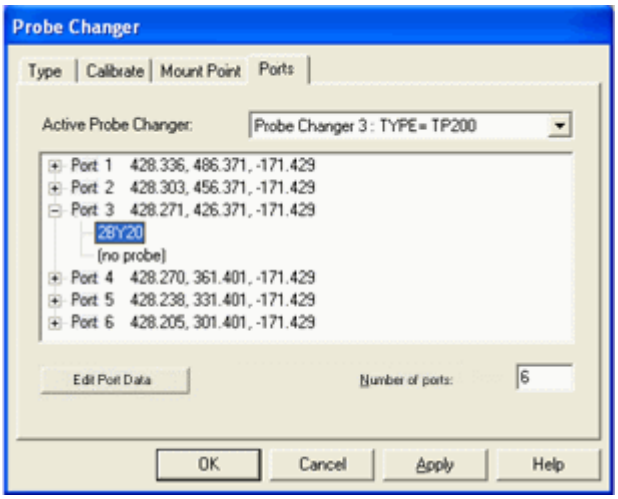


Probe Changer Dialog Box - Mount Point Tab

2. Confirm the wrist angle for the change cycle by typing in the angle in the **A Angle** and **B Angle** boxes.

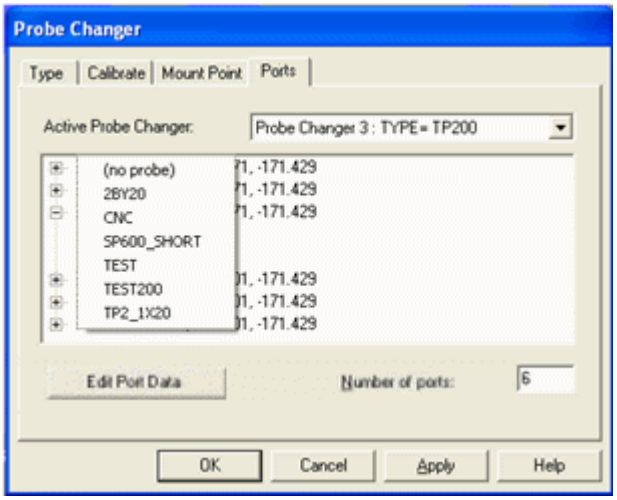
3. Specify the rack's safe position (a position where probes can be swapped in and out safely) in machine coordinates. You can type this manually into the **X**, **Y**, and **Z** boxes in the **Machine Position** area, or click the **Read Machine** button.

4. Select the **Ports** tab:



Probe Changer Dialog Box - Ports Tab

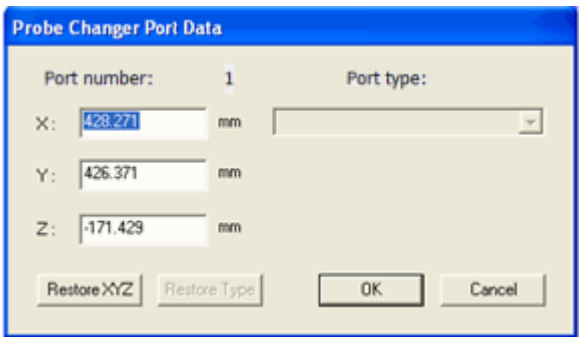
5. To define the contents of each port in the rack, expand the plus sign **(+)** for each port and then double-click the **(no probe)** item. A list of available probes appears. For example:



List of available probes

6. Select the probe file to add to the current port.

7. If necessary, adjust the port position. Select the port's position and then click the **Edit Port Data** button. The **Probe Changer Port Data** dialog box appears. For example:



Probe Changer Port Data Dialog Box

8. Modify the **X**, **Y**, **Z** position of the port.
9. Click **OK** to close the **Probe Changer Port Data** dialog box.
10. Click **OK** to close the **Probe Changer** dialog box.

Calibrating the Senmation Probe Changer

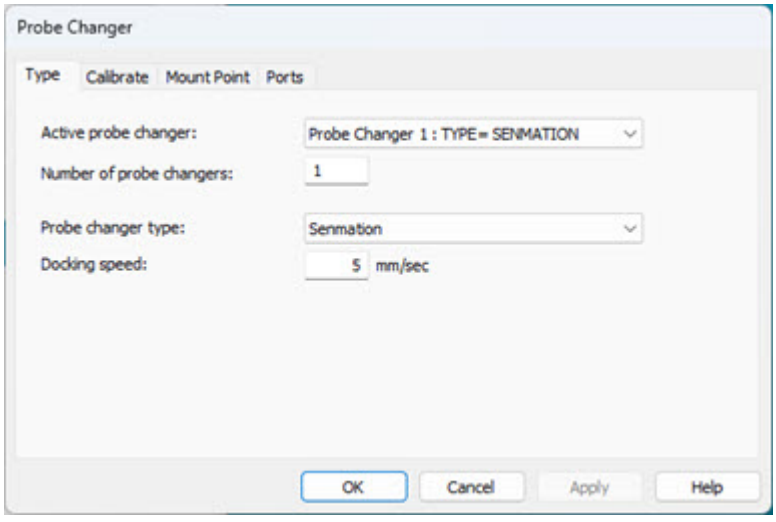
The calibration process for the Senmation probe changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the Senmation probe changer.

To calibrate your probe changer:

Step 1 - Select the Senmation Probe Changer

To select the Senmation probe changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **Senmation**:



Probe Changer Dialog Box - Type Tab

3. Click **Apply** to make the probe changer active and load the settings that are relevant to it. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** list, select the probe changer that you want to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.
6. In the **Docking speed** box, type a value. A value between 10 and 20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to increase or decrease the docking speed. For the first replacement attempt, use a reduced speed to help protect the machine. After you finish testing, you can return to the regular speed.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply your changes, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

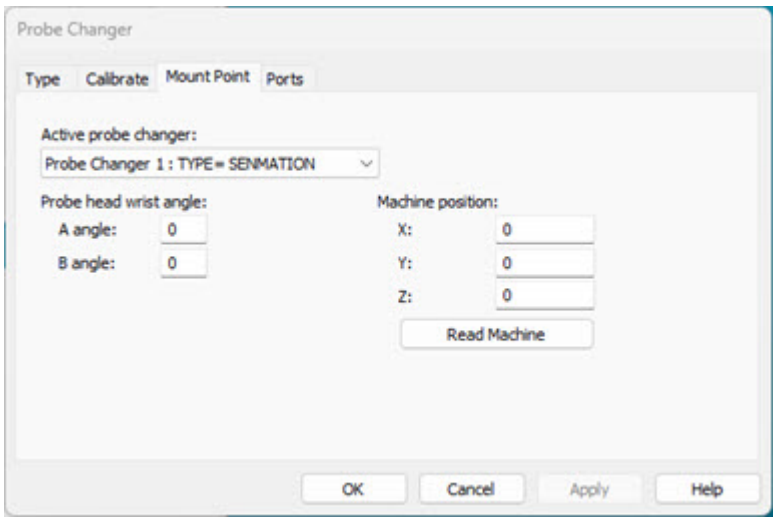
In the next step, you define the mount point.

Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=SENMATION**.
3. Rotate the probe head wrist to a suitable **A angle** and **B angle**.
4. Use the jog box to manually move the machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also enter these values manually.
6. To save your changes, click **Apply**.

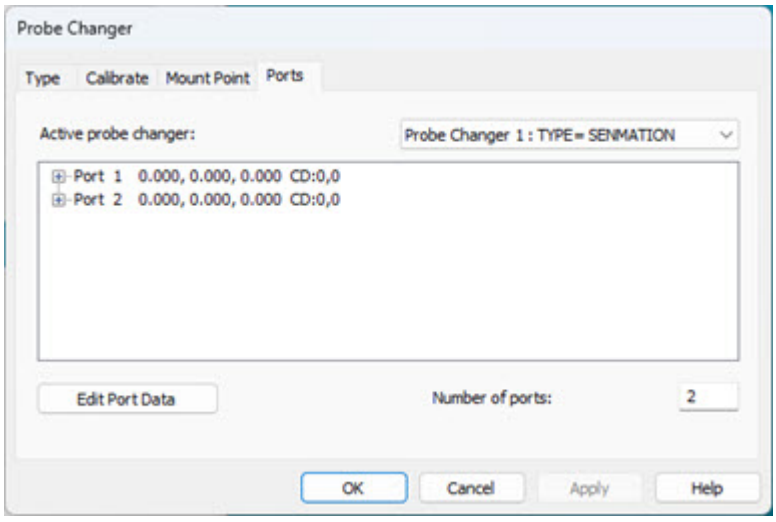
In the next step, you define the ports.

Step 3 - Define the Ports

Before calibrating the rack, define the number of ports on the probe changer.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

2. In the **Active probe changer** list, select **TYPE=SENMATION**.
3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

You are now ready to prepare the probe changer for calibration. The next step describes how to select the probe and tip to use for calibration.

Step 4 - Prepare for Calibration

This step describes how to prepare the Senmation probe changer for calibration.

In this version, you can calibrate the Senmation change rack only with a rotary probe head. Use a 3 x 50 mm or 5 x 50 mm tip for calibration.

Use a straight tip that points down and has no cubes or other joints.

Before you calibrate the change rack, you must calibrate the following probe angles so that PC-DMIS can calculate accurate probe head offset distances and ensure safe probe exchange: A0B0, A0B90, A0B-90, A0B180, A45B0, A90B0, A90B90, A90B180. If any of these angles are not calibrated, PC-DMIS displays a message similar to the following:

PC-DMIS

In order to calibrate senmation tool changer, you need to use a probe with the following calibrated tip angles:

T1A0B0

T1A45B0

T1A90B0

T1A90B90

T1A90B180

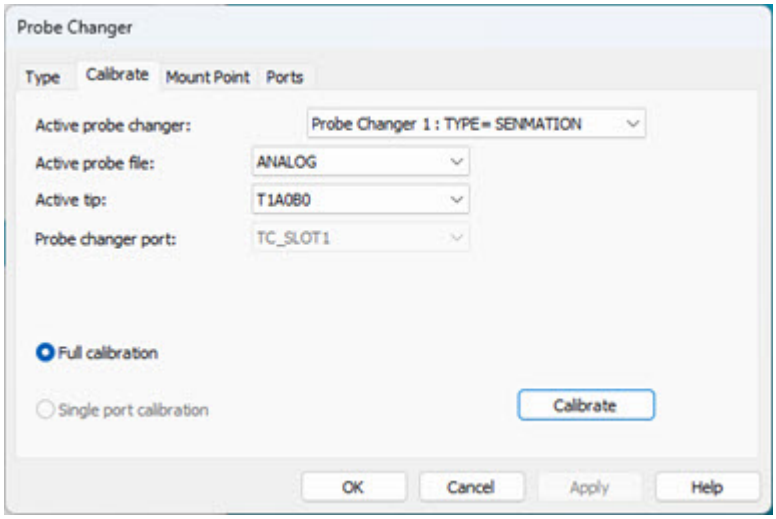
T1A0B90

T1A0B-90

T1A0B180

To begin the calibration process, do the following:

1. Select the Calibrate tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=SENMATION**.
3. The **Active probe file** list defaults to the probe file specified in the current measurement routine. A rotary probe head is required for calibration.
4. The **Active tip** list defaults to the tip specified in the current measurement routine. One calibrated tip is required.
5. Click **Calibrate** to begin the calibration process.

Orient the change rack as level as possible and aligned with the machine beam direction to help ensure safe probe exchange.

In the next step, take the required manual hits.

Step 5 - Take the Manual Hits

When you click the **Calibrate** button on the Calibrate tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), a series of message boxes guides you through the process of taking manual hits. The number of manual hits required depends on the number of ports in the probe changer.

For example, a two-port rack requires five manual hits to fully define the rack's orientation. Follow the prompts and take the required hits.

The first prompt appears:

PC-DMIS Message

From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the front of the locating block of port 1.

1. Take the first manual hit on the front of the locating block of port 1. (The first port is the leftmost port; take the manual hits in this order.) A second prompt appears:

   PC-DMIS Message

   From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the front of the locating block of port 2.

2. Take the second manual hit on the front of the locating block of port 2. A third prompt appears:

   PC-DMIS Message

   From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the top of the locating block of port 1.

3. Take the third manual hit on the top of the locating block of port 1. A fourth prompt appears:

   PC-DMIS Message

   From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the left of the locating block of port 1.

4. Take the fourth manual hit on the left of the locating block of port 1. A fifth prompt appears:

   PC-DMIS Message

   From the viewpoint of facing the front of the probe changer, please use the probe tip to take one hit on the left of the locating block of port 2.

5. Take the fifth manual hit on the left of the locating block of port 2.

6. After you finish taking the manual hits, move the probe head to a safe position near the first port. Automatic calibration then begins.

If a collision occurs during automatic calibration, open the Settings Editor application and adjust the approach and prehit distances. For more details, see the `TCSenmationDccPrehitSENMATION`, `TCSenmationDccLeftApproachSENMATION`, and `TCSenmationDccRightApproachSENMATION` entries in the "Option" topic of the PC-DMIS Settings Editor documentation.

You have now completed the probe changer calibration process.

Calibrating the SP600 Probe Changer

These topics describe how to define and calibrate the SP600 Probe Changer.

There are no inserts or extensions used for any of the port positions.

A Renishaw SP600 Stylus Change Rack (SCR600)

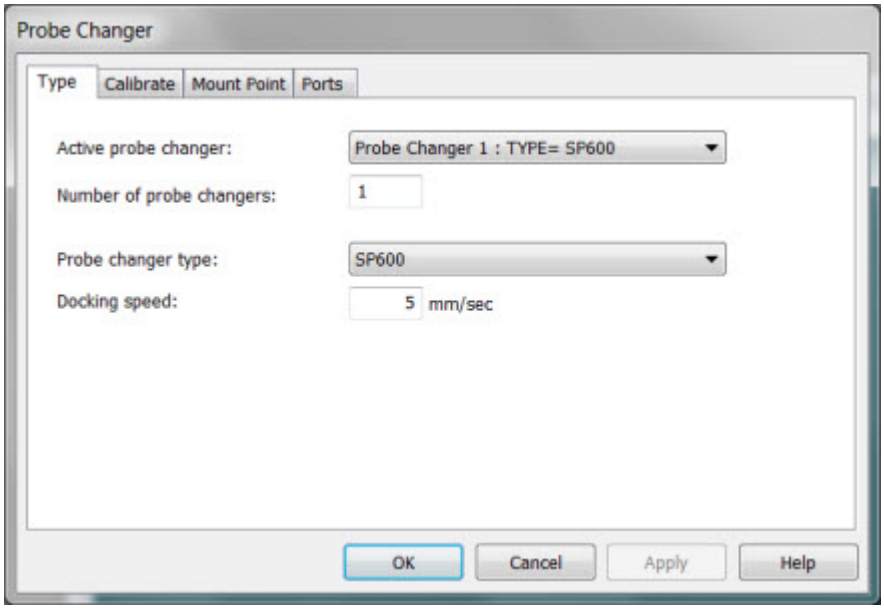
The SP600 Probe Changer DOES NOT need to be mounted on the machine table parallel to either the X or Y axis. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the SP600 Probe Changer

To select the SP600 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **SP600**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** box, select the list item that represents the probe changer to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of different types of probe changers.
6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

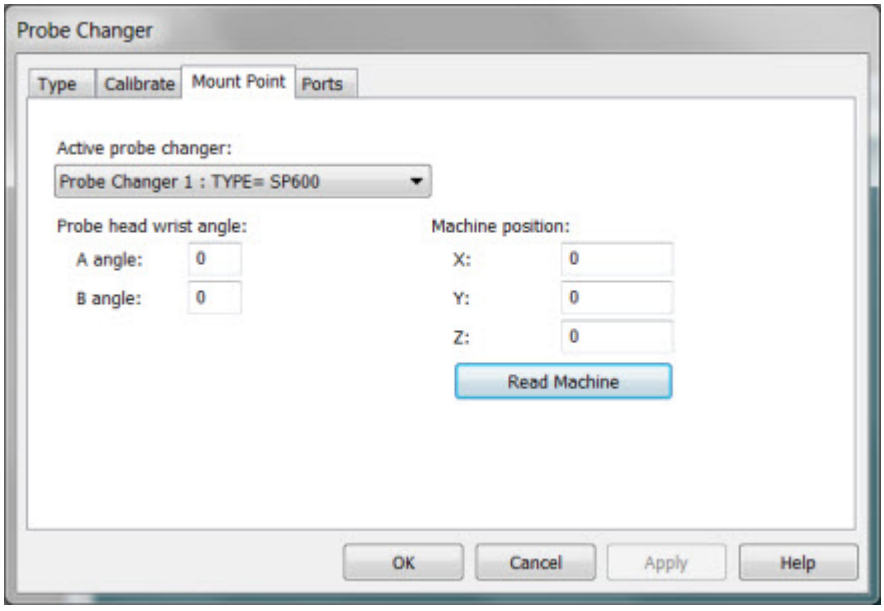
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point

The mount point for your SP600 Probe Changer is the location in front of your probe changer where the machine moves to before it picks up or drops off a probe. You should determine a location that avoids collisions with the probe changer or part.

To define the mount point for your probe changer:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=SP600**.

3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Typically, but not always, these values are 0 and 0, respectively. You need to use a

calibrated probe rotation that ensures the probe has the ability to move in and out of the probe rack during the necessary steps of the probe rack calibration procedure.

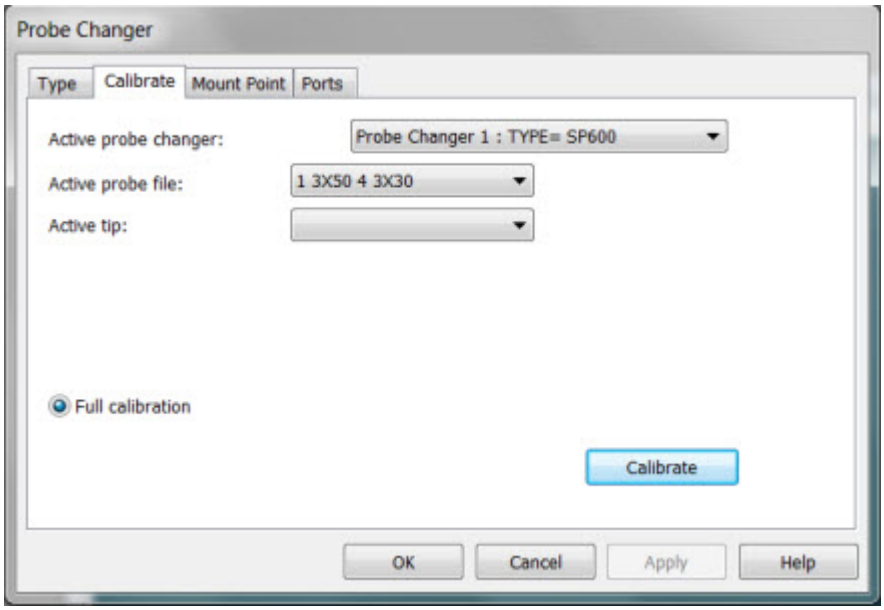
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine position** values with the current position. You may also manually type these values.
6. To save your changes, click **Apply**.

You are now ready to begin calibration. The next step starts the calibration process.

Step 3 - Prepare for Calibration

To begin the calibration procedure for the SP600 Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **TYPE=SP600**.
3. In the **Active tip** list, select the tip ID to be used for calibration. This is usually **T1A0B0**.
4. Click the **Calibrate** button.

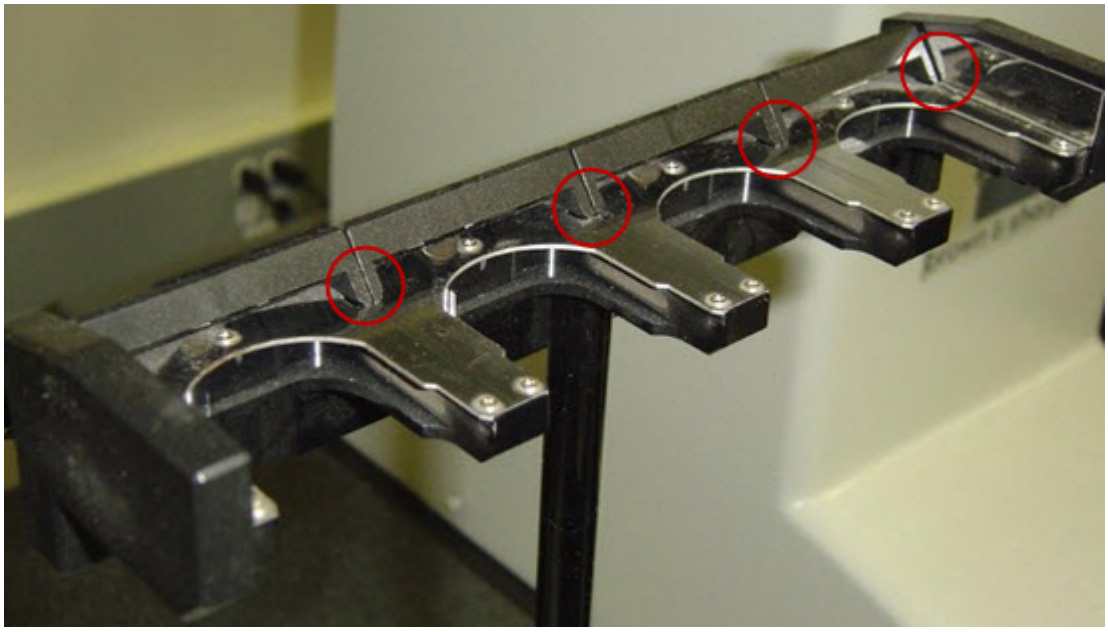
Once you click **Calibrate**, the following message box appears:

PC-DMIS Message

Open all lids and remove any probe from port 1.

When ready, please take a hit in the center of the vertical face between ports 1 and 2.

1. To open all the lids, push them all the way back and down. Plastic extensions on each side of the lid slide into corresponding holes in the rack surface to keep the lids open.
2. With the lids open, remove all of the modules and styli by sliding them forward and out of the ports.



SP600 Probe Changer showing empty ports held open by plastic extensions on each side of the port cover that slide into corresponding holes in the rack surface (indicated by the red circles)

3. When you are ready to take the first measurement, click **OK**.

In the next step, you take the first manual hit.

Step 4 - Take the First Manual Hit

PC-DMIS moves you through the process of measuring three manual hits through a series of message boxes. Three hits are required to fully define the orientation of this rack in the machine's XY plane. It is not necessary to align the rack to any single axis, as the manual hits identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

The first manual hit is taken in the XY plane, on the vertical face in front of the rack between port 1 and port 2.

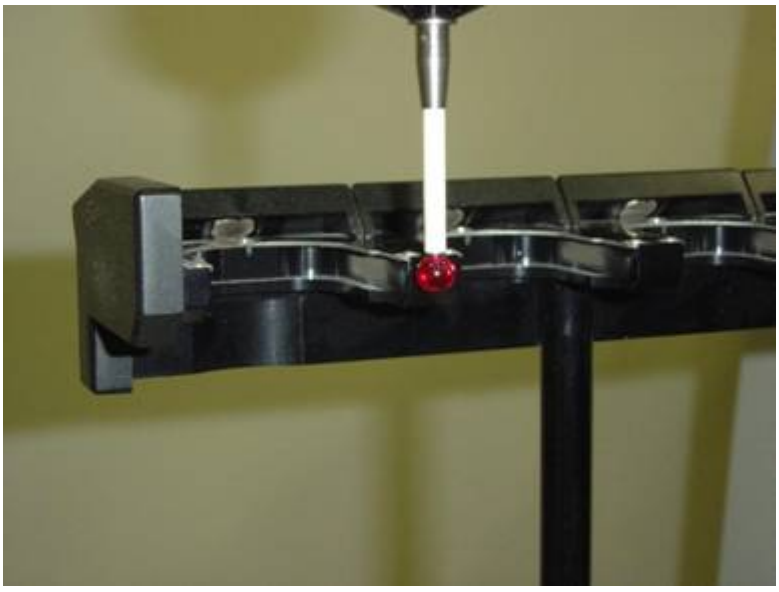
The prompt for the first hit on the front face is:

PC-DMIS Message

Open all lids and remove any probe from port 1.

When ready, please take a hit in the center of the vertical face between ports 1 and 2.

Use your machine's jog box to measure the first hit on the front face between port 1 and port 2.



In the next step, you take the second manual hit.

Step 5 - Take the Second Manual Hit

For the SP600 Probe Changer, you will take the second manual hit in the Z direction, on the horizontal face on top of the rack between port 2 and port 3.

The prompt for the second hit on the top is:

PC-DMIS Message

Please take a hit on top of the metallic docking plate in the middle between ports 2 and 3.

Use your machine's jog box to measure the second hit on the top metal surface between port 2 and port 3.

In the next step, you take the third manual hit.

Step 6 - Take the Third Manual Hit

For the SP600 Probe Changer, you will take the third and final manual hit in the XY plane, on the vertical face in front of the rack between port 3 and port 4.

The prompt for the third hit in front is:

PC-DMIS Message

Please take a hit in the center of the vertical face between ports 3 and 4. After completing this hit, DCC calibration will begin.

Use your machine's jog box to measure the third hit on the front vertical surface, between port 3 and port 4 as shown below.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

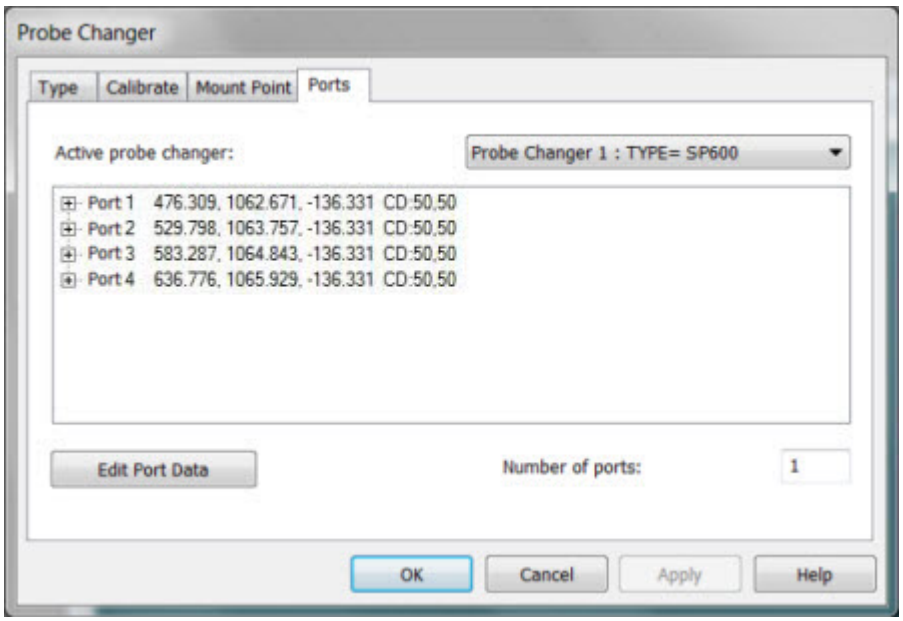
This set of two hits establishes the location and orientation for the tool changer.

At this point, the system goes into DCC Mode and measures a series of hits required to accurately locate and orient the rack.

In the next step, you review the calibration results.

Step 7 - Review the Calibration Results

1. For the SP600 Probe Changer, once the DCC measuring finishes, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). This tab displays the calibration information for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Some things to watch for are the position of the rack and the spacing of the ports. Any significant deviations from these expectations may be due to a bad hit.

    - This rack is not necessarily aligned parallel to any axis of the CMM.
    - However, the X and Y values should show equal spacing between ports, roughly 53.5 mm apart.
    - Similarly, the Z values should be nearly identical, since the ports are all at the same height.

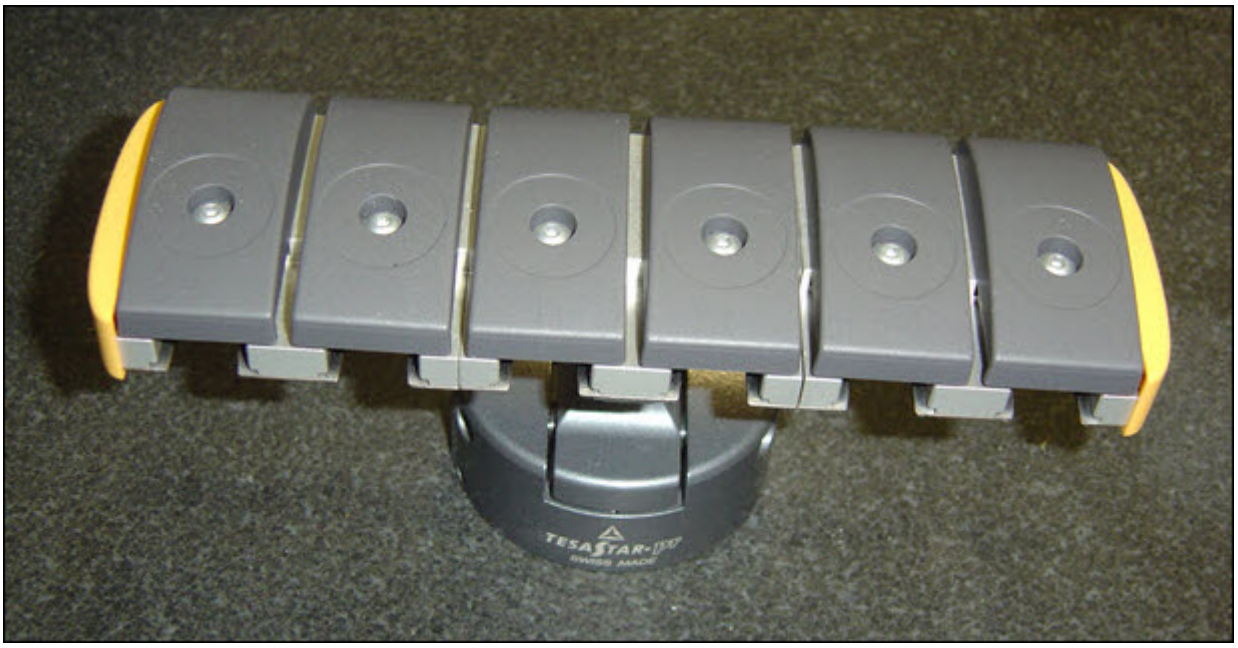
The results during measurement routine execution are:

1. Probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the "unload" port (the port that used to contain the currently used probe entity), which pushes the lid backwards. The current "puck", a black conical piece of hardware that connects to the bottom of the probe body, is held in place by the rack while the probe body lifts away to detach.
3. From there, the probe moves over to the next "load" position. The magnetic connection automatically engages to load the new module.
4. It then moves back to the rack’s mount point.
5. From there, it continues to measure.

Calibrating the TESASTAR-PR / HR-P Probe Changer

The calibration process for the TESASTAR-PR / HR-P Probe Changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the TESASTAR-PR / HR-P Probe Changer.

Inserts or extensions are not used for any of the port positions.



TESASTAR-PR / HR-P Probe Changer rack

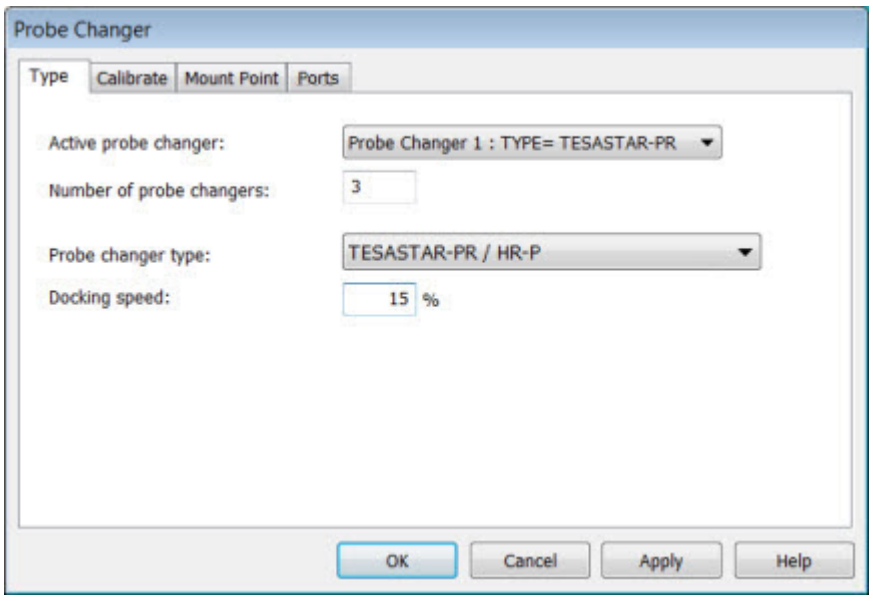
> This probe changer MUST be mounted on the machine table parallel to a machine axis, such as the X or Y axis. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the TESASTAR-PR / HR-P Probe Changer

To select the TESASTAR-PR / HR-P Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **TESASTAR-PR / HR-P**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** box, select the list item that represents the probe changer you want to define. If this is the first probe changer, it appears as **Probe Changer 1: TYPE=None**.

5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.

6. In the **Docking speed** box, type a value. A value between 15-20% is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the mount point and wrist angles for your system.

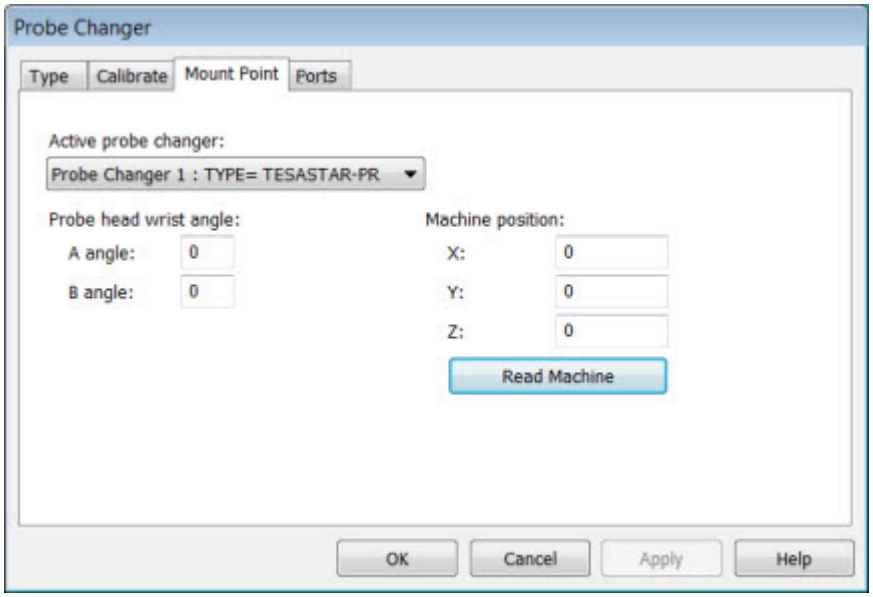
Step 2 - Define the Mount Point and Wrist Angle

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that

avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=TESASTAR-PR**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use a calibrated probe rotation to ensure the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your machine's jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine position** values with the current position. You can also manually type these values.
6. To save your changes, click **Apply**.

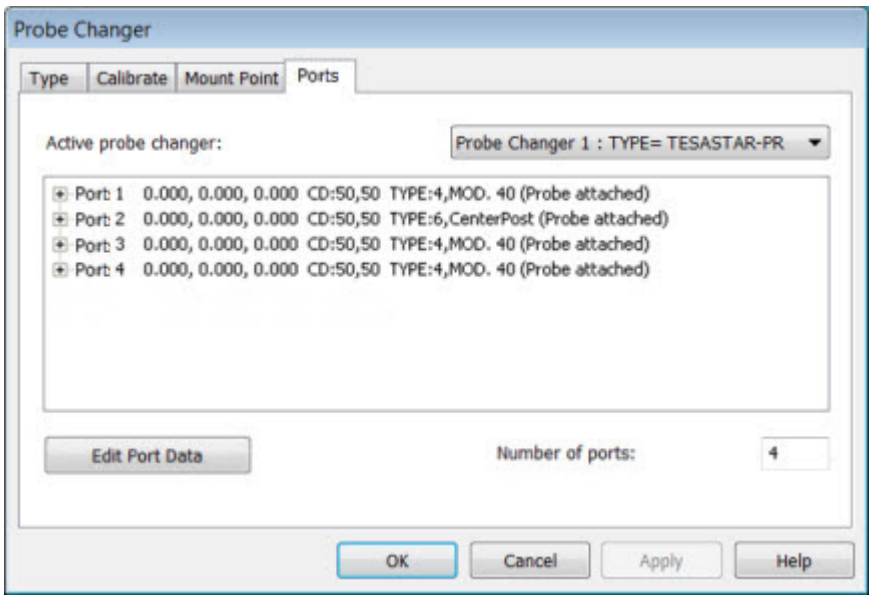
In the next step, you define the ports for your system.

Step 3 - Define the Ports

The final step prior to the actual calibration of the TESASTAR-PR / HR-P Probe Changer is to define the ports on the probe changer.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

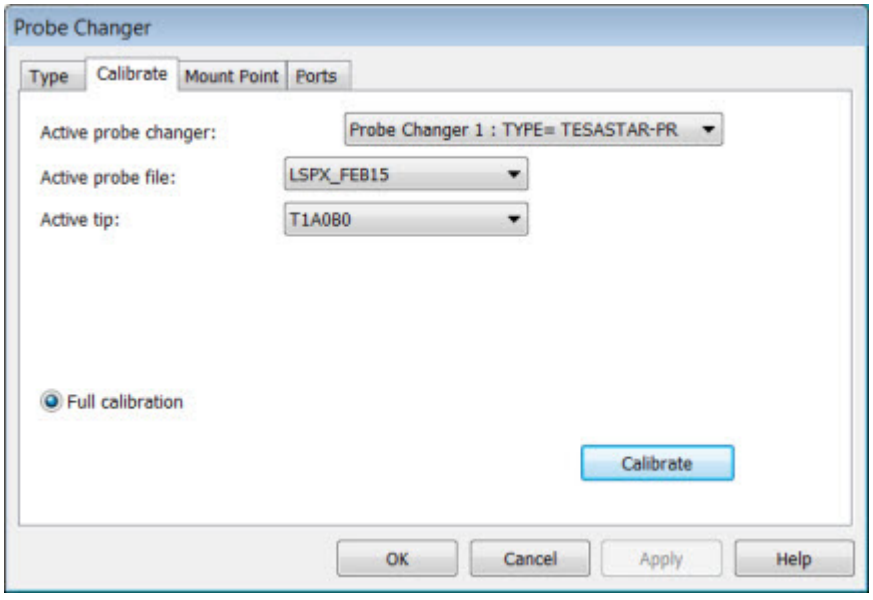
2. In the **Active probe changer** list, select **TYPE=TESASTAR-PR**.
3. In the **Number of ports** box, type the number of ports that you want to define. This should match the number of physical ports on the probe changer.
4. To save your changes, click **Apply**.

The next step starts the calibration process.

Step 4 - Prepare for Calibration

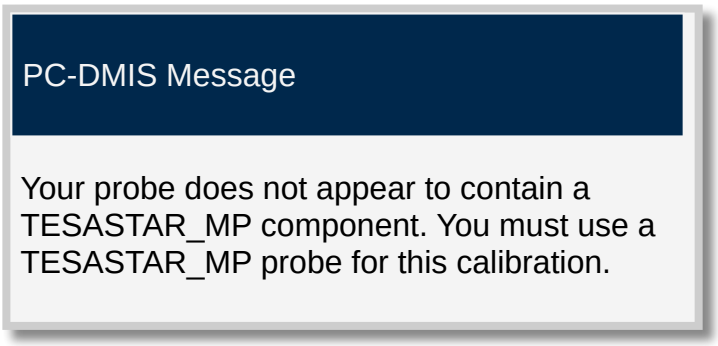
To begin the calibration process for the TESASTAR-PR / HR-P Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

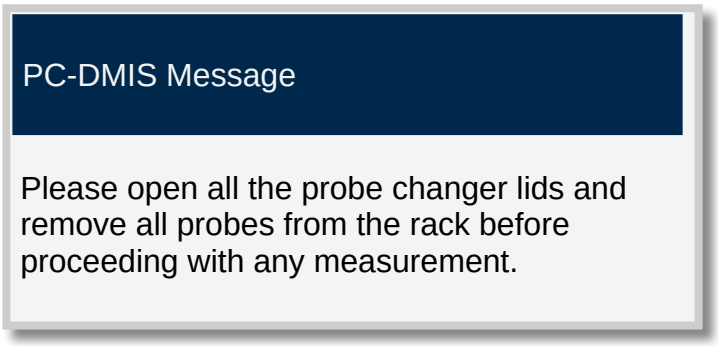
2. Click the **Calibrate** button. If the proper probe assembly is not currently loaded, the following message appears:



   If this is the case, it is necessary to load the proper probe components and start the calibration again.

3. The **Active probe file** and **Active tip** items both default to the current measurement routine settings. If needed, change these to the probe and tip that you intend to use for the probe changer calibration.

4. Open the lids on each port by sliding them back until they click into place.



   PC-DMIS prompt to open the port lids before calibration.

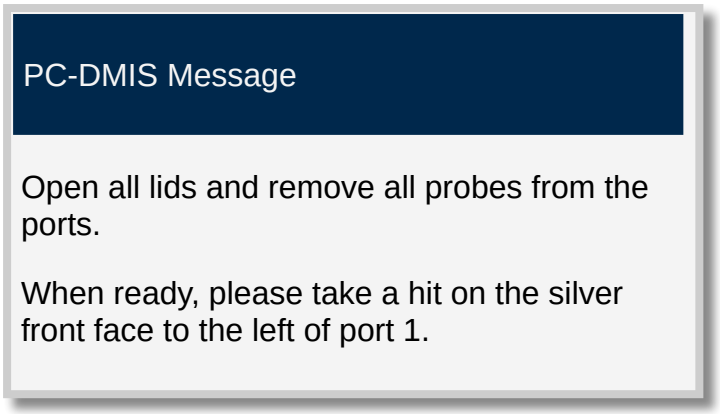
   Example showing the open port lids before calibration.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

Once you start the calibration procedure for the TESASTAR-PR / HR-P Probe Changer, the ports must be emptied of all probes. You will take the first hit on the front face to the left of the first port.

PC-DMIS prompts you through the process of measuring the manual hits through a series of message boxes. The following message prompts you to open all the lids and take the first hit:

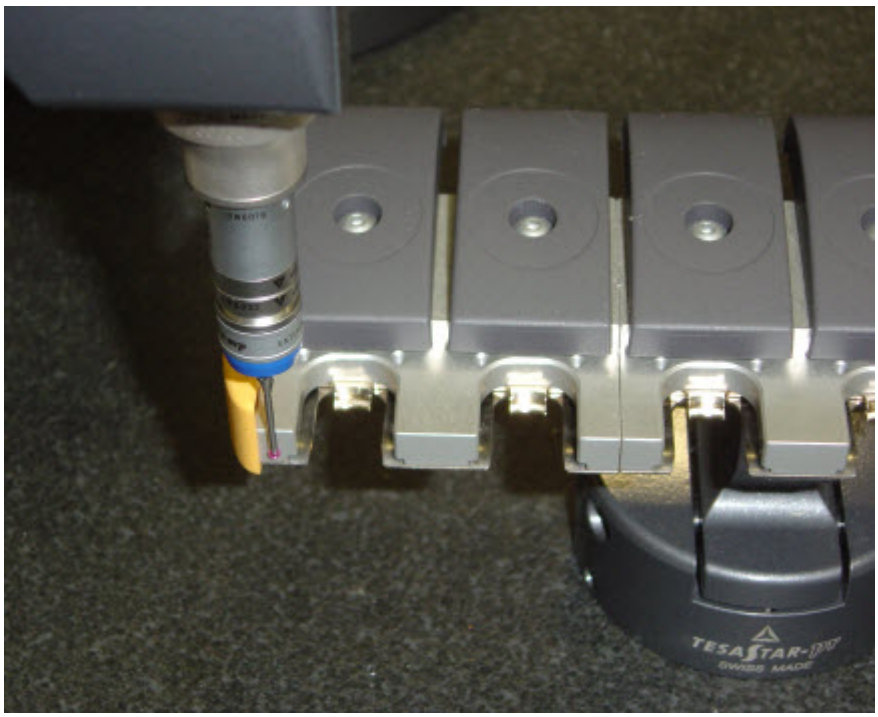


Prompt for the first manual hit in the front face, left of port 1

1. When you are ready to take the first measurement, click **OK**.
2. Remove all of the modules and styli by sliding them forward and out of the ports.

   Three hits are required to fully define the orientation of this rack in the machine’s XY plane. It is not necessary to align the rack to any single axis, as the manual hits will identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

   The first manual hit is taken in the XY plane, on the vertical face in the back of port #1.
3. Use your machine's jog box to measure the first hit on the front face to the left of port #1.

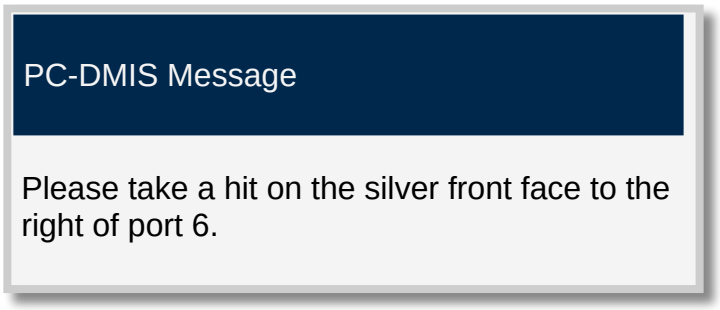


Location of first manual hit during calibration

In the next step, you take the second manual hit.

Step 6 - Take the Second Manual Hit

You will take the second manual hit for the TESASTAR-PR / HR-P Probe Changer on the front face to the right of the last port. Once the first manual hit is completed, the following message box appears:



Prompt for the second manual hit inside the last port

Use your machine's jog box to measure the second hit on the front surface to the right of the last port.

Location of second manual hit during calibration

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

After it takes this hit on the back of the last port, PC-DMIS drives in DCC Mode and takes some additional measurements working backwards from the last port towards the first port. When this is complete, it positions the probe above the rack and provides instructions to proceed.

In the next step, you take the third manual hit.

Step 7 - Take the Third Manual Hit

You will take the third manual hit for the TESASTAR-PR / HR-P Probe Changer on the top surface to the right of the last port. Once you complete the second manual hit, the following message box appears:

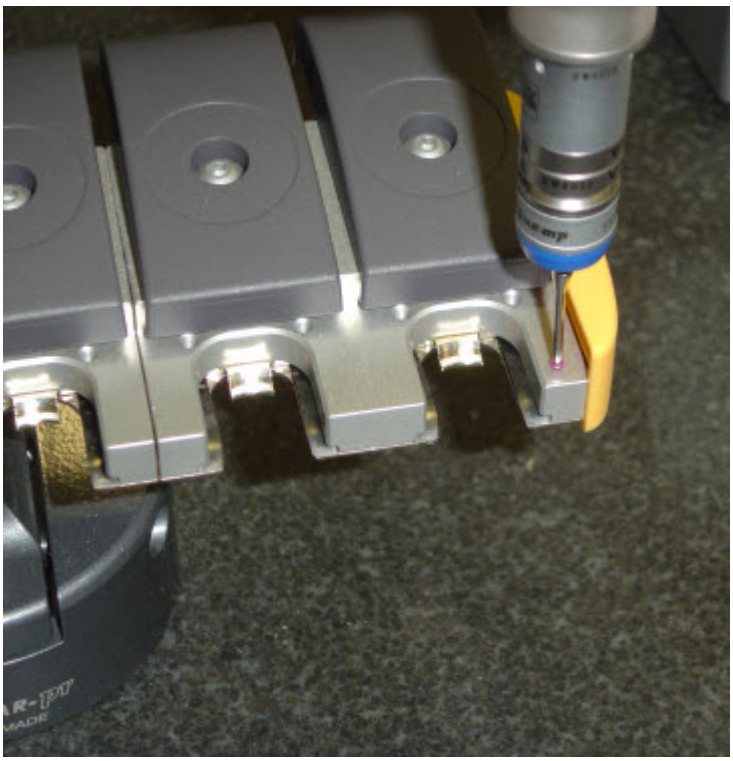
PC-DMIS Message

Please take a hit on top on the silver surface to the right of port 6.

After completing this hit, DCC calibration will begin

Prompt for the third manual hit on the top to the right of the last port

Use your machine's jog box to measure the third hit on the top surface to the right of the last port.



Location of third manual hit during calibration

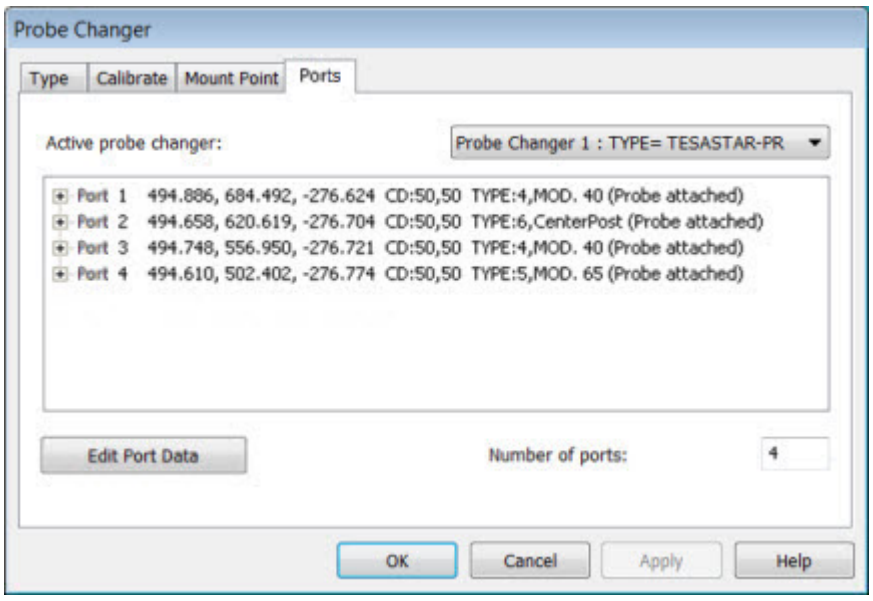
WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

After it takes this hit on the back of the last port, PC-DMIS drives in DCC Mode and takes some additional measurements. It works backwards from the last port towards the first port. When this is complete, the **Calibrate** tab appears.

The next step is to review the calibration results after DCC calibration is complete.

Step 8 - Review the Calibration Results

1. When the calibration of the TESASTAR-PR / HR-P Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). The calibration information now exists for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

    - This rack is not necessarily aligned parallel to any axis of the CMM. However, the X and Y values should show equal spacing between ports (roughly 30 mm apart).
    - The Z values should be nearly identical since the ports are all at the same height.

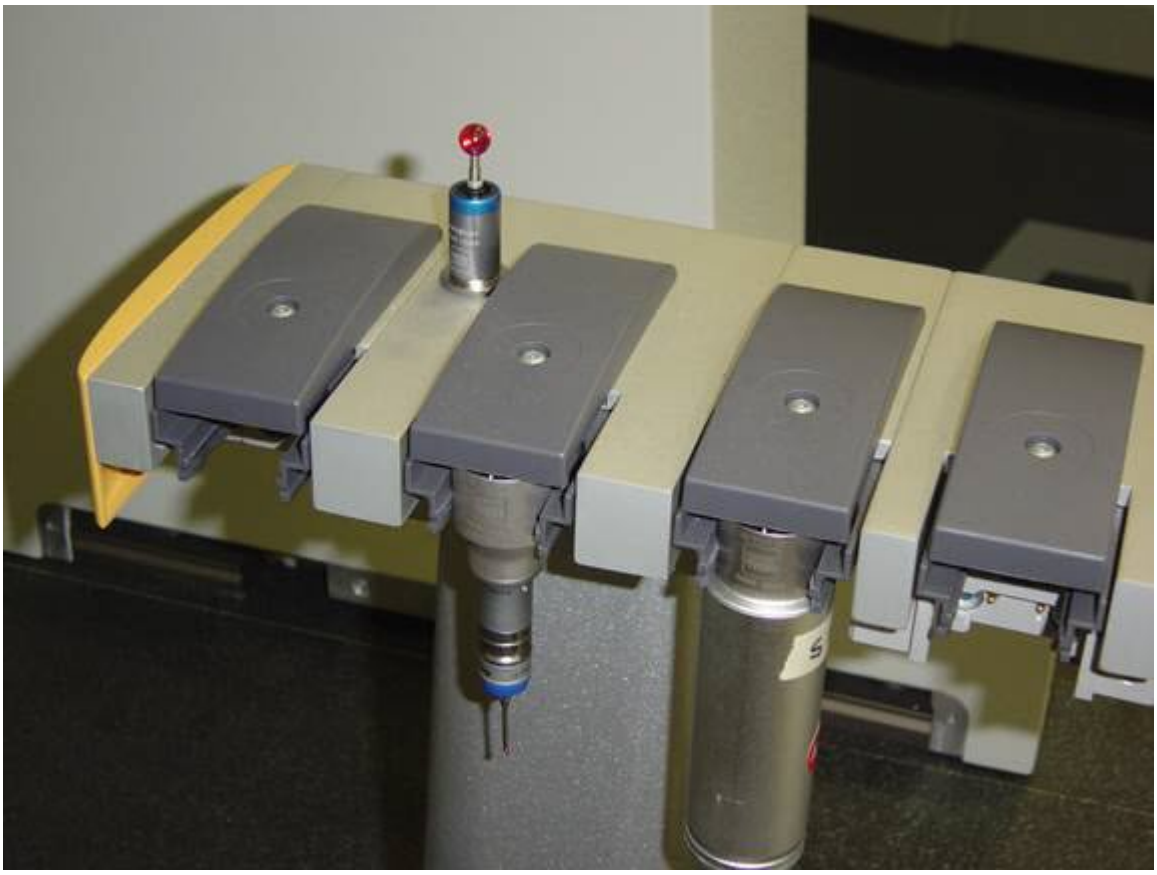
The results during measurement routine execution are:

1. Probe entities that are added to each of the ports are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the "unload" port to drop off the current probe.
3. The current probe remains in the port while the probe body lifts away to detach.
4. From there, the probe moves over to the next "load" position and down onto the new probe. This automatically engages the new module.
5. It then moves back out of the port and over to the rack’s mount point.
6. From there, it continues to measure.

Calibrating the TESASTAR-R / HR-R Probe Changer

While this calibration process is specific to the TESASTAR-R / HR-R Probe Changer, it is similar to the steps required for any probe changer that PC-DMIS supports.

This documentation expects that you are positioning the probe changer so that the center post of the probe changer is vertical to the CMM table with the ports parallel to the CMM table:

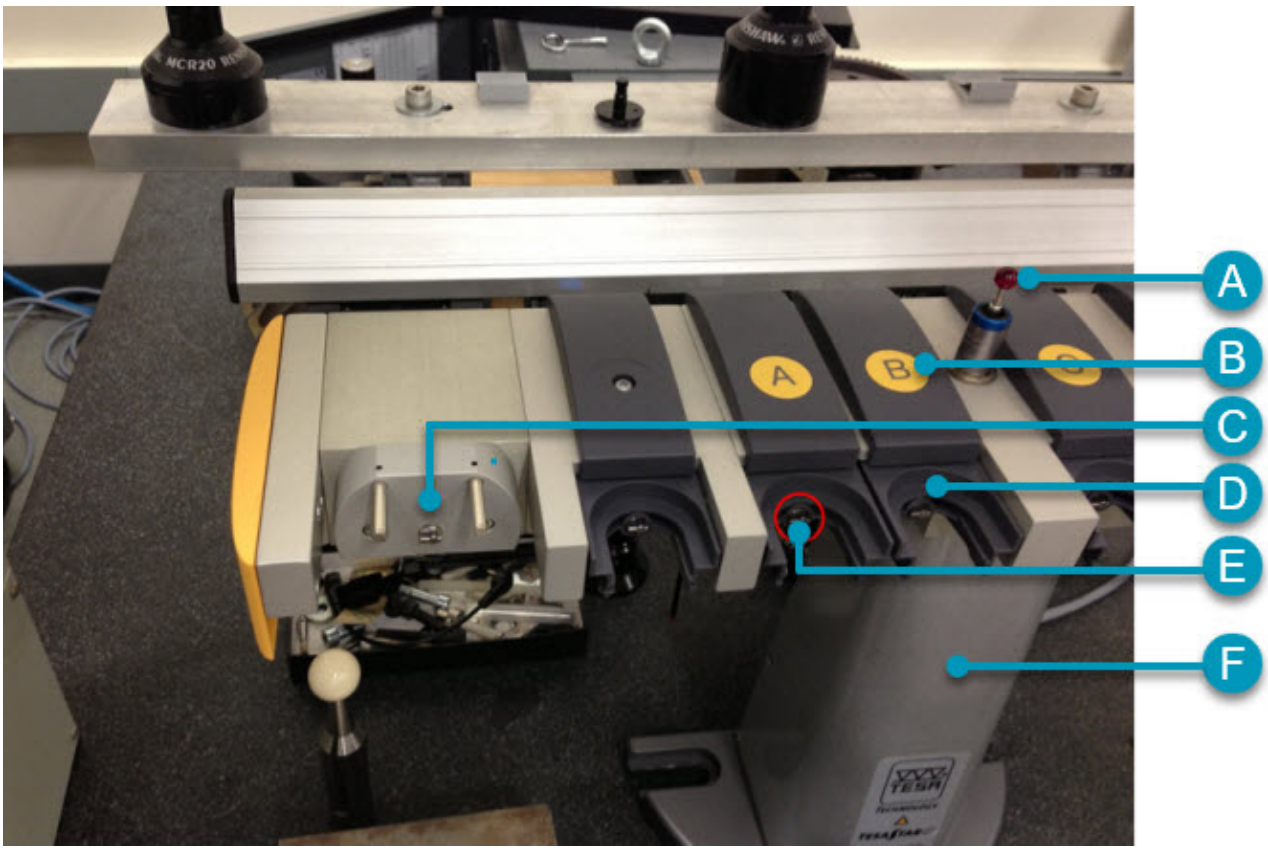


TESASTAR-R / HR-R Probe Changer

This probe changer MUST be mounted on the machine table parallel to a machine axis, such as the X or Y axis. For mounting instructions, consult the documentation that came with your probe changer.

You should mount all HD ports on the end of the tool changer rack. If you would like to mount the HD ports in the middle of the rack, you must set the `UseTCVerticalClearanceForHD_TESASTAR-R` entry to True in the PC-DMIS Settings Editor. This will allow the wrist to move up during tool changes to avoid the HD ports. For more information, see the PC-DMIS Settings Editor documentation.

Probe Changer Components



TESASTAR-R / HR-R Probe Changer components:
***A** - Datum Sphere*
***B** - Lid/Cover*
***C** - HD Port*
***D** - Port*
***E** - Key*
***F** - Center Post*

Wrist Types

The procedure for calibrating the TESASTAR-R / HR-R Probe Changer varies depending on the type of wrist. This documentation defines the wrists as follows:

- HD wrist with an HDKJ extension:

- Wrist:

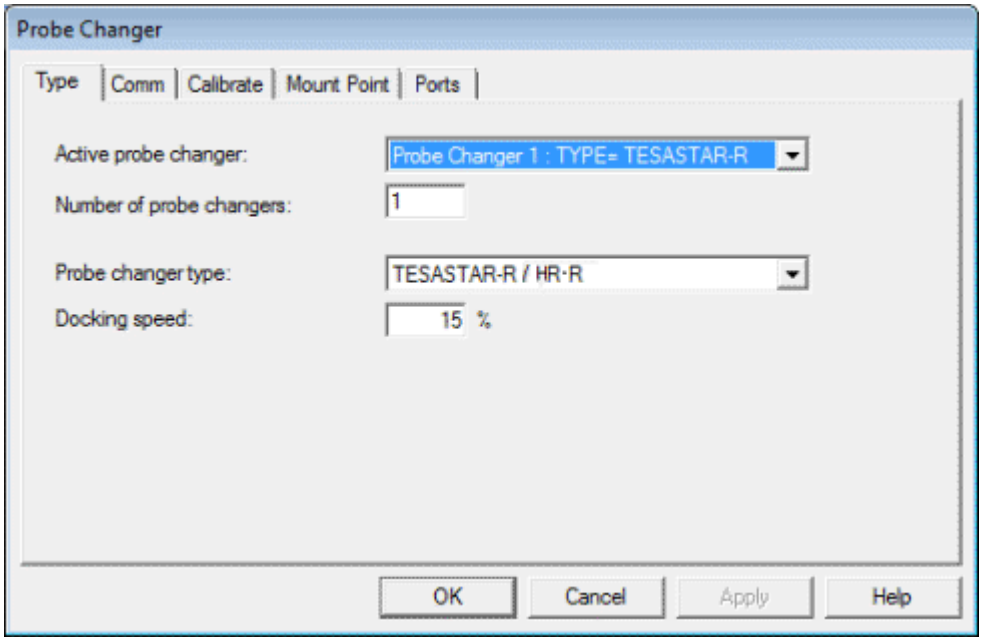
Calibration

To calibrate your probe changer:

Step 1 - Select the TESASTAR-R / HR-R Probe Changer

To select the TESASTAR-R / HR-R Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **TESASTAR-R / HR-R**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** box, select the list item that represents the probe changer you want to define. If this is the first probe changer, this shows **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of probe changers that you want to define.
6. In the **Docking speed** box, type a value. A value between 15-20% is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

> To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

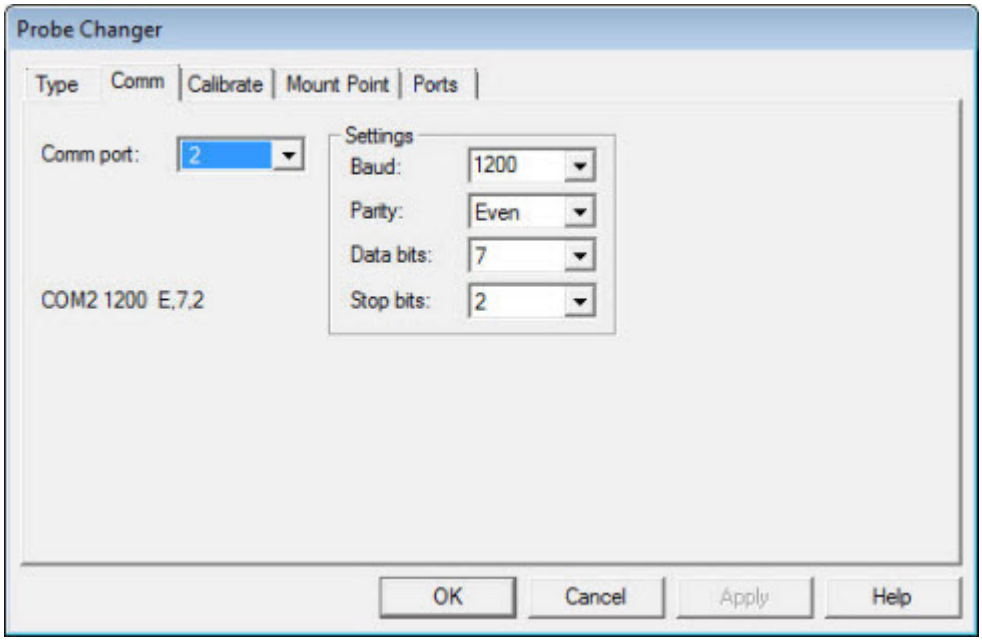
In the next step, you define the communications port.

Step 2 - Define the Communications Port

You must enable communications with the TESASTAR-R / HR-R Probe Changer before you can use it to calibrate it or automatically change probes.

To define the communications port:

1. Select the **Comm** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Comm Tab

2. For specific settings to use, refer to the documentation on the probe changer itself and then fill in the information on the **Comm** tab.
3. Click **Apply** and then **OK**.
4. Close and restart PC-DMIS to initialize these new parameters and enable PC-DMIS to communicate with the probe changer.

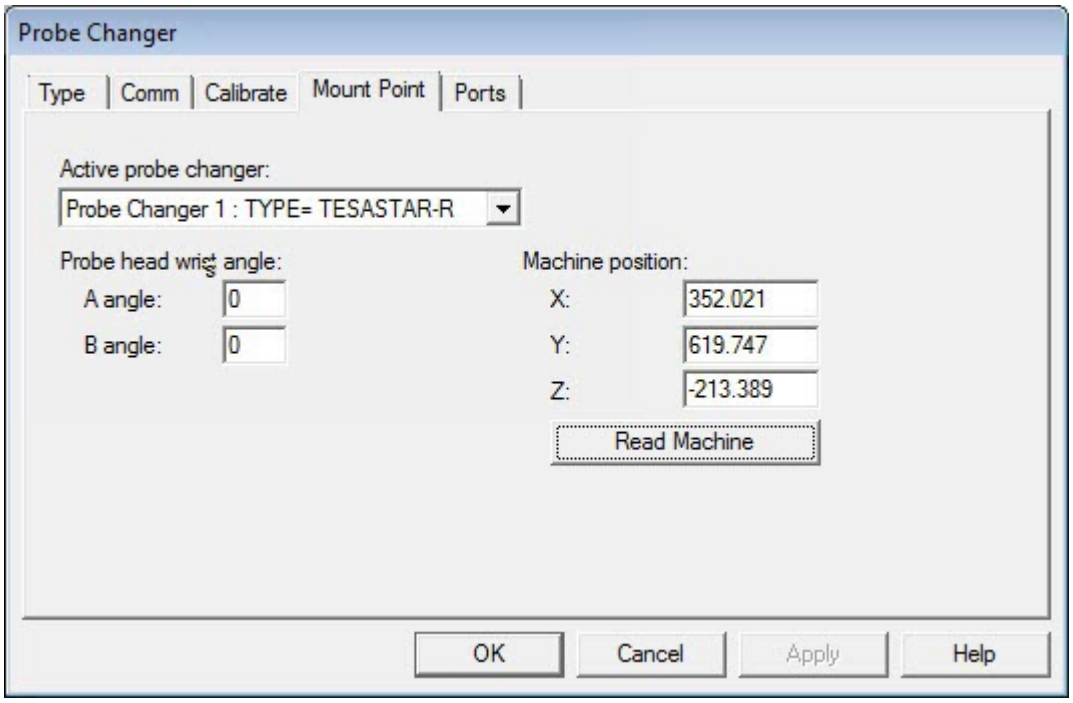
In the next step, you define the mount point and wrist angles for your system.

Step 3 - Define the Mount Point and Wrist Angle

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=TESASTAR-R**.

3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use the wrist angles that align the key port on the joint on the probe component with the key blade in the back of the port that turns the key to lock/unlock the joint.

   HD joints and regular joints both contain keys that have to be locked/unlocked. When assembled together, the key on the HD joint and the key on the regular joint are normally 180 degrees opposite of each other.

   - If your rack has HD ports, the wrist angles used should be the ones that align the key on the HD joint with the back of the port.
   - If your rack does NOT have HD ports, the wrist angles should be the ones that align the key on the regular joint with the back of the port.

4. Use your jog box to manually move your machine to the desired mount point position.

5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine position** values with the current position. You can also manually type these values.

6. To save your changes, click **Apply**.

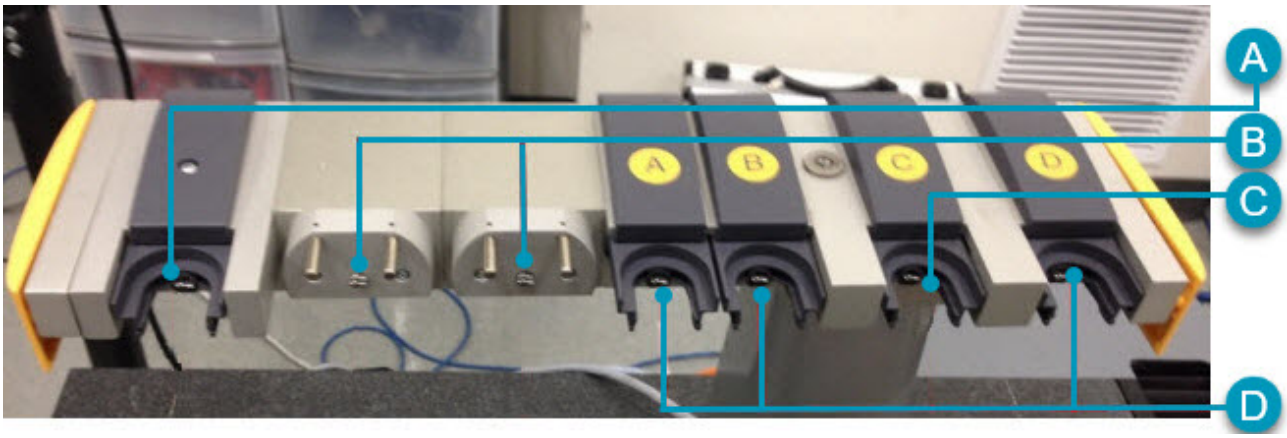
It is important that you correctly set the mount point angles before calibrating.

In the next step, you define the ports for your system.

Step 4 - Define the Ports

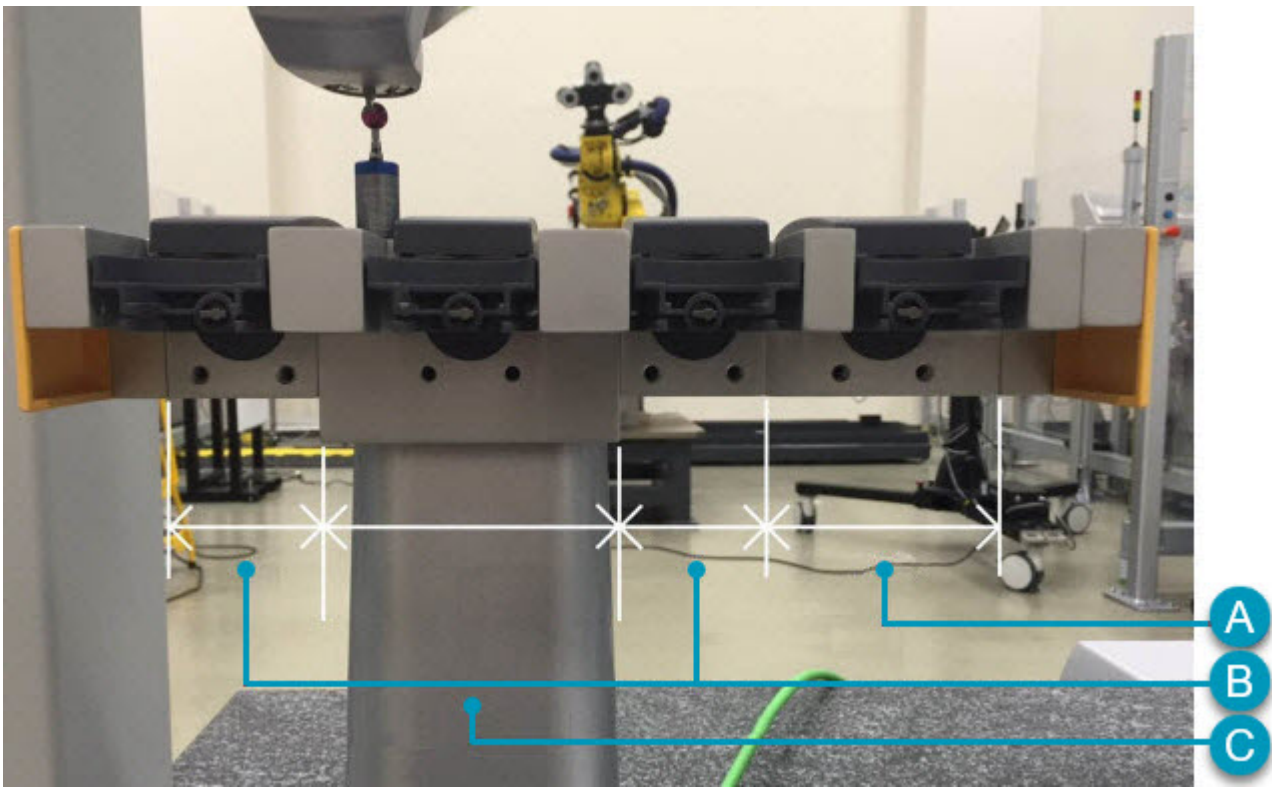
The final step prior to the actual calibration of the TESASTAR-R / HR-R Probe Changer is to define the number and configuration of the ports on the probe changer. For an example of how to define ports, see "Example of Defining Ports" at the end of this topic.

The TESASTAR-R / HR-R rack has four types of ports:



Types of ports:
***A** - MOD .65: Has a circular port*
***B** - HD: Has a port with two pins for changing the HD components*
***C** - Center Post: Has a circular port and is mounted on the probe changer post*
***D** - MOD .40: Has a circular port*

Another image of the ports on the TESASTAR-R / HR-R rack appears below:

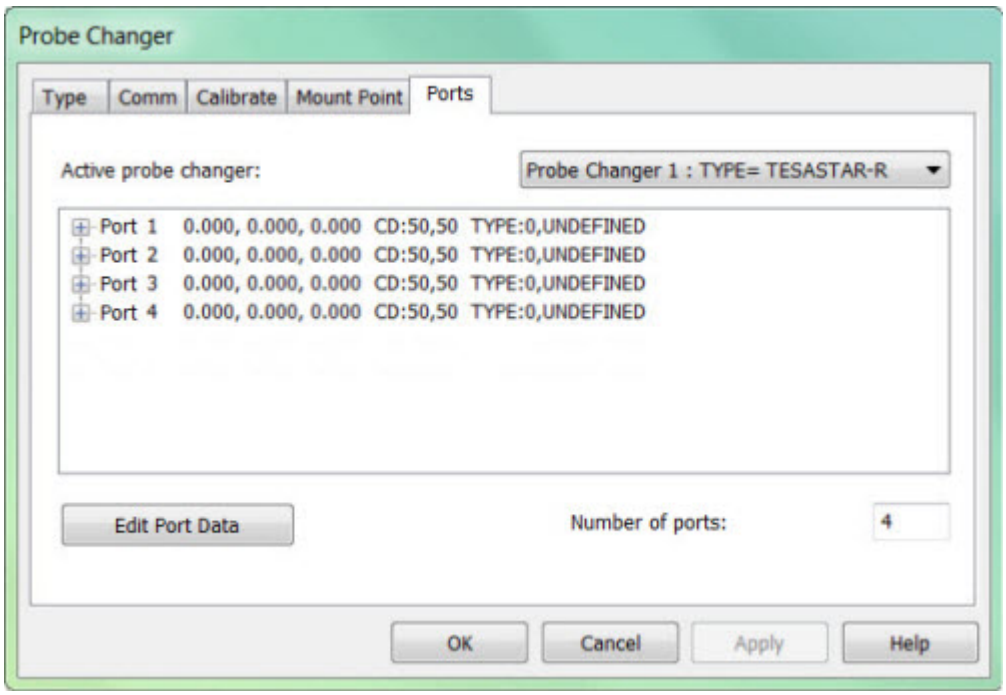


Types of ports:
***A** - MOD .65*
***B** - MOD .40*
***C** - Center Post*

TKJ extension-only ports have an extension-only location associated with each empty HD extension.

Defining the Number of Ports

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):

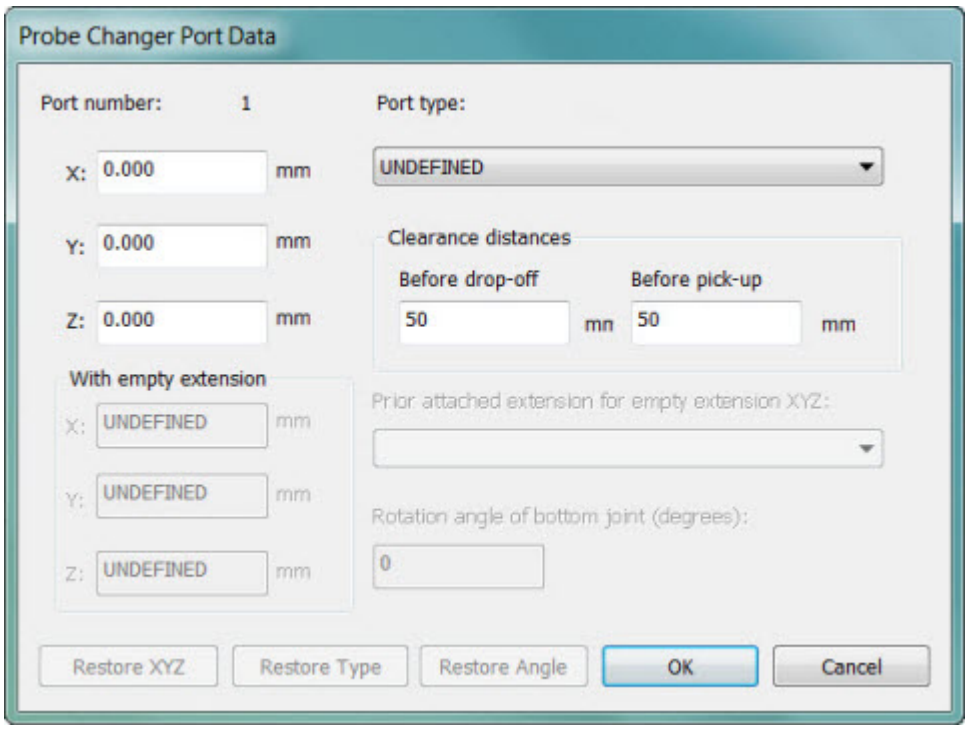


Probe Changer dialog box with undefined ports

2. In the **Active probe changer** list, select **TYPE=TESASTAR-R**.
3. In the **Number of ports** box, type the number of ports that you want to define. This should match the number of physical ports on the probe changer.
4. Click **Apply** to store this parameter.

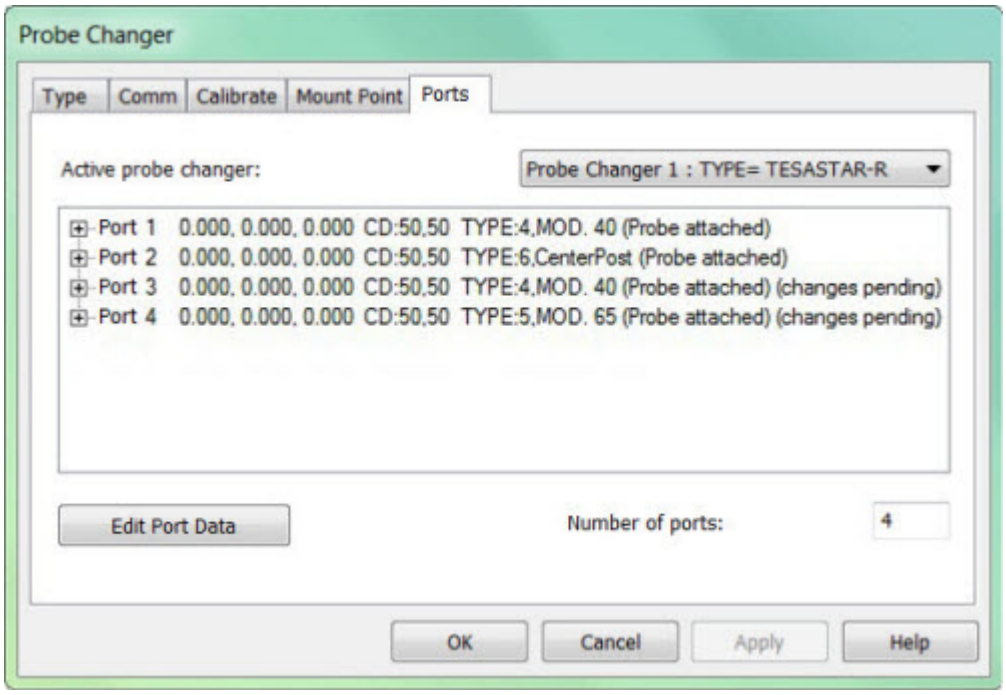
After you define the number of ports, you need to define the configuration of each port. You can do several different configurations depending on the size and location of the port hardware. For the proper configuration of each port, consult your probe changer's documentation.

1. Select the port item in the list and then click **Edit Port Data** to display the **Probe Changer Port Data** dialog box:



Probe Changer Port Data dialog box

2. In the **Probe Changer Port Data** dialog box, if **Port type** shows UNDEFINED, select the appropriate type for the port.
3. Click **OK** to return to the **Probe Changer** dialog box.
4. Define any remaining ports. As you define each port type, the port description in the list shows the "(changes pending)" text appended.
5. Click **Apply**. The text is removed on all of the ports.



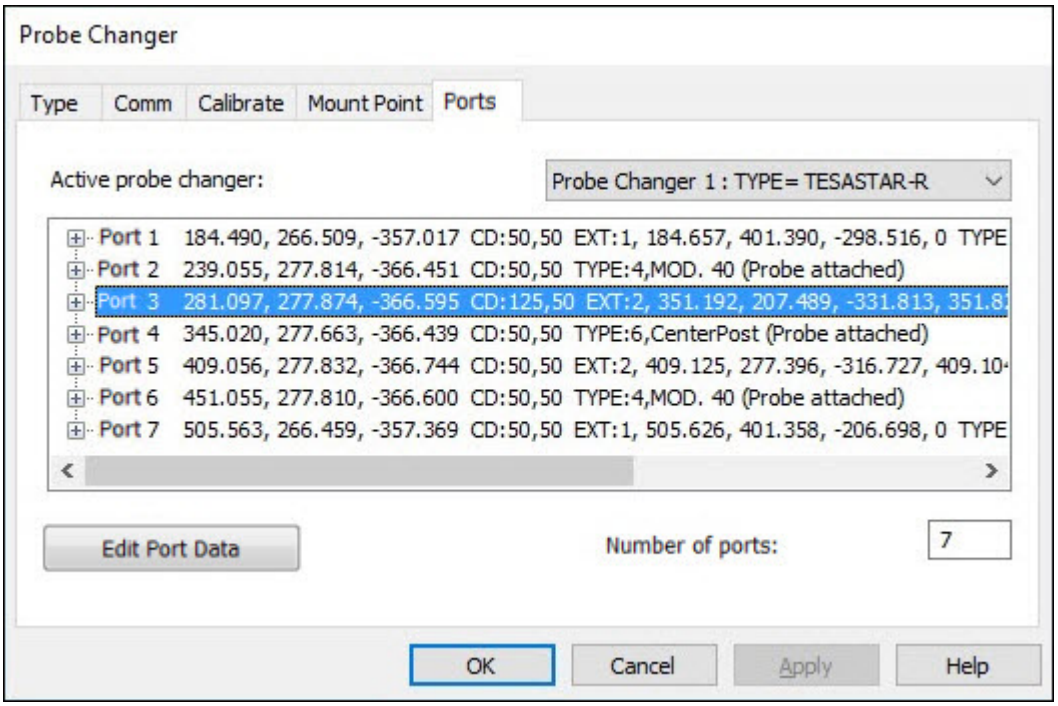
Probe Changer dialog box with all ports defined

If you plan to use any of the ports to hold probe extensions, you need to define them in this step before you proceed. Ports that hold extensions require additional steps in the calibration process where you need to take additional hits on the datum sphere with and without the extension.

You are now ready to begin calibration.

Example of Defining Ports

The example below shows seven defined ports:



Example of Probe Changer dialog box with seven defined ports

In this example, ports 1 and 7 are HD extension-only ports. Ports 3 and 5 are TKJ extension-only ports. The one in port 3 has a 90-degree rotation of the bottom joint.

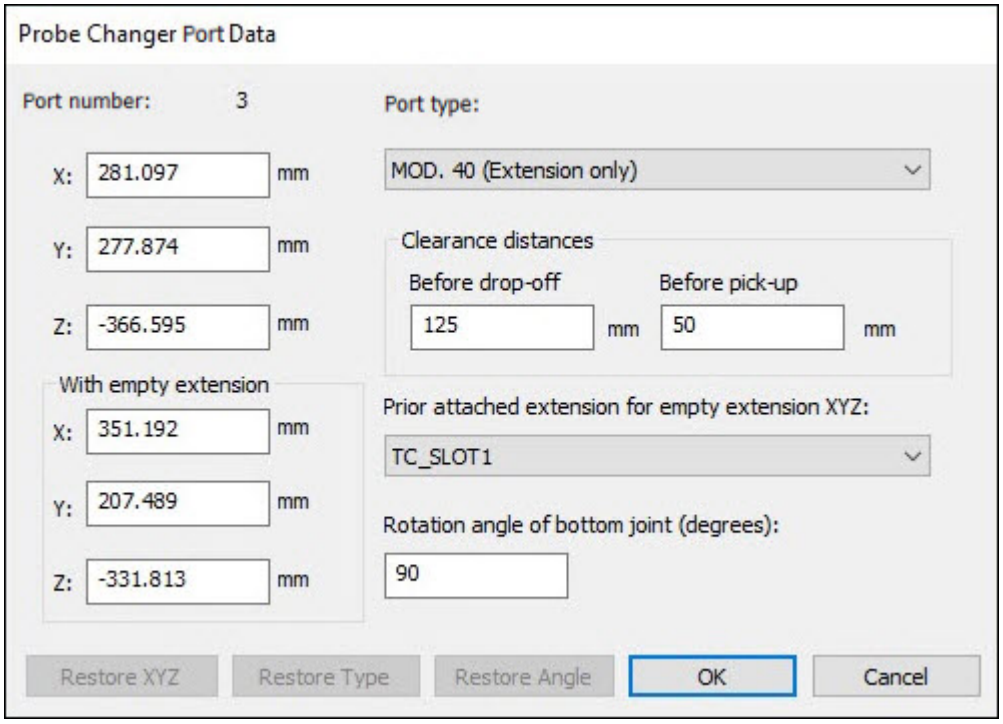
Note the following:

- If you combine HD extensions with regular TKJ extensions by loading an HD extension and then use it to load a TKJ extension, and if you have more than one HD port with an extension that can be used with the TKJ extension, the incremental offset gained by loading that TKJ extension varies slightly depending on which HD extension you used it with.
- If you use multiple HD extensions with TKJ extensions, the calibration requires that you attach and measure the TKJ extension with each HD extension. After calibration, there is a separate XYZ for the "With empty extension" result for each HD extension it was used with. You can select which result to view or edit by selecting the port associated with the HD extension it was used with.

- If you do not have more than one HD extension, or if you have not yet calibrated to allow the multiple results to become available, no selection is available.

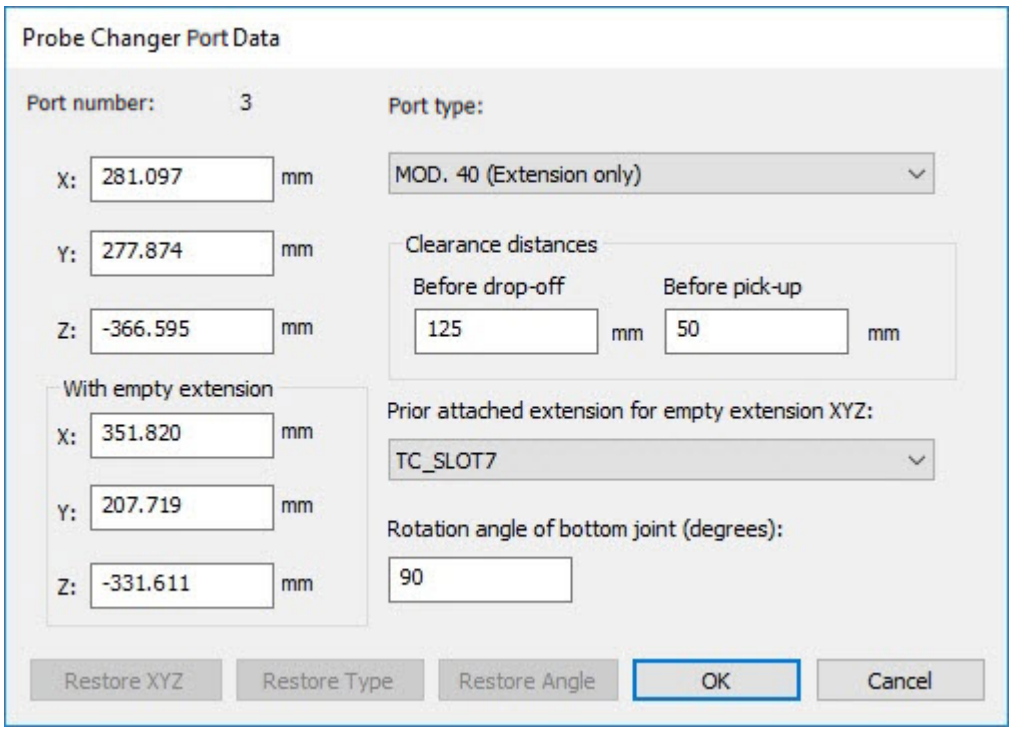
If you select port 3 (a TKJ extension-only port) and then click the **Edit Port Data** button, the **Probe Changer Port Data** dialog box appears. In the **Prior attached extension for empty extension XYZ** list, you would select the port with the results you want to view.

- The image below shows the results with the HD extension from port 1:



Example of Probe Changer Port Data dialog box

- The image below shows the results if you use the same TKJ extension with the HD extension in port 7:



Example of Probe Changer Port Data dialog box

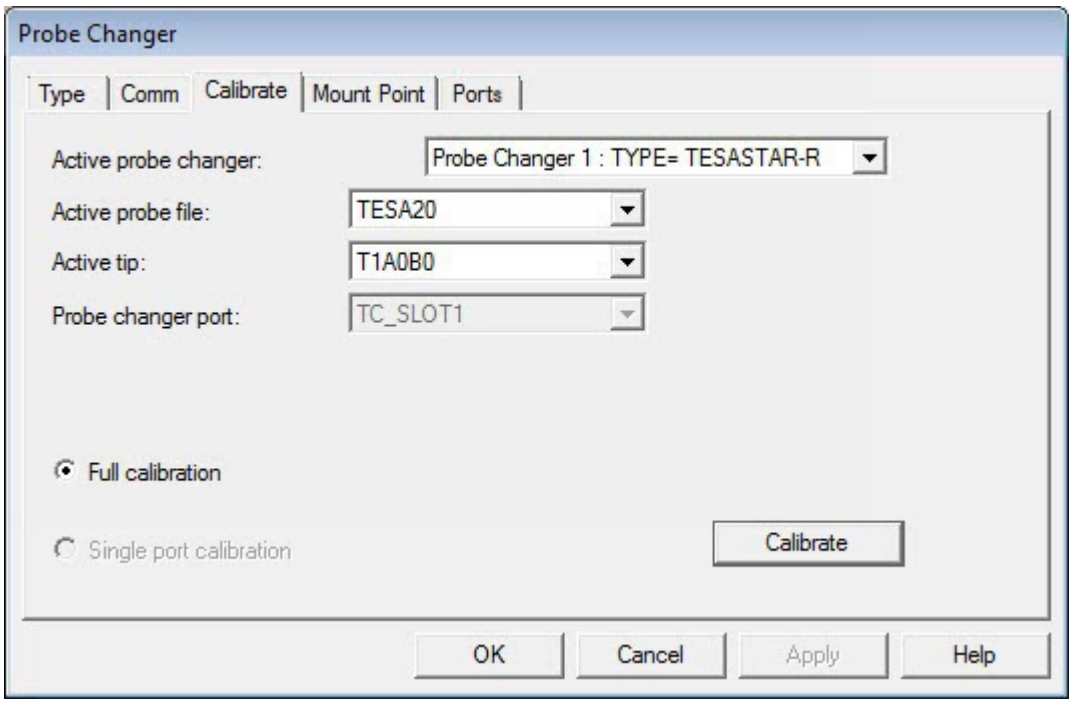
The differences in the extension-only location for the port 1 and port 7 scenarios above are relatively small. However, they are significant for correct positioning for probe change purposes.

The next step starts the calibration process.

Step 5 - Prepare for Calibration

To begin the calibration process for the TESASTAR-R / HR-R Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):

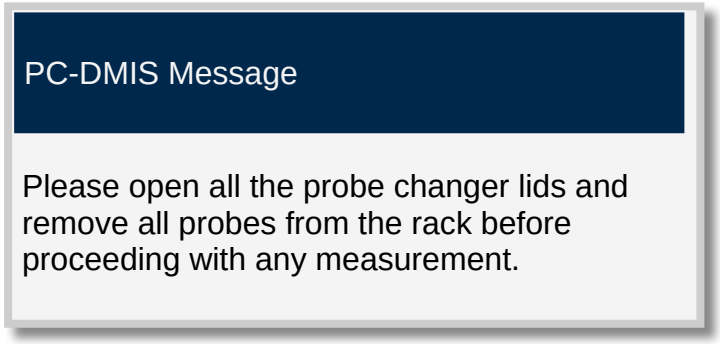


Probe Changer Dialog Box - Calibrate Tab

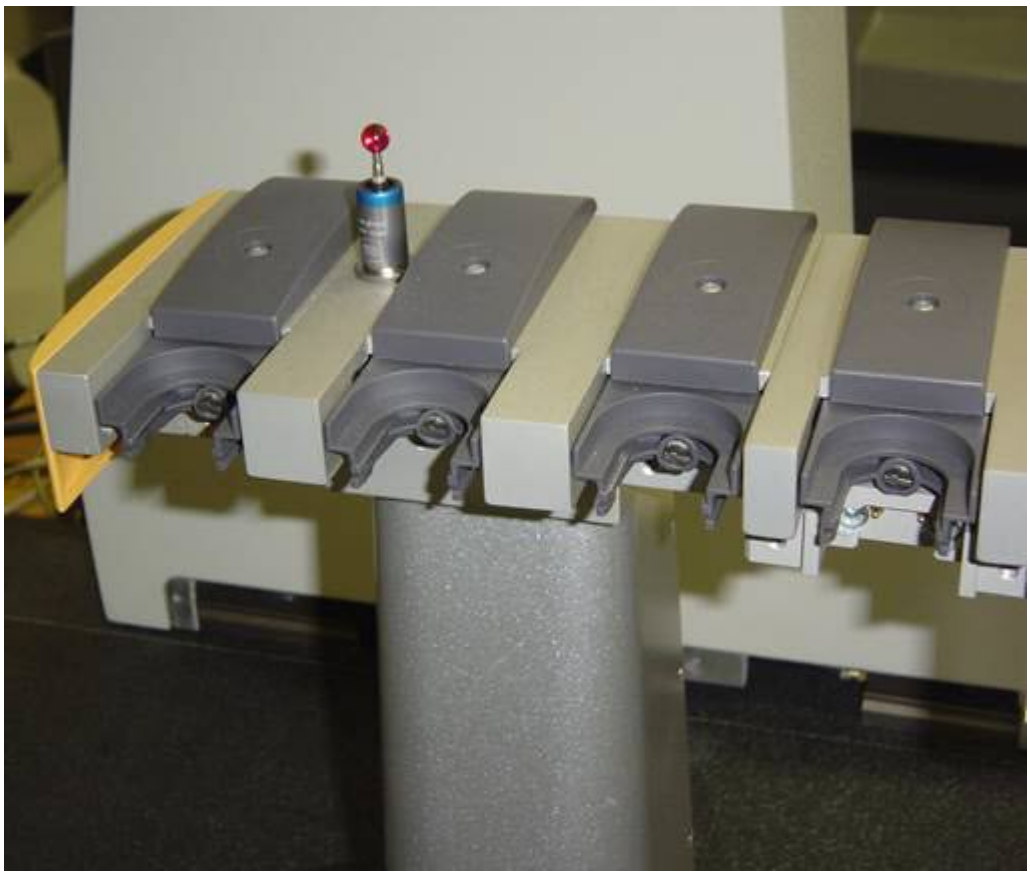
2. Click the **Calibrate** button.

3. The **Active probe file** and **Active tip** items both default to the current measurement routine settings. If needed, change these to the probe and tip that you intend to use for the probe changer calibration.

4. Open the lids on each port by sliding them back until they click into place.



PC-DMIS prompt to open the port lids before calibration.



Example showing the open port lids before calibration.

In the next step, you take the first manual hit.

Step 6 - Take the First Manual Hit

In this step for the TESASTAR-R / HR-R Probe Changer, PC-DMIS prompts you to take a manual hit on the port. This is the first of two hits required to fully define the orientation of this rack in the machine's XY plane. You do not need to align the rack to any single axis since the manual hits identify any rotation involved.

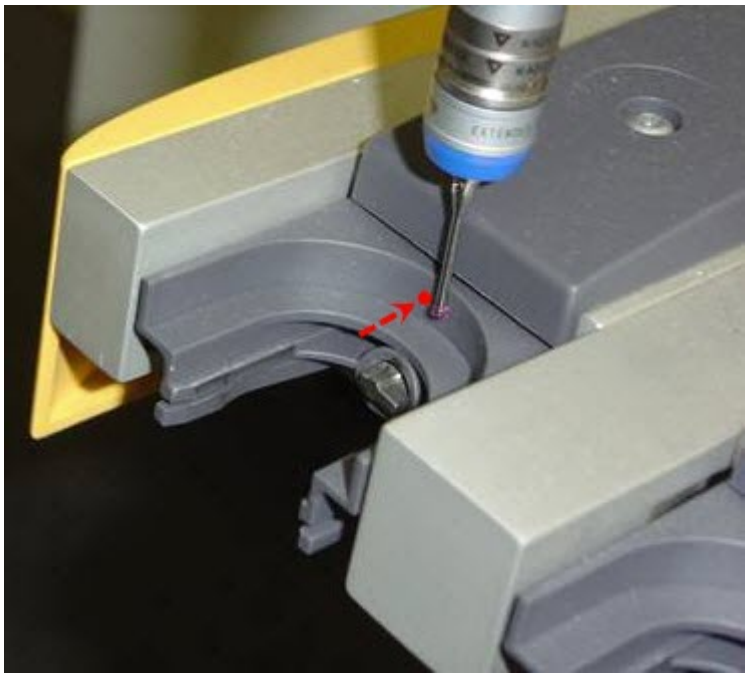
PC-DMIS moves you through the process of measuring the manual hits through a series of message boxes:

PC-DMIS Message
Please take a hit in the back of the circle for port 1. The hit should be taken in the middle of the larger circle above and behind the key.

Prompt for the first manual hit in the back of the first port

1. If any modules or styli reside in the ports, remove them now by sliding them forward and out of the ports.

2. When you are ready to take the first measurement, click **OK**.
3. Follow the prompts and take the needed hits as directed in the instructions for each hit. If you have HD ports, you will take the first hit on the **first regular port** and **not** on any HD port that may precede it in the rack.



Use your machine's jog box to carefully take the first manual hit on the vertical face of the radius in the back of the first port, as indicated by the bottom photo

In the next step, you take the second manual hit.

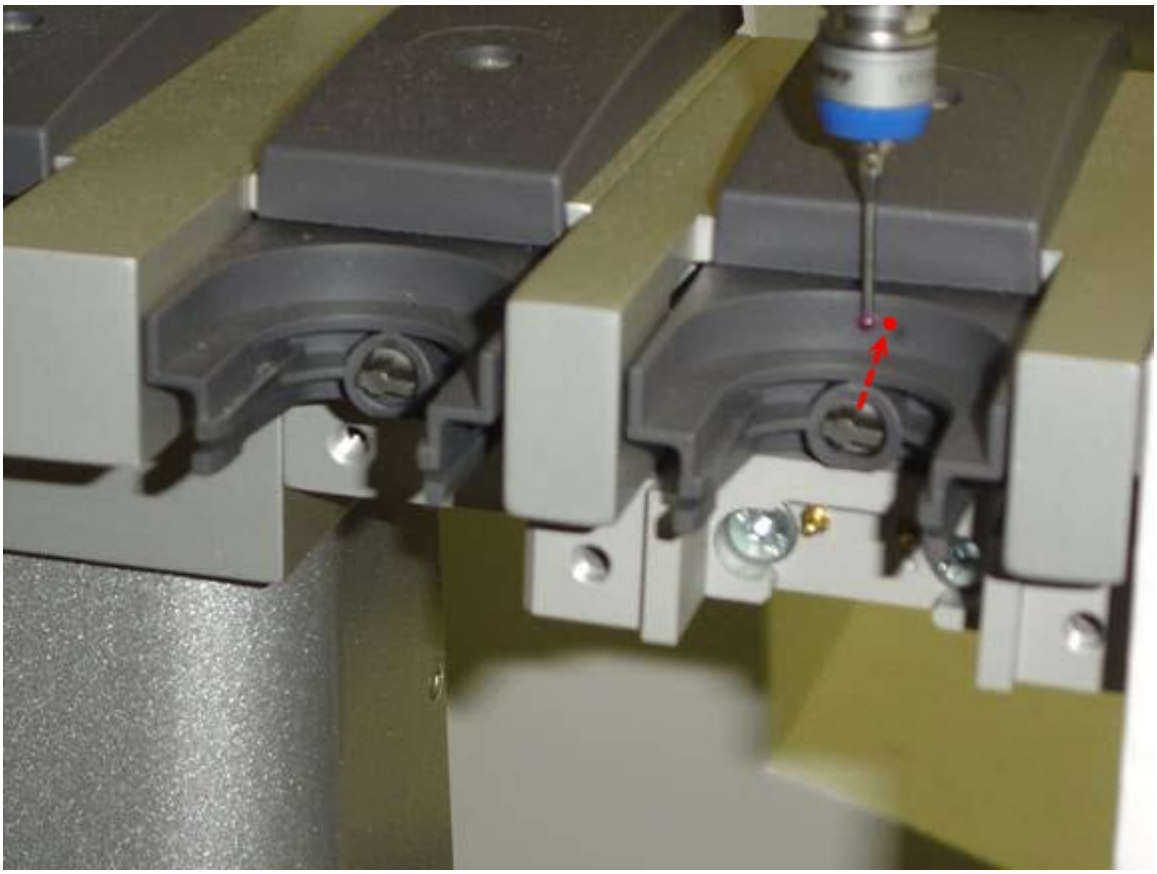
Step 7 - Take the Second Manual Hit

In this step for the TESASTAR-R / HR-R Probe Changer, take the second manual hit on the vertical face of the radius in the back of the last regular port.

PC-DMIS Message
Please take a hit in the back of the circle for port 4.

The hit should be taken in the middle of the larger circle above and behind the key.

Prompt for the second manual hit inside the last regular port



Use your machine's jog box to carefully take the second manual hit on the vertical face of the radius in the back of the last regular port, as indicated by the bottom photo

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

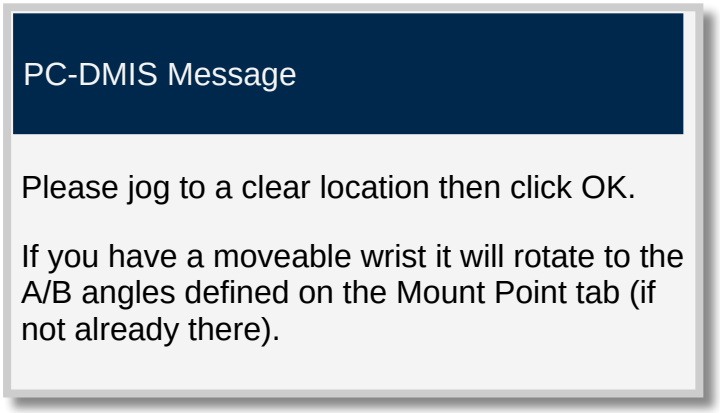
After you take this hit on the back of the last regular port, PC-DMIS drives in DCC Mode and takes additional measurements, working backward from the last regular port toward the first port.

When this is complete, and if you are using an HD wrist, it takes additional measurements working backwards from the last HD port towards the first HD port. When this finishes, PC-DMIS positions the probe above the rack and provides instructions to proceed.

In the next step, you determine the location of the datum sphere.

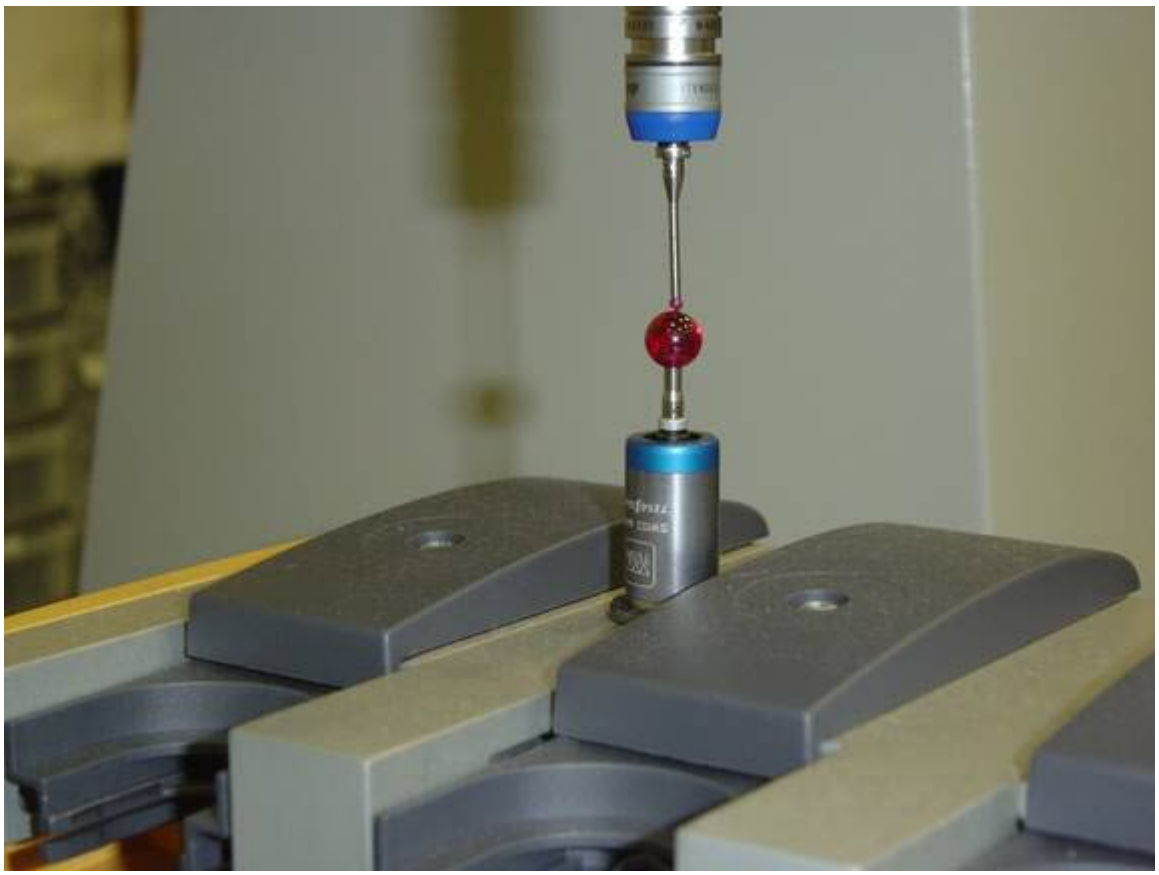
Step 8 - Take the Probe Hit on the Datum Sphere

After the initial DCC port measurement finishes, you must accurately determine the location of the datum sphere. You do this by taking a series of hits on the datum sphere itself. A prompt appears to move the probe clear of the rack in case it needs to turn the wrist:



PC-DMIS prompt to move the probe to a safe location before wrist adjustment.

1. Move the probe to the safe location and then click **OK**.
2. When the **Execution** dialog box prompts you, use the bottom of the stylus ruby to take a manual hit on the top of the datum sphere ruby:



Use your machine's jog box to take a manual hit on the top of the datum sphere

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Once the hit is taken, the system moves into DCC and measures three sample hits on the datum sphere. It then measures five more hits for the sphere feature.

Finally, it lifts above the sphere and displays the following prompt:

PC-DMIS Message

Please jog to a clear location then click OK.

If you have a moveable wrist it will rotate to the A/B angles defined on the Mount Point tab (if not already there).

PC-DMIS prompt to prepare for wrist rotation.

In the next step, you take the AutoJoint hit on the datum sphere.

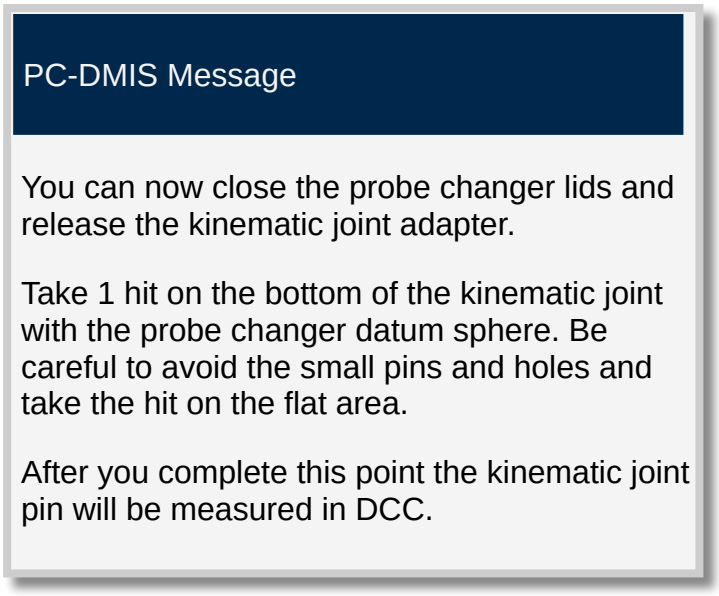
Step 9 - Take the AutoJoint Hit on the Datum Sphere

To determine the overall probe length, the system needs one hit on the probe and then another hit on the quick-connect kinematic joint (or AutoJoint). Instructions to remove the probe for this second hit on the datum sphere appear. The instructions for the hit varies if you have an HD wrist. Both sets of instructions are included in this section.

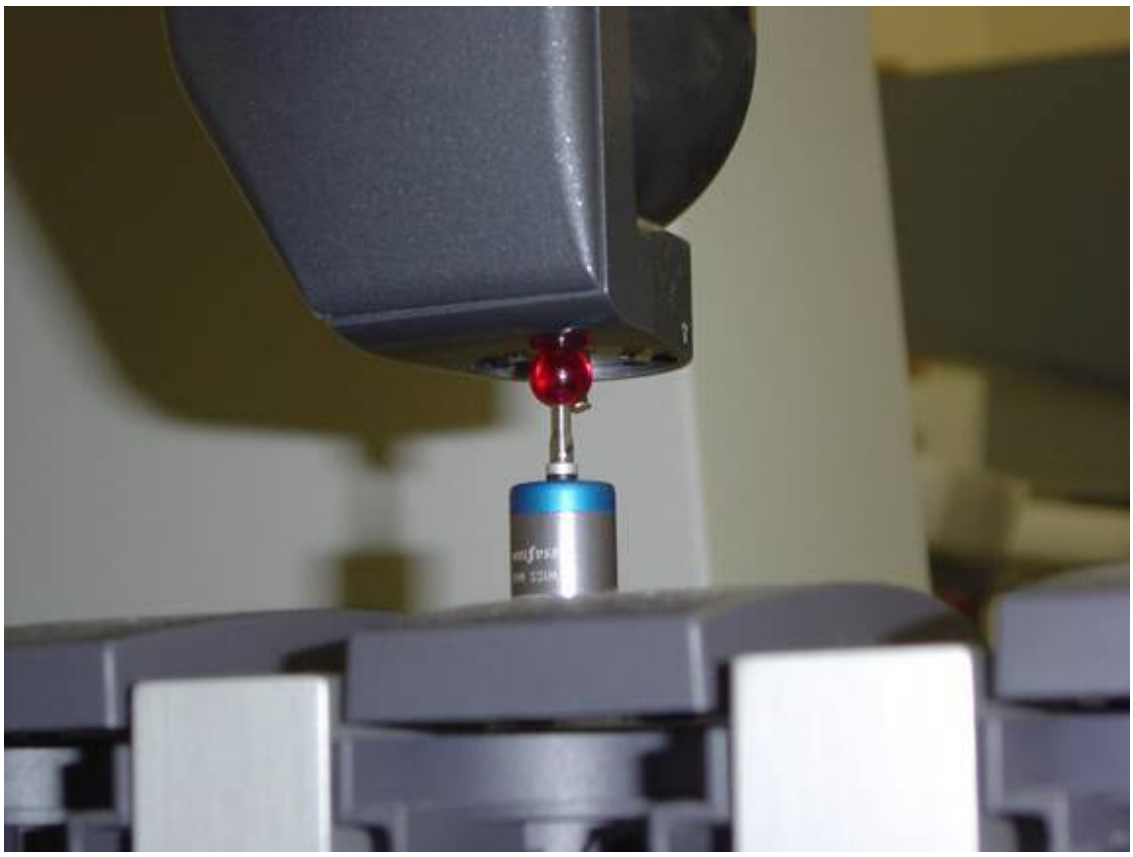
WARNING: At different times throughout this procedure, the machine moves in DCC mode. When this happens, to avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

1. Take a manual hit as instructed.

    - For the AutoJoint:



PC-DMIS prompt to take a hit on the datum sphere with AutoJoint.



Use your machine's jog box to position the AutoJoint connection above the datum sphere and take a manual hit on the top of the datum sphere

- For the HD AutoJoint:

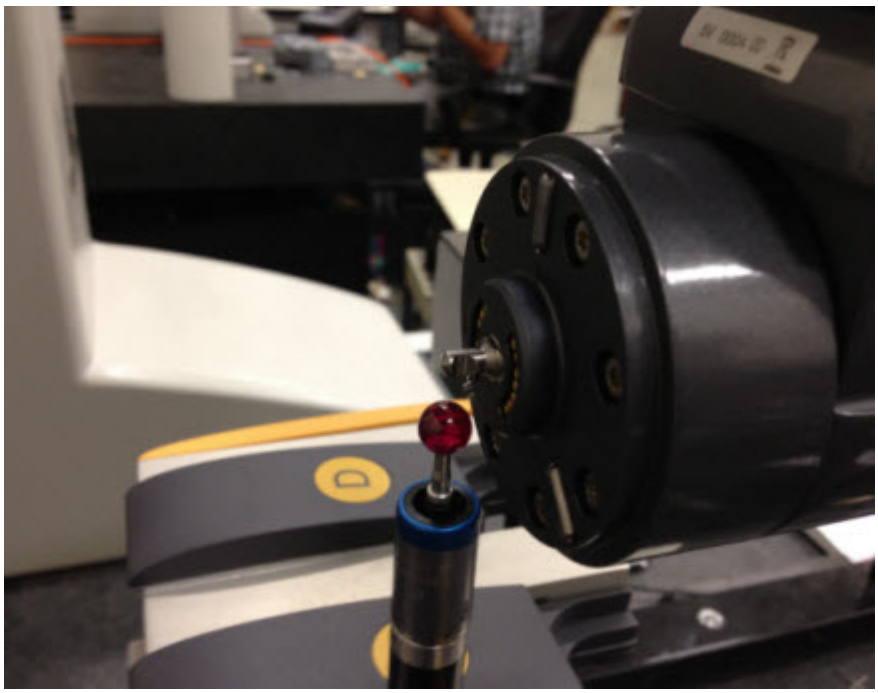
PC-DMIS Message

You can now close the probe changer lids and release the HD kinematic joint adapter.

Take 1 hit on the side of the kinematic joint close to the joint pin with the probe changer datum sphere. Be careful to avoid the small pins and holes and take the hit on a flat area.

After you complete this point the HD kinematic joint pin will be measured in DCC.

PC-DMIS prompt to take a hit on the datum sphere with HD AutoJoint.



Use your machine's jog box to take a manual hit on the side of the datum sphere using the empty AutoJoint connection

2. After this manual hit, lift the wrist clear of the datum sphere. The system proceeds under DCC to measure the sphere with the pin that is protruding from the bottom of the AutoJoint.

   When this finishes, calibration is complete.

PC-DMIS Message
All measurement is now complete. Please reconnect the probe used to measure the ports and then click OK.

In the next step, you measure any defined extensions.

Step 10 - Measure Extensions

After you measure the AutoJoint, you are asked to measure any defined extensions.

PC-DMIS Message
Please attach only the extension to be used in port 2. When you click OK if you have a wrist it may rotate (if needed) to allow for any rotation of the bottom joint.

When prompted then take 1 hit on the bottom of the kinematic joint with the probe changer datum sphere.

Be careful to avoid the small pins and holes and take the hit on a flat area.

After you complete this point the kinematic joint pin will be measured in DCC.

Prompt for measuring any defined extensions

1. Attach the extension and take a hit on the bottom of the extension as directed.
2. After the manual hit, lift the wrist clear of the datum sphere.

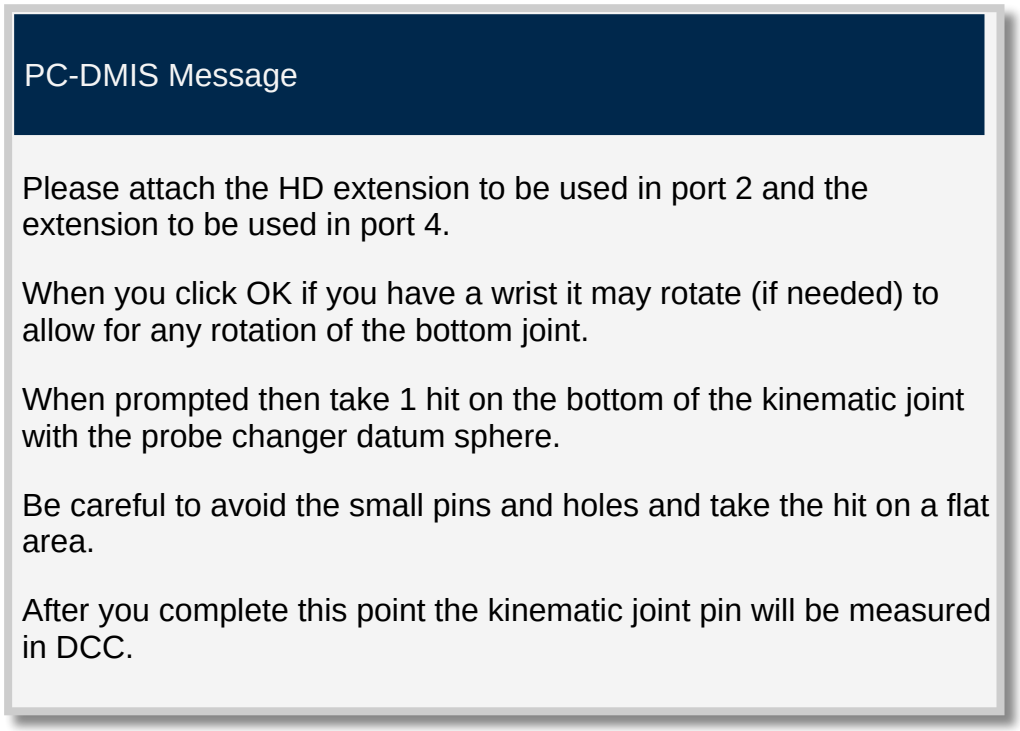
> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

The system then goes into DCC Mode and proceeds to measure the sphere with the pin that is protruding from the bottom of the extension.

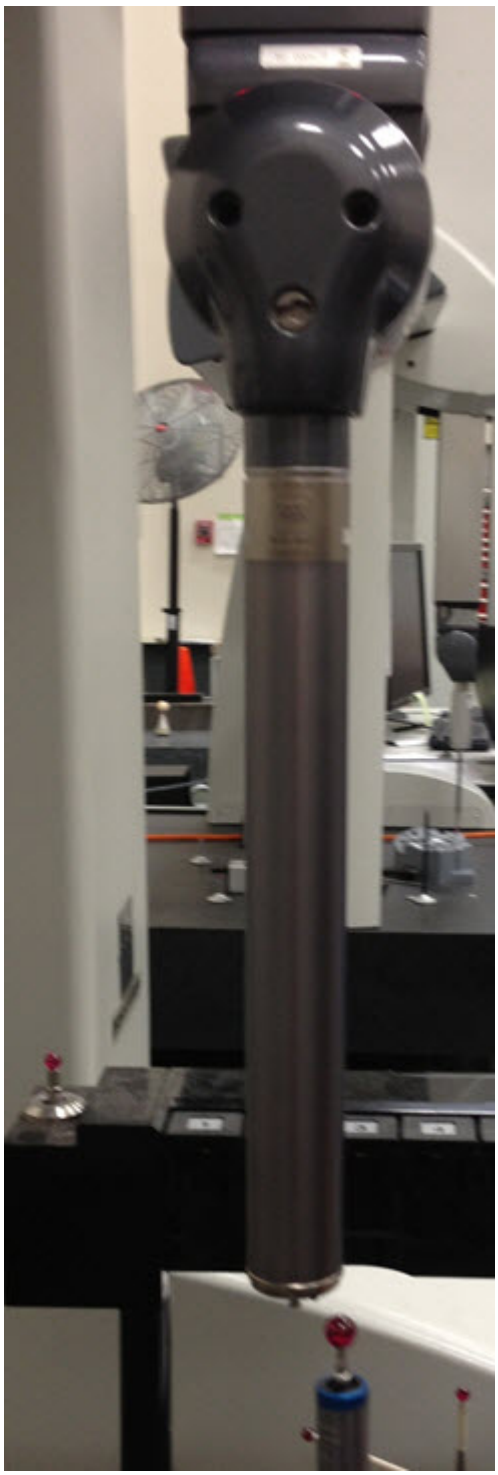
Use your machine's jog box to take a manual hit on the top of the datum sphere using the empty extension

3. After the HD extension has been measured, you are asked to measure each TKJ extension by individually attaching them to the HD extension and measuring the joint at the bottom.

   After you measure the TKJ extensions, if you have additional HD extensions, you will be asked to repeat the sequence of attaching and measuring just the HD extension followed by attaching and measuring each TKJ extension.



Prompt for measuring any remaining extensions

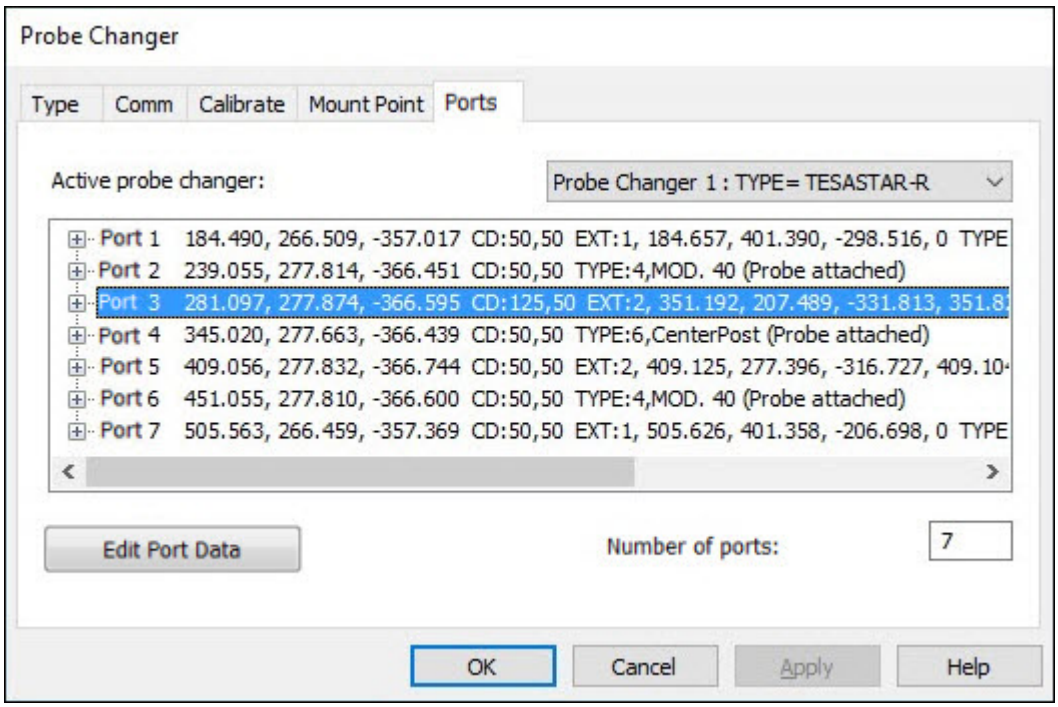


Use your machine's jog box to take a manual hit on the top of the datum sphere using the empty extension

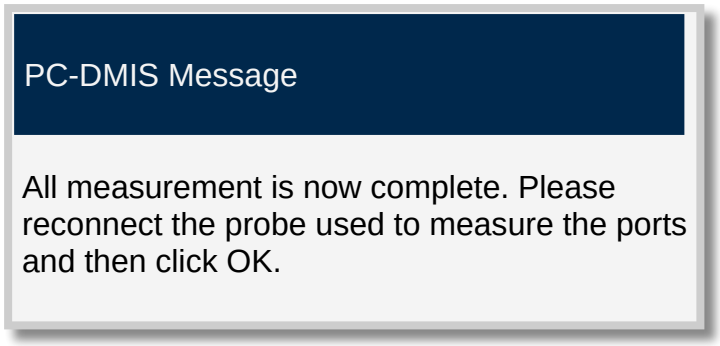
In the next step, you review the calibration results.

Step 11 - Review the Calibration Results

1. When the calibration of the TESASTAR-R / HR-R Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). The calibration information now exists for the location of each calibrated port. For example:



Ports tab with calibration results



Prompt for reconnecting the probe

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

    - The probe changer must be aligned parallel to either the X or the Y axis of the CMM.
    - The Z values for all regular ports should be nearly the same and the Z for all HD ports should be nearly the same. However, the Z for regular ports will not be the same as the Z for HD ports.

    When you use an HD wrist, you should remove the datum sphere from the rack after calibration is complete. This prevents collisions when using the two adjacent ports for tool changes.

    How a `LOADPROBE` command works with the TESASTAR-R / HR-R Probe Changer:

1. During measurement routine execution, the probe entities that you added to each port are automatically picked up from the port whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. Before picking it up, the probe body moves to the mount point and then into the empty port to drop off the current probe.
3. The rack's keys rotate to the release position. The current probe remains in the port while the probe body lifts away to detach.
4. The probe body moves over the load position above the port that contains the probe.
5. The probe body moves down onto the new probe. The keys again rotate to automatically engage the new module.
6. The probe body moves back out of the port and over to the rack’s mount point.
7. The CMM continues to measure the part with the newly loaded probe.

The probe change cycle varies slightly from the steps described above for HD ports and extensions.

Calibrating the TP20 Probe Changer

The calibration process for the TP20 Probe Changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process in this documentation is specific to the TP20 Probe Changer.

Inserts or extensions are not used for any of the port positions.

TP20 Probe Changer Rack

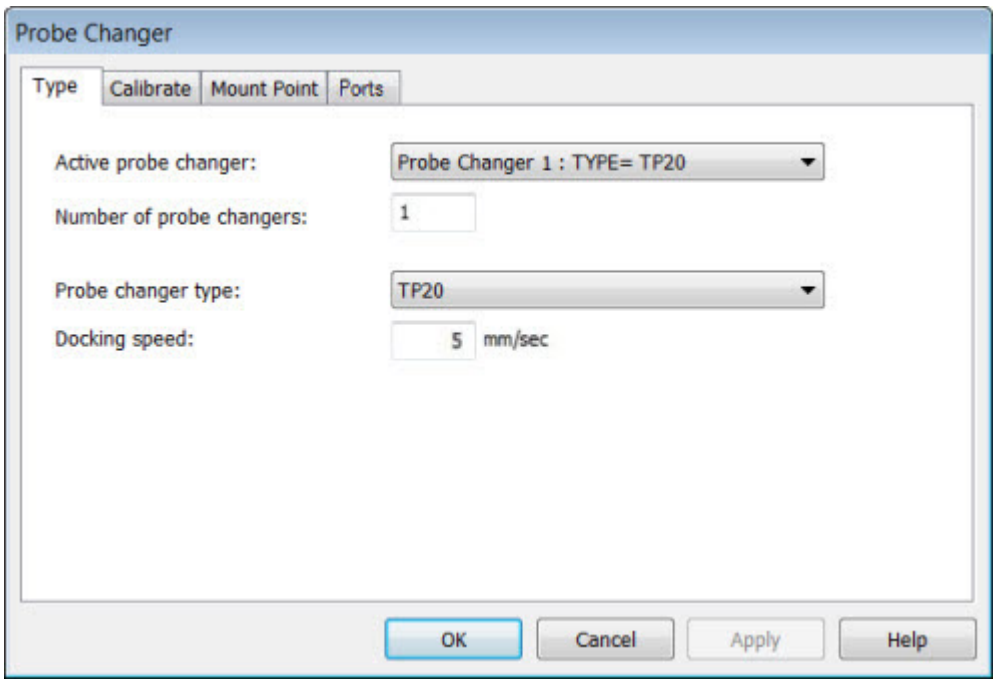
> **!** This probe changer DOES NOT need to be mounted on the machine table parallel to a machine axis, such as the X or Y axis. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the TP20 Probe Changer

To select the TP20 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **TP20**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** box, select the list item that represents the probe changer you want to define. If this is the first probe changer, this shows **Probe Changer 1: TYPE=None**.

5. In the **Number of probe changers** box, specify the number of different types of probe changers you want to define.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

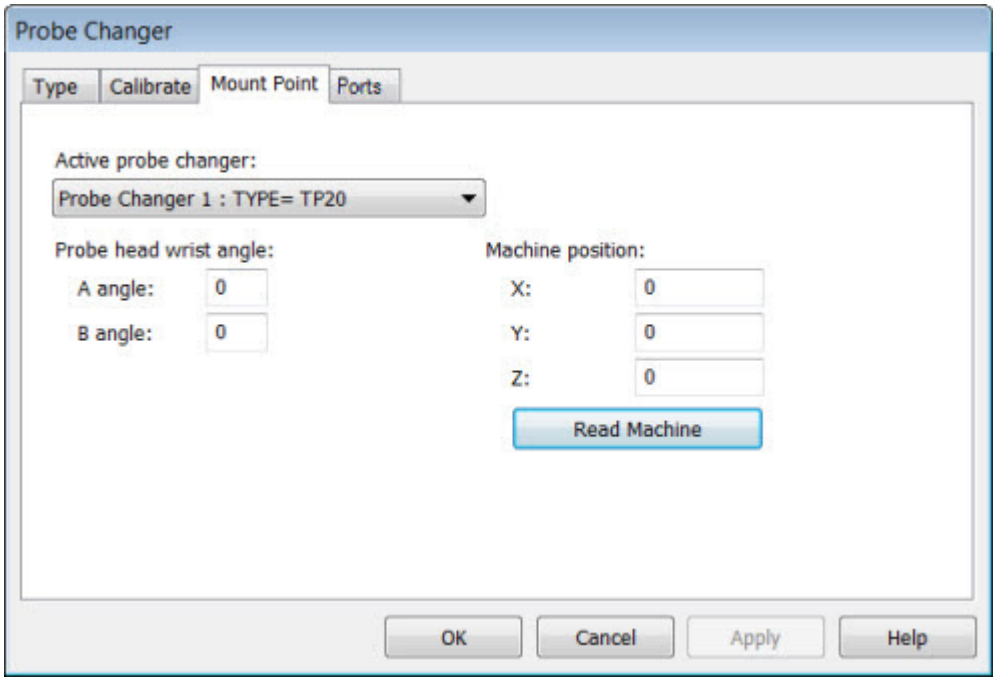
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point and Wrist Angle

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**).



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active Probe Changer** list, select **TYPE=TP20**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use a probe rotation to ensure the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine position** values with the current position. You can also manually type these values.
6. To save your changes, click **Apply**.

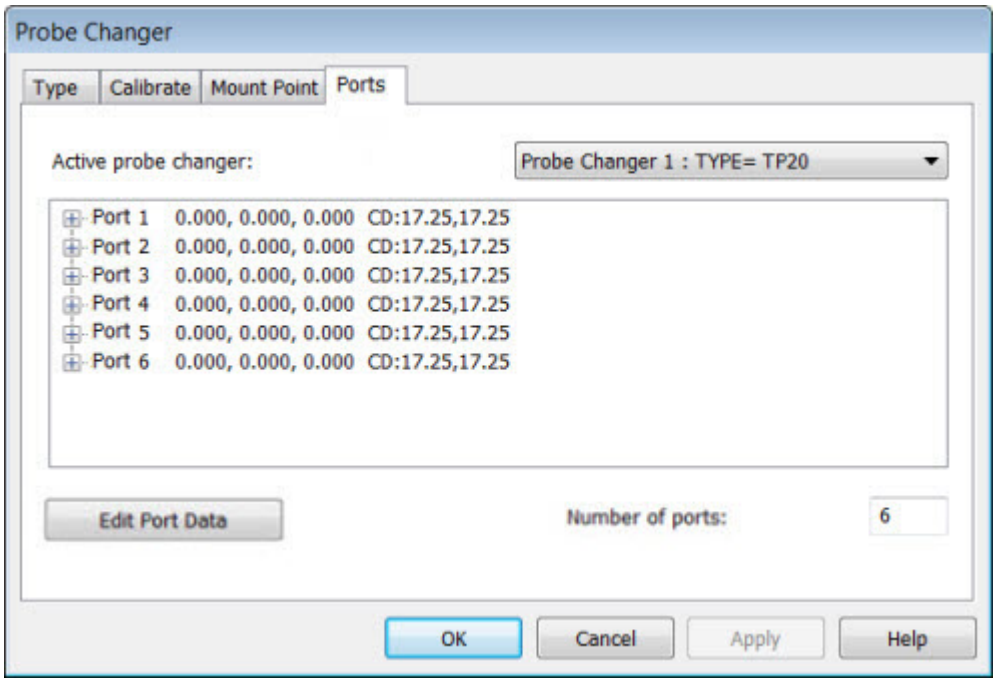
In the next step, you define the ports for your system.

Step 3 - Define the Ports

The final step prior to the actual calibration of the TP20 Probe Changer is to define the ports on the rack that you are calibrating.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

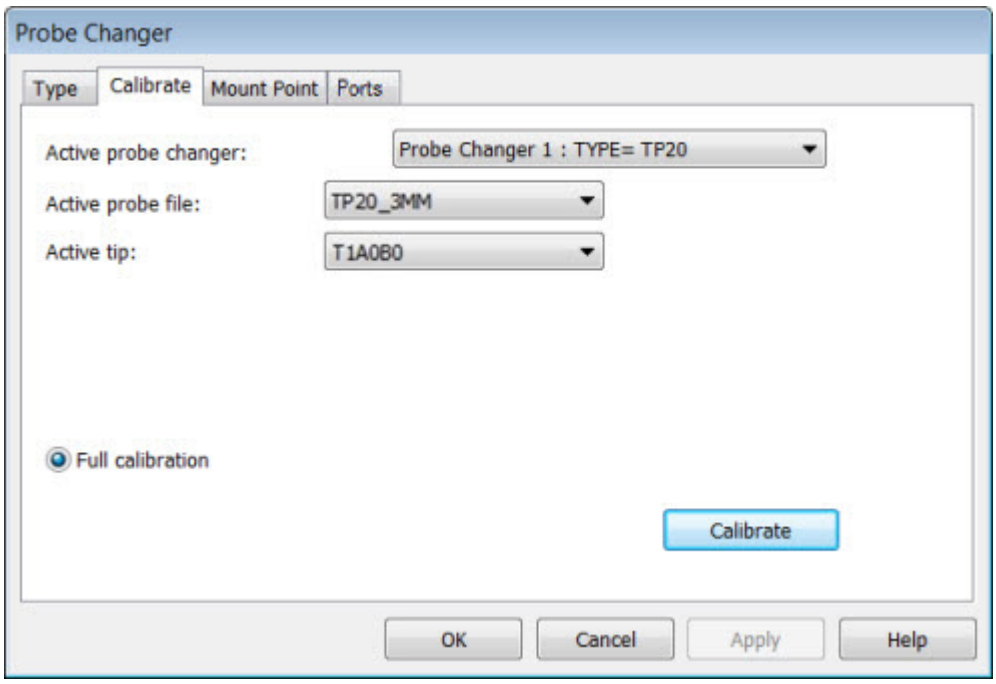
2. In the **Active probe changer** list, select **TYPE=TP20**.
3. In the **Number of ports** box, type the number of ports that you want to define.
4. Click **Apply** to store this parameter.

The next step starts the calibration process.

Step 4 - Prepare for Calibration

To begin the calibration process for the TP20 Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):

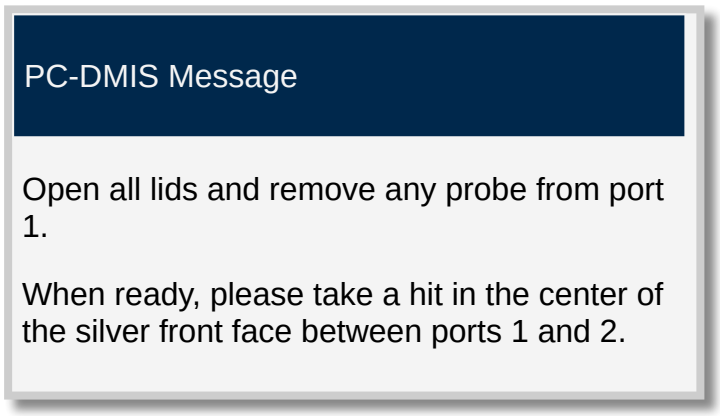


Probe Changer Dialog Box - Calibrate Tab

2. Click the **Calibrate** button.

3. The **Active probe file** and **Active tip** items default to the current measurement routine settings. If needed, change these to the probe and tip that you intend to use for the probe changer calibration.

4. Open the lids on each port by sliding them back until they click into place.



PC-DMIS prompt to open the port lids prior to calibration.

Open port lids prior to calibration.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

When you start the calibration procedure for the TP20 Probe Changer, the ports must be emptied of all probes.

You will take the first hit on the front face to the left of the first port. This message prompts you to open the lids and take the first hit:

PC-DMIS Message

Open all lids and remove any probe from port 1.

When ready, please take a hit in the center of the silver front face between ports 1 and 2.

PC-DMIS prompt to take the first hit to the left of port 1.

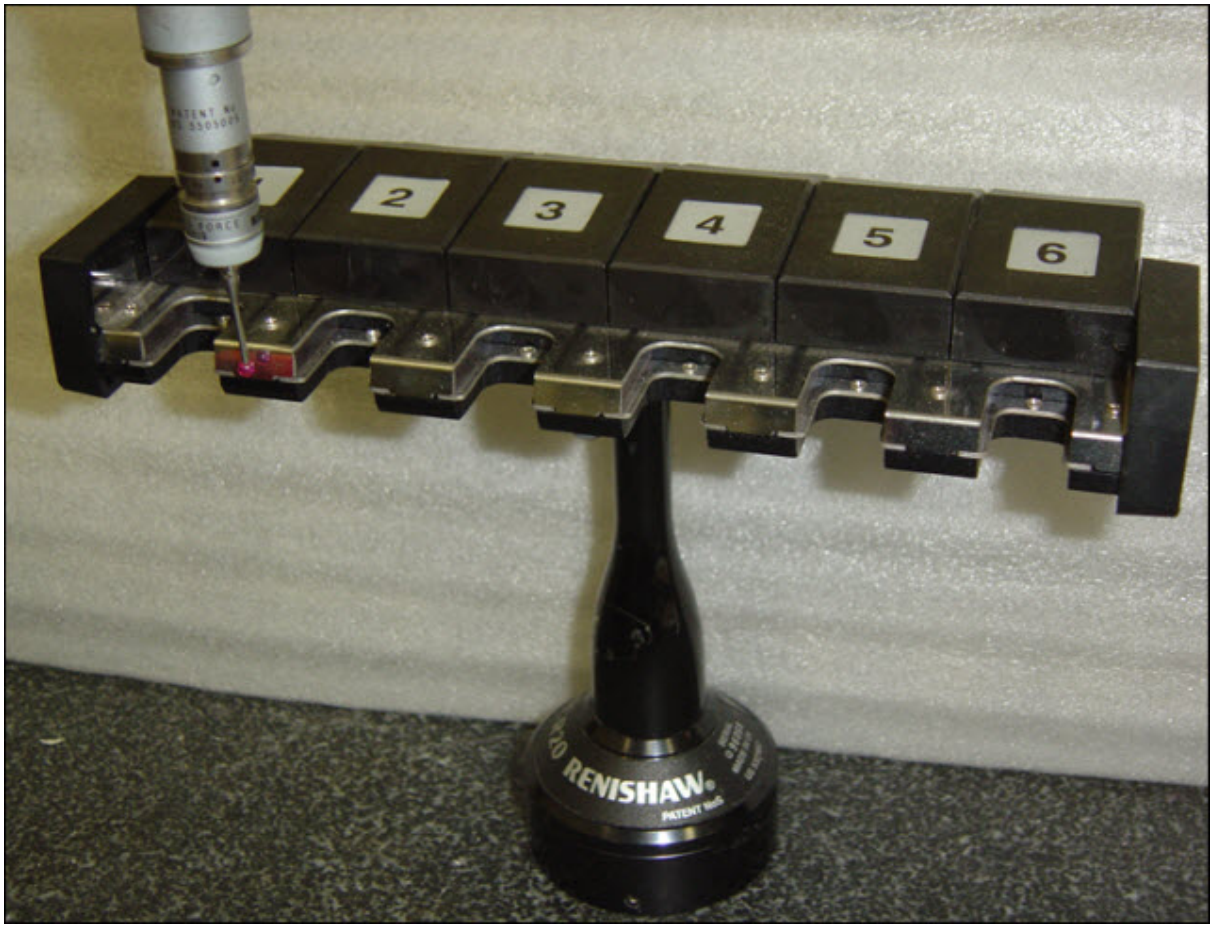
1. When you are ready to take the first measurement, click **OK**.
2. Remove all of the modules and styli by sliding them forward and out of the ports.

   You are prompted through the process of measuring three manual hits through a series of messages. Three hits are required to fully define the orientation of this rack in the machine's XY plane. It is not necessary to align the rack to any single axis, as

the manual hits identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

You will take the first manual hit in the XY plane, on the vertical face in the back of port 1.

3. Use your machine's jog box to measure the first hit on the front face to the left of port #1:

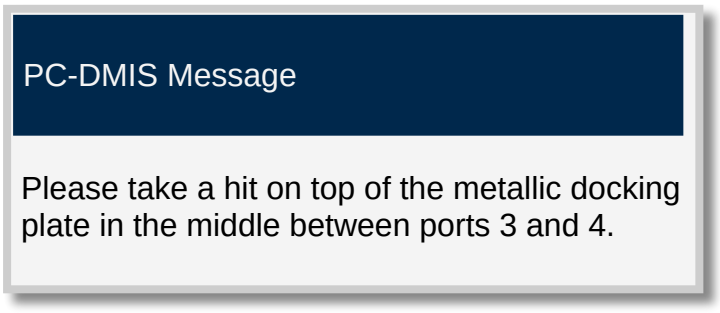


Example showing the location of the first manual hit during TP20 calibration.

In the next step, you take the second manual hit.

Step 6 - Take the Second Manual Hit

For the TP20 Probe Changer, you will take the second manual hit on the top metal surface between ports 3 and 4.



PC-DMIS prompt to take the second hit on the top, in the middle of the rack.

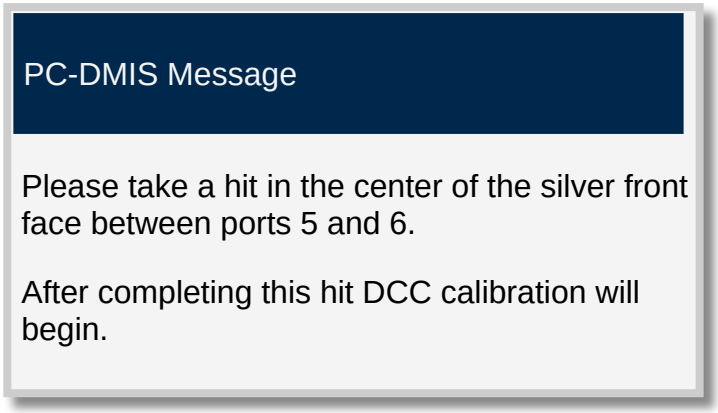
Use your machine's jog box to measure the second hit on the top surface between ports 3 and 4:

Example showing the location of the second manual hit during TP20 calibration.

In the next step, you take the third manual hit.

Step 7 - Take the Third Manual Hit

For the TP20 Probe Changer, you will take the third hit on the front surface and to the left of the last port.



PC-DMIS prompt to take the third hit on the top, to the left side of the last port.

Use your machine's jog box to measure the third hit on the top surface to the left of the last port:

Example showing the location of the third manual hit during TP20 calibration.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

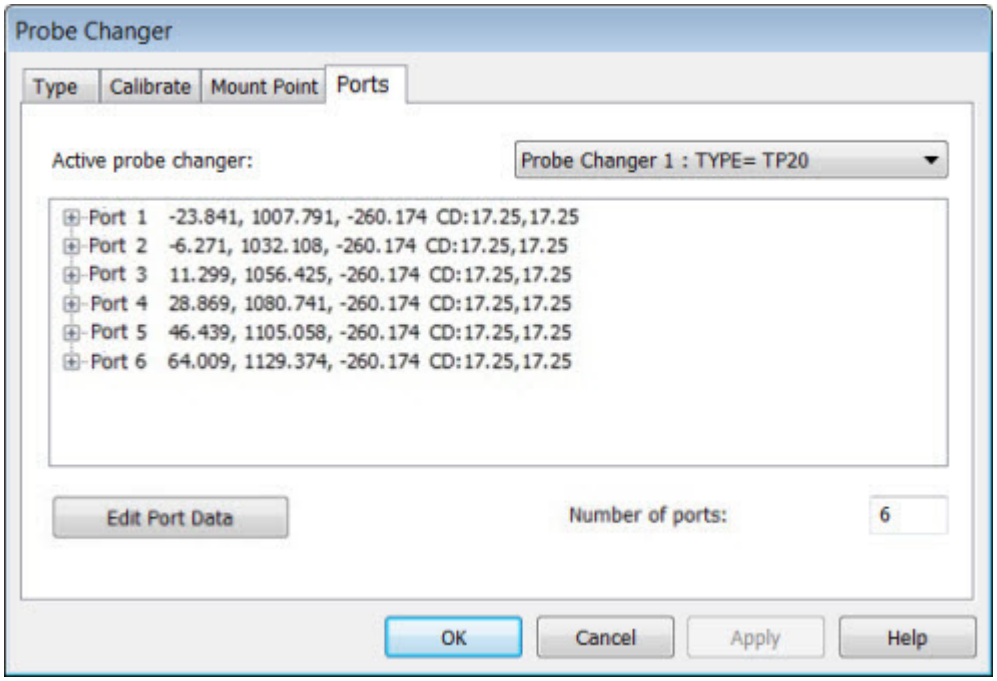
After it takes this hit on the back of the last port, PC-DMIS drives in DCC Mode and takes some additional measurements. It works backwards from the last port toward the first port. It actually measures the same three locations in reverse order in DCC Mode. Two additional hits are then taken on either side of port 1.

There is no specific message to indicate that the calibration is complete. The PC-DMIS **Execution** dialog box closes, and the **Calibrate** tab reappears.

The next step is to review the calibration results.

Step 8 - Review the Calibration Results

1. When the calibration of the TP20 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). The calibration information now exists for the location of each calibrated port. For example:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, look for the following. Any significant deviations from these expectations may be due to a bad hit.

    - This probe changer is not necessarily aligned parallel to any axis of the CMM. However, the X and Y values should show equal spacing between ports (roughly 30 mm apart).
    - The Z values should be nearly identical since the ports are all at the same height.

The results during measurement routine execution are:

1. Probe entities that are added to each port are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.
2. The probe body moves to the mount point and then into the "unload" port to drop off the current probe.
3. The current probe remains in the port while the probe body lifts away to detach.
4. From there, the probe moves over to the next "load" position and then down onto the new probe, and automatically engages the new module.
5. It then moves back out of the port and over to the probe changer’s mount point.

6. From there, it continues to measure.

Calibrating the TP200 Probe Changer

The calibration process for the TP200 Probe Changer represents the steps that you would take to prepare to use any of the probe changers that PC-DMIS supports. The process discussed here is specific to the TP200 Probe Changer.

Inserts or extensions are not used for any of the port positions.

TP200 Probe Changer Rack

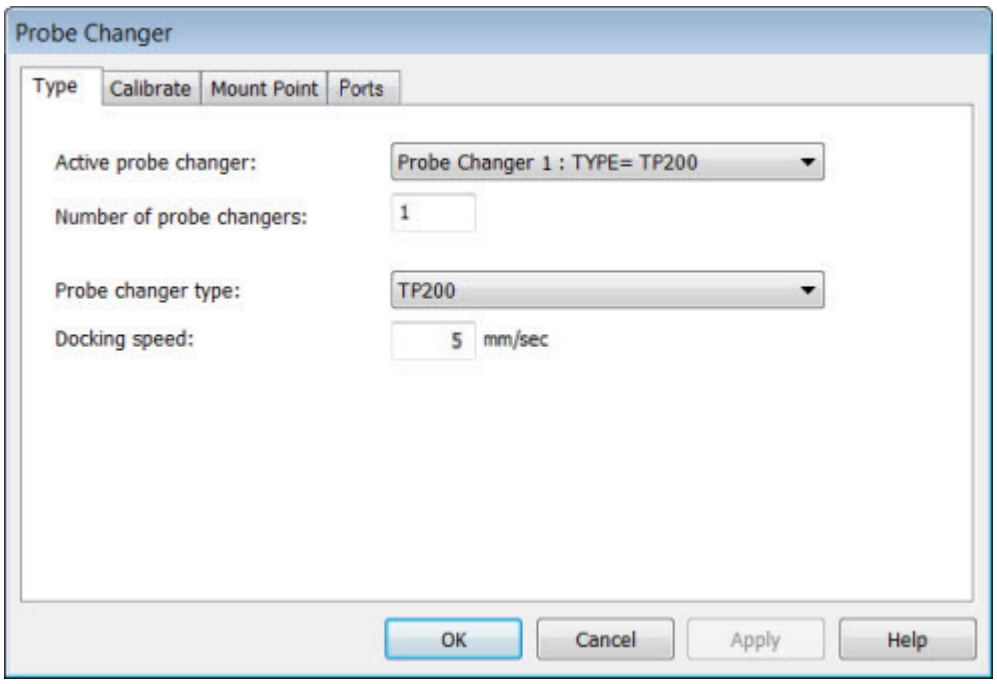
> This probe changer DOES NOT need to be mounted on the machine table parallel to a machine axis, such as the X or Y axis. For mounting instructions, consult the documentation that came with your probe changer.

To calibrate your probe changer:

Step 1 - Select the TP200 Probe Changer

To select the TP200 Probe Changer:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **TP200**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.

4. In the **Active probe changer** box, select the list item that represents the probe changer that you want to define. If this is the first probe changer, this shows **Probe Changer 1: TYPE=None**.

5. In the **Number of probe changers** box, specify the number of different types of probe changers that you want to define.

6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

   To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

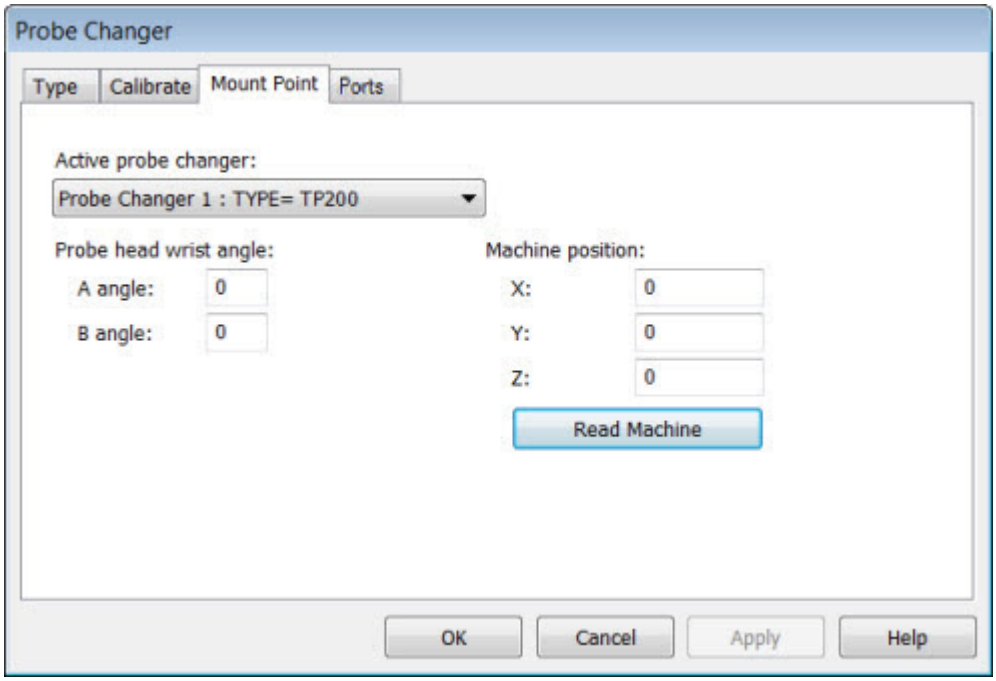
In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

Step 2 - Define the Mount Point and Wrist Angle

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. From the **Active Probe Changer** list, select **TYPE=TP200**.
3. If necessary, alter the probe head wrist angle for both **A angle** and **B angle**. Often, but not always, these values are both 0 (zero). You need to use a probe rotation to ensure the probe has the ability to move in and out of the probe changer during the necessary steps of the probe rack calibration procedure.
4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z Machine position** values with the current position. You can also manually type these values.
6. To save your changes, click **Apply**.

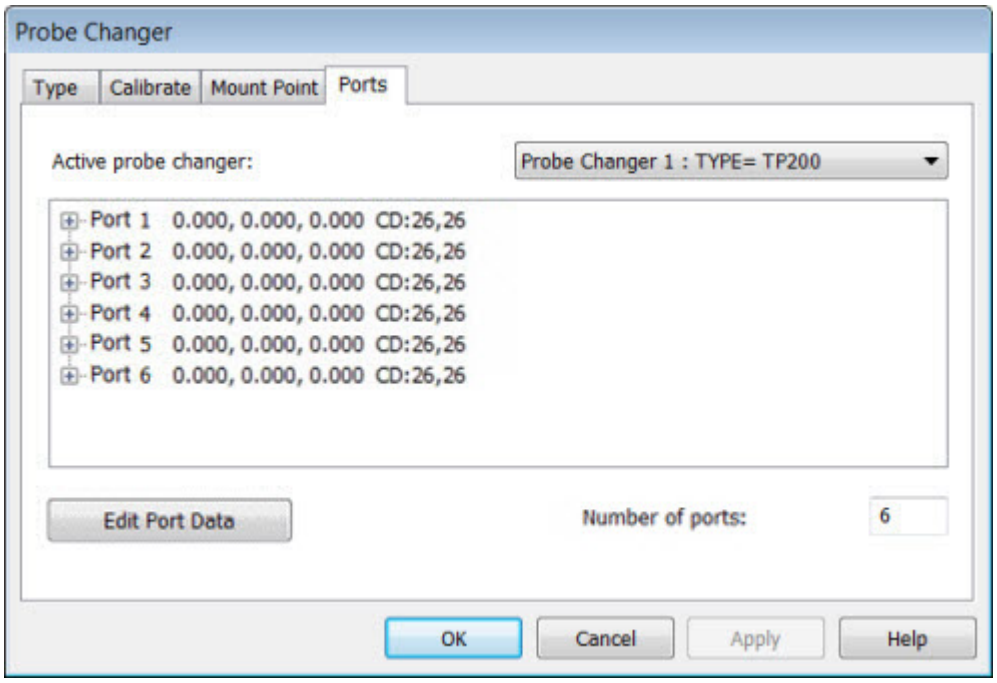
In the next step, you define the ports for your system..

Step 3 - Define the Ports

The final step prior to the actual calibration of the TP200 Probe Changer is to define the ports on the rack that you are calibrating.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

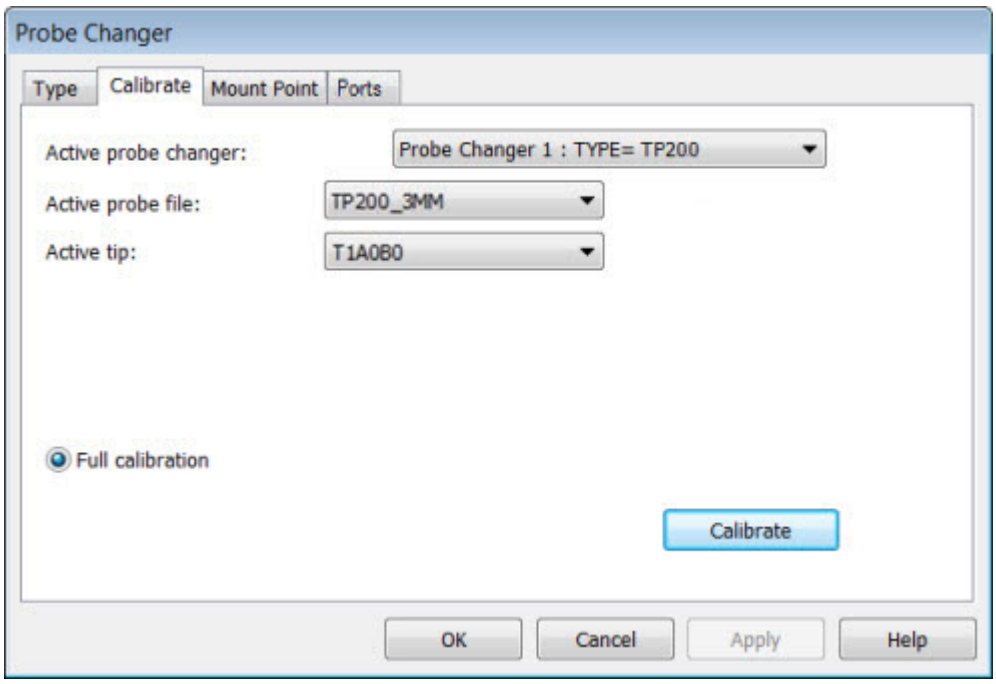
2. In the **Active probe changer** list, select **TYPE=TP200**.
3. In the **Number of ports** box, type the number of ports that you want to define.
4. Click **Apply** to store this parameter.

The next step starts the calibration process.

Step 4 - Prepare for Calibration

To start the calibration process for the TP200 Probe Changer:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

2. Click the **Calibrate** button.
3. The **Active probe file** and **Active tip** items default to the current measurement routine settings. If necessary, change these to the probe and tip that you intend to use for the probe changer calibration.
4. Open the lids on ports 3 and 4 on each side of the divider (center post) by sliding them back and slightly to the side until they click into place.

Example showing the open port lids before calibration.

In the next step, you take the first manual hit.

Step 5 - Take the First Manual Hit

When you start the calibration procedure for the TP200 Probe Changer, the ports must be emptied of all probes.

You will take the first hit on the front (vertical) face of the divider, to the right of port 3. This message box prompts you to open the lids and take the first hit:

PC-DMIS Message

Unplug the cable from the rack, open lids 3 and 4 and remove any probe from ports 3 and 4.

When ready, please take a hit on the front face of the divider between ports 3 and 4.

The hit should be taken on the lower widest portion as close as possible to the outer edge on the left-hand side next to port 3.

PC-DMIS prompt for the first hit.

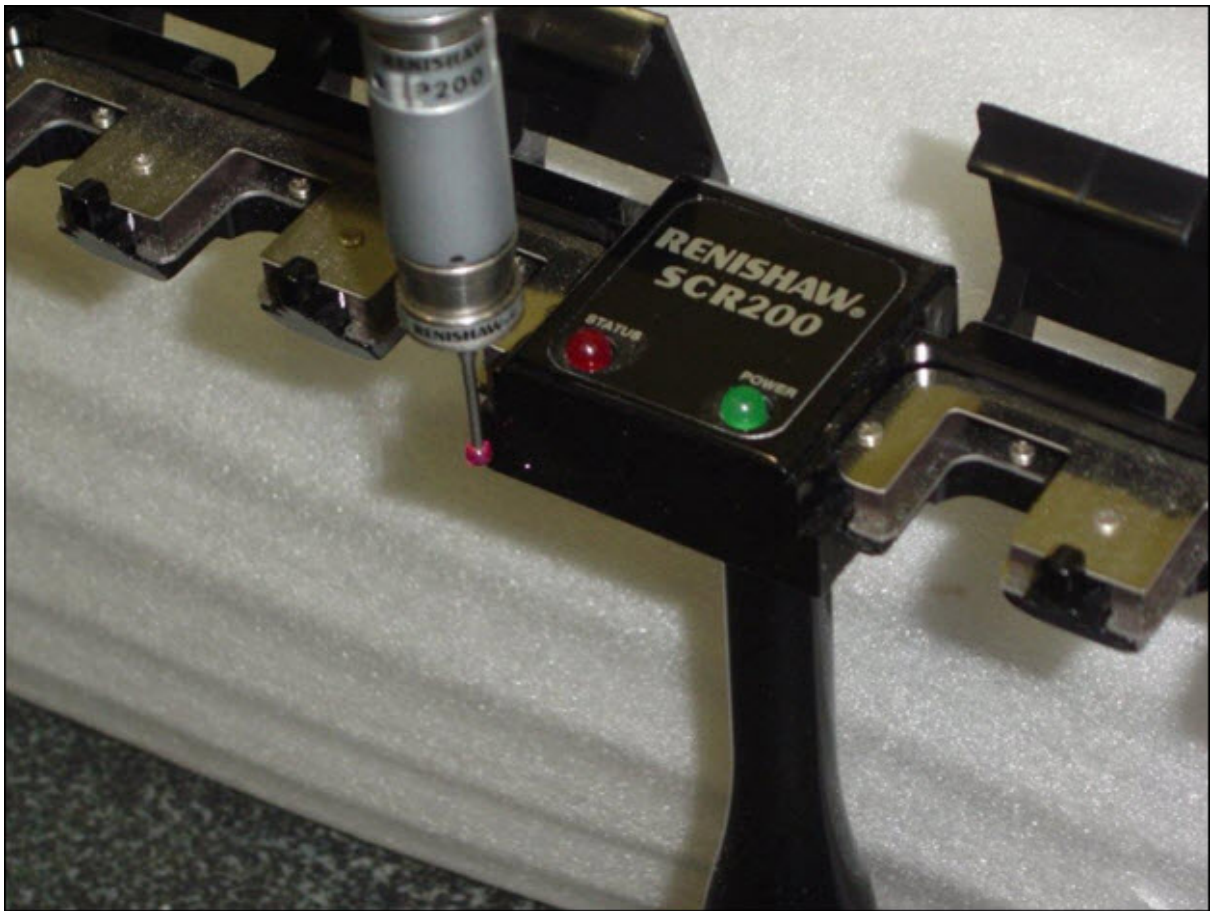
1. When you are ready to take the first measurement, click **OK**.

2. Remove all of the modules and styli by sliding them forward and out of the ports.

   You are prompted through the process of measuring three manual hits through a series of messages. Three hits are required to fully define the orientation of this rack in the machine’s XY plane. It is not necessary to align the rack to any single axis, as the manual hits will identify any rotation involved. Follow the prompts and take the needed hits as directed in the instructions for each hit.

   You will take the first manual hit in the XY plane on the front (vertical) face of the divider, to the right of port 3.

3. Use your machine's jog box to measure the first hit on the front (vertical) face of the divider, to the right of port 3:



Example showing the location of the first manual hit during TP200 calibration.

In the next step, you take the second manual hit.

Step 6 - Take the Second Manual Hit

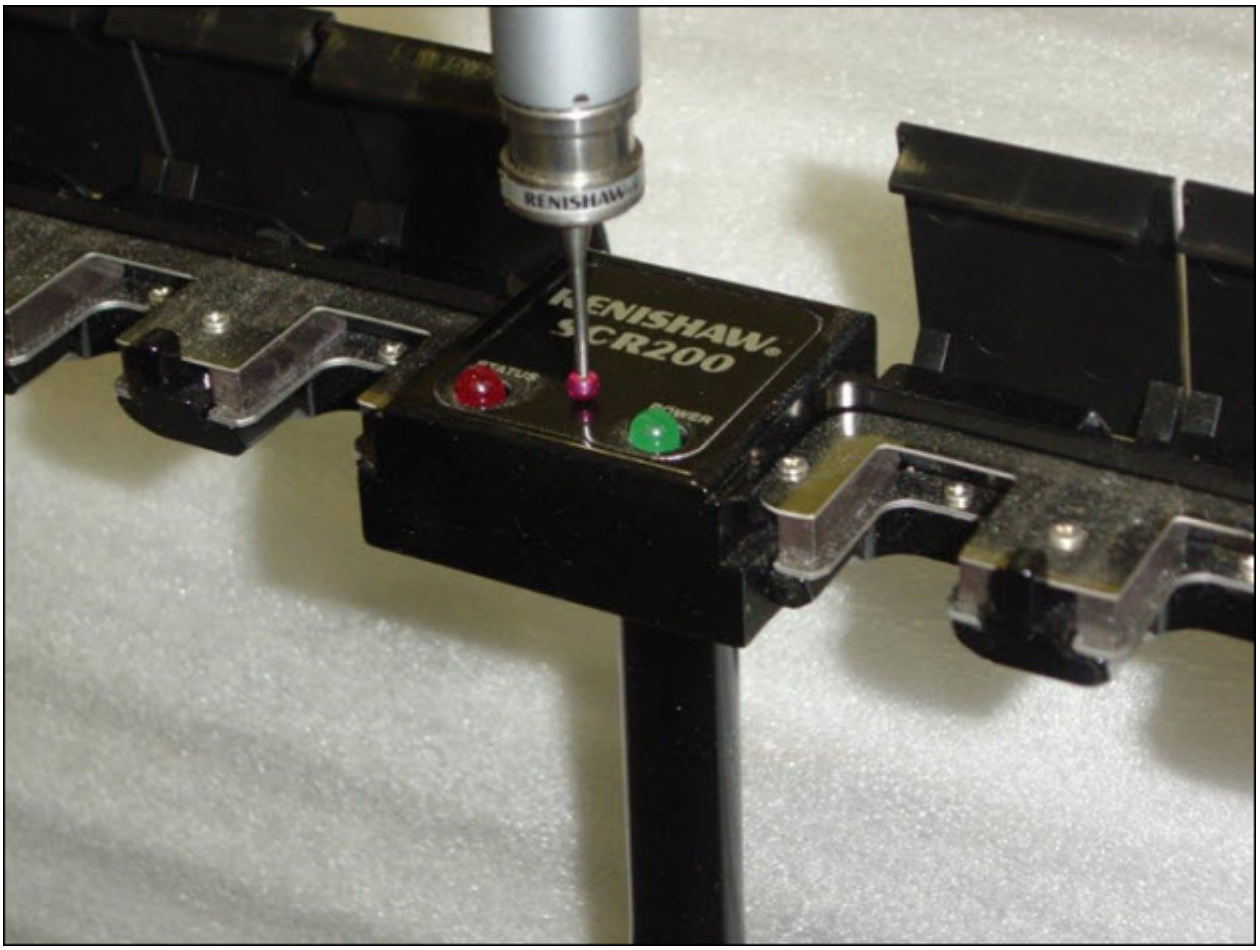
For the TP200 Probe Changer, you will take the second manual hit on the top (horizontal) face of the divider and between ports 3 and 4. The message box is:

PC-DMIS Message

Please take a hit on the top in the middle of the rack.

PC-DMIS prompt for the second hit.

Use your machine's jog box to measure the second hit on the top (horizontal) surface of the divider and between ports 3 and 4:



Example showing the location of second manual hit during TP200 calibration.

In the next step, you take the third manual hit.

Step 7 - Take the Third Manual Hit

For the TP200 Probe Changer, you will take the third hit on the front (vertical) face of the divider and to the left of port 4.

PC-DMIS Message

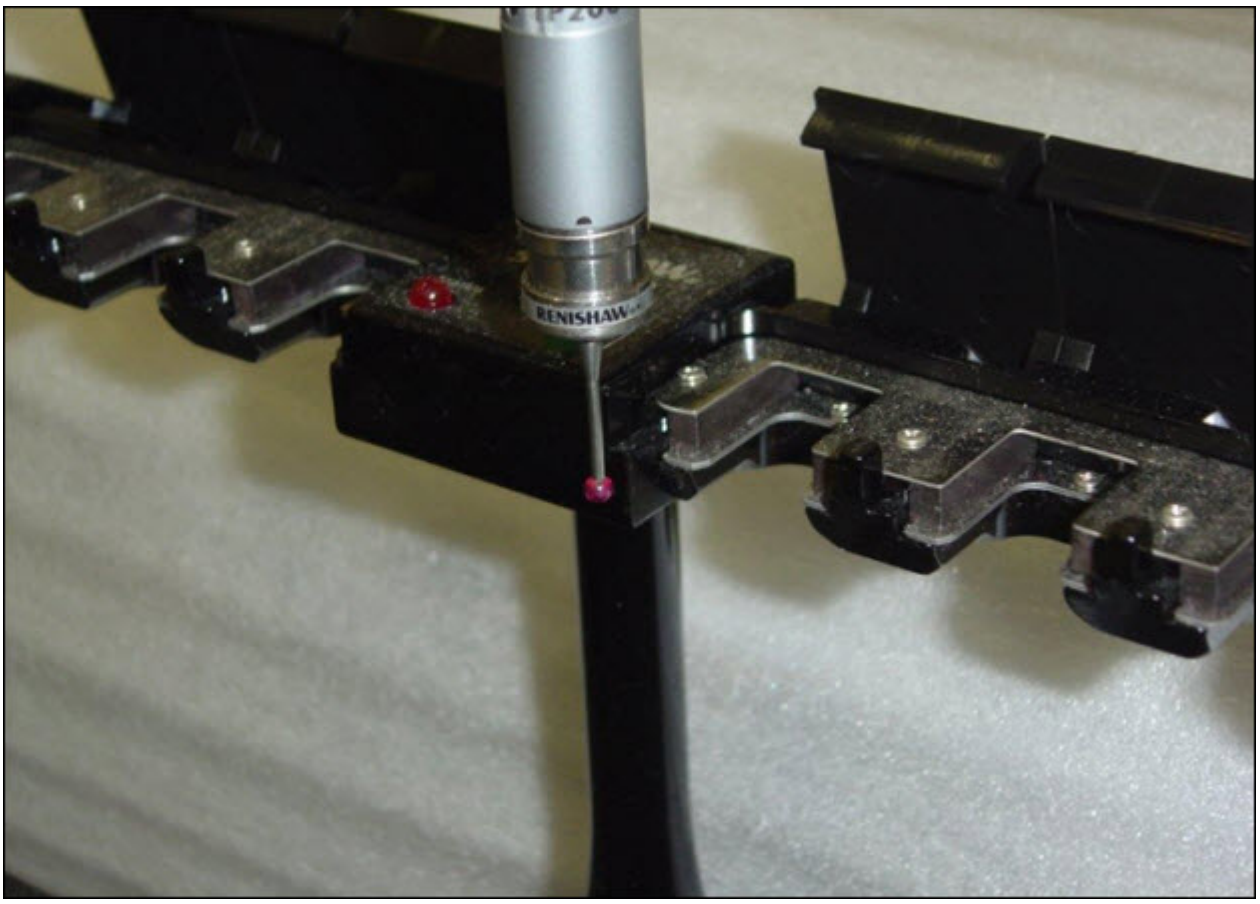
Please take a hit on the front face of the divider between ports 3 and 4.

The hit should be taken on the lower widest portion as close as possible to the outer edge on the right-hand side next to port 4.

After completing this hit DCC calibration will begin.

PC-DMIS prompt for the third hit.

Use your machine's jog box to measure the third hit on the front (vertical) surface of the divider, to the left of port 4:



Example showing the location of the third manual hit during TP200 calibration.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

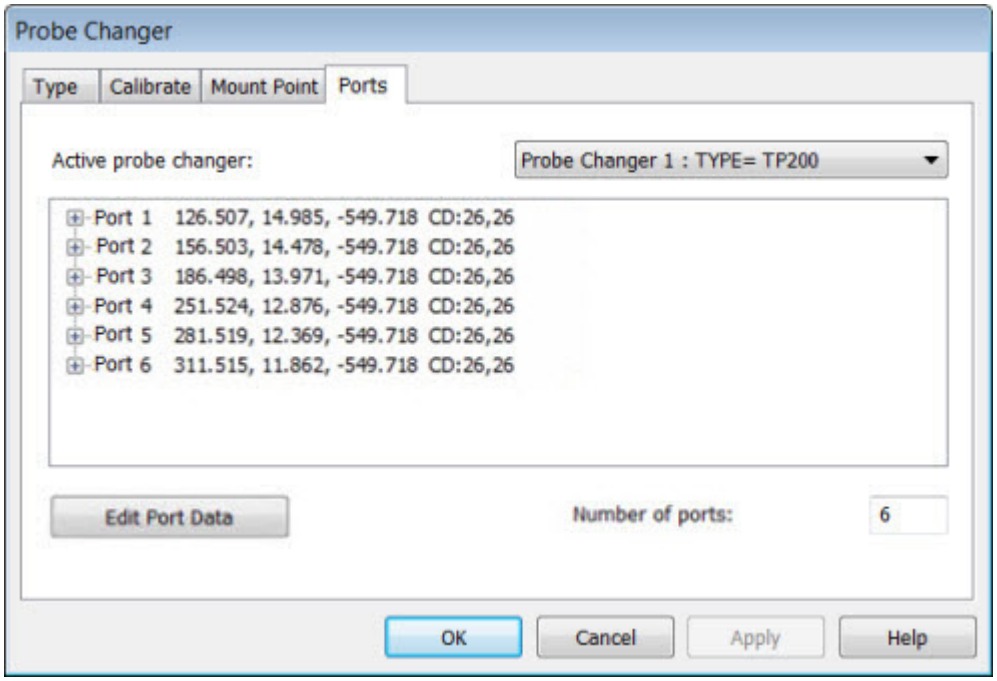
After it takes this hit on the back of the last port, PC-DMIS drives in DCC Mode and takes some additional measurements. It works backwards from the last port toward the first port. It actually measures the same three locations in reverse order in DCC Mode. Two additional hits are taken on either side of port 1.

There is no specific message to indicate that the calibration is complete. The PC-DMIS **Execution** dialog box closes, and the **Calibrate** tab reappears.

The next step is to review the calibration results.

Step 8 - Review the Calibration Results

1. When the calibration of the TP200 Probe Changer is complete, select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). The calibration information now exists for the location of each calibrated port:



Probe Changer dialog box - Ports tab with calibration results

2. When you view the results, watch for the following discontinuities. Any significant deviations from these expectations may be due to a bad hit.

   - This probe changer is not necessarily aligned parallel to any axis of the CMM. However, the X and Y values should show equal spacing between ports (roughly 30 mm apart), with about 65 mm between ports 3 and 4 for the divider.
   - The Z values should be nearly identical since the ports are all at the same height.

The results during measurement routine execution are:

1. Probe entities that are added to each port are automatically picked up from that location whenever PC-DMIS executes a `LOADPROBE` command for that probe.

2. The probe body moves to the mount point and then into the "unload" port to drop off the current probe.
3. The current probe remains in the port while the probe body lifts away to detach.
4. From there, the probe moves over the next "load" position and then down onto the new probe, and automatically engages the new module.
5. It then moves back out of the port and over to the probe changer's mount point.
6. From there, it continues to measure.

Calibrating the Travel Rack HR-X1-TR

The calibration process discussed in this section is specific to the Travel Rack HR-X1-TR.

The probe changer mounts onto the drive leg of the CMM. For mounting instructions, consult the documentation that came with your Travel Rack HR-X1-TR.

Mounted Travel Rack HR-X1-TR

The probe changer travels with the moving portal of the machine. The stylus change is done horizontally. Therefore, the TESASTAR-M brings the sensor into the right position.

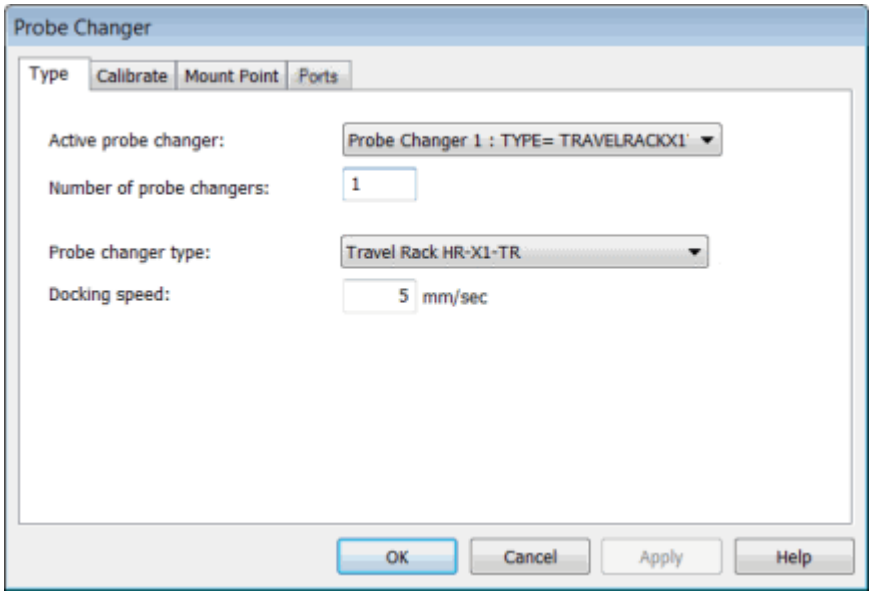
Since the change rack moves with the portal, there are no additional travelling distances to perform a change of configuration, as would be the case with a standard rack. This enables quick changes of the styli configuration anywhere in the measurement volume.

To calibrate your probe changer:

Step 1 - Select the Travel Rack HR-X1-TR

To select the Travel Rack HR-X1-TR:

1. In the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**), select the **Type** tab.
2. In the **Probe changer type** list, select **Travel Rack HR-X1-TR**:



Probe Changer Dialog Box - Type Tab

3. To make this probe changer active and load the settings that are relevant to this probe changer, click **Apply**. Other tabs become visible after you click **Apply**.
4. In the **Active probe changer** box, select the list item that represents the probe changer that you want to define. If this is the first probe changer, this shows **Probe Changer 1: TYPE=None**.
5. In the **Number of probe changers** box, specify the number of probe changers you want to define.
6. In the **Docking speed** box, type a value. A value between 10-20 mm/sec is appropriate for this machine configuration. You can adjust this value at any time to speed up or slow down the system.

> ⚠ To avoid equipment damage due to excessive docking speed, gradually increase the **Docking speed** value.

7. Click **Apply** to apply any changes made for this probe changer, and then click **OK** to close the dialog box.

For details on controller configuration settings, see the documentation that came with your controller.

In the next step, you define the location your probe body moves to when using the probe changer to switch probe components.

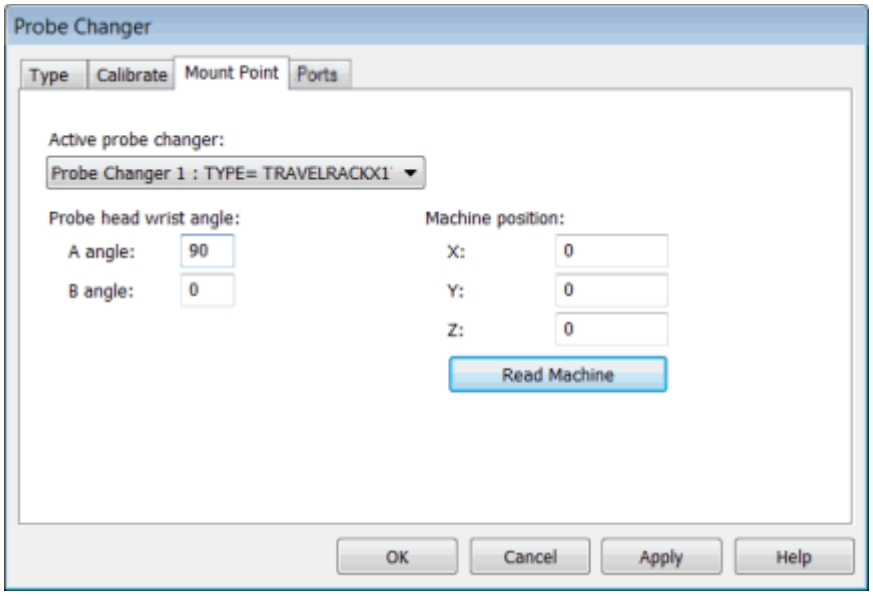
Step 2 - Define the Mount Point

The probe changer mount point is the location in front of the probe changer where the machine moves to before it picks up or drops off a probe. You should pick a location that avoids collisions with the probe changer, the part, clamps, or any other object in the workspace.

To define the mount point for your probe changer, follow these steps:

The Travel Rack HR-X1-TR is capable of changing the probe at any location of one machine axis (the default is the machine X axis for the Leitz CMM). Therefore, the mount point is effective for only the other two axes.

1. Select the **Mount Point** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Mount Point Tab

2. In the **Active probe changer** list, select **TYPE=TRAVELRACKX1V**.
3. Enter the probe head wrist angle for both **A angle** and **B angle**. The A and B angle combination should be one for which the probe is pointing either parallel to or away from the rack. It is automatically rotated into the rack during the change. We recommend that you set this to A=90 and B=0.

> For this probe changer, never set these probe angles to A=90 and B=90. If you do, PC-DMIS shows an error message and cannot change probes.

4. Use your jog box to manually move your machine to the desired mount point position.
5. Click the **Read Machine** button to populate the **X**, **Y**, and **Z** machine position values with the current position. You can also type these values.
6. To save your changes, click **Apply**.

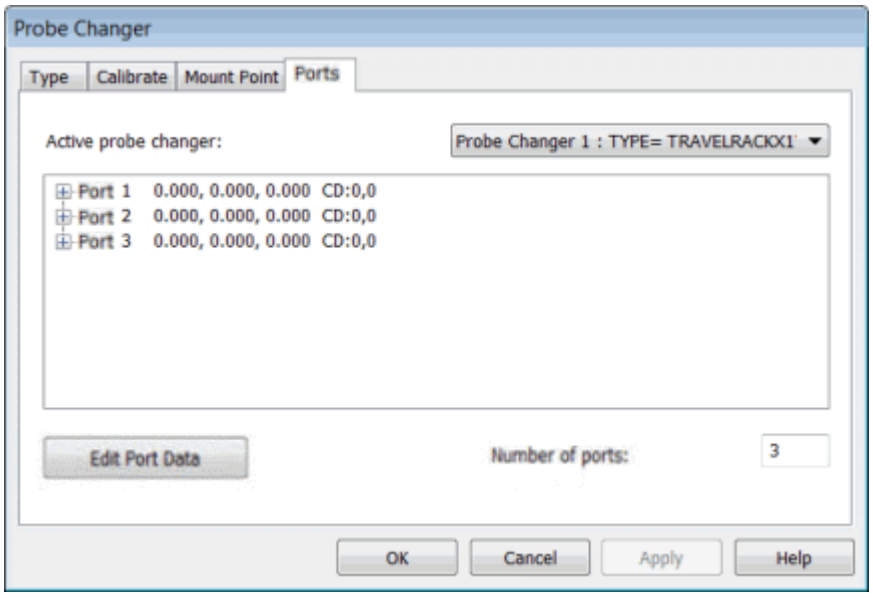
In the next step, you define the ports.

Step 3 - Define the Ports

The final step prior to calibrating the Travel Rack HR-X1-TR is to define the ports on the rack.

To define the ports, do the following:

1. Select the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Ports Tab

2. In the **Active probe changer** list, select **TYPE=TRAVELRACKX1V**.
3. In the **Number of ports** box, type the number of ports on your probe changer.
4. To save your changes, click **Apply**.

The next step starts the calibration process.

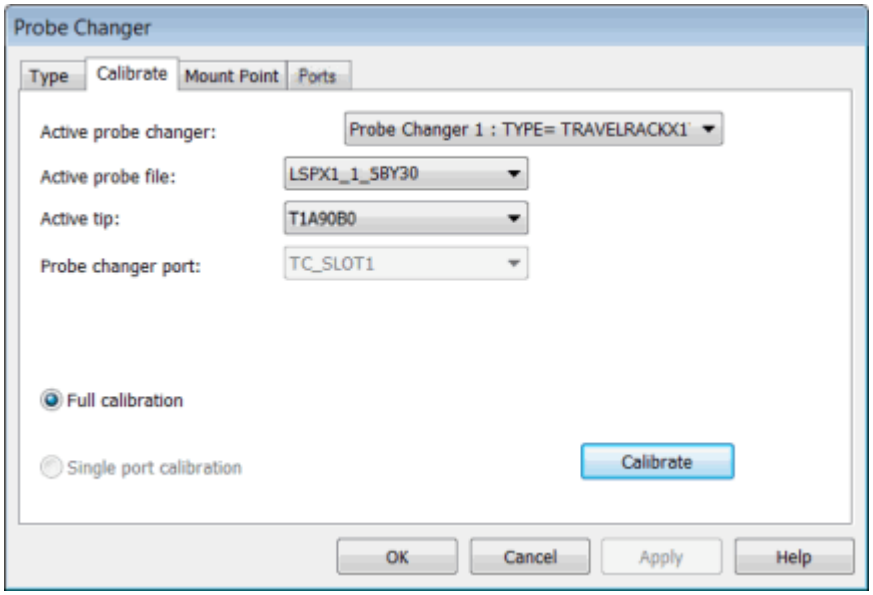
Step 4 - Prepare for Calibration

This step starts the calibration process for the Travel Rack HR-X1-TR. The calibration process requires a tip with at least 50 mm of length. It should not exceed 5 mm in diameter.

To calibrate the probe changer, you need tip angles A90 and B90.

To start the calibration process:

1. Select the **Calibrate** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**):



Probe Changer Dialog Box - Calibrate Tab

2. In the **Active probe changer** list, select **Probe Changer 1: TYPE=TRAVELRACKX1V**.
3. The entry in the **Active probe file** list defaults to the current measurement routine setting. If this is not what is to be used for the rack calibration, select the proper probe.
4. The entry in the **Active tip** list defaults to the angles from the **Mount Point** tab, but this rack type requires a different tip for the calibration. It must be the one that points directly into the rack so that it can take measurements inside the ports. This is usually T1A90B90.
5. Click **Calibrate**.
6. When you are ready to take the first measurement, click **OK**.

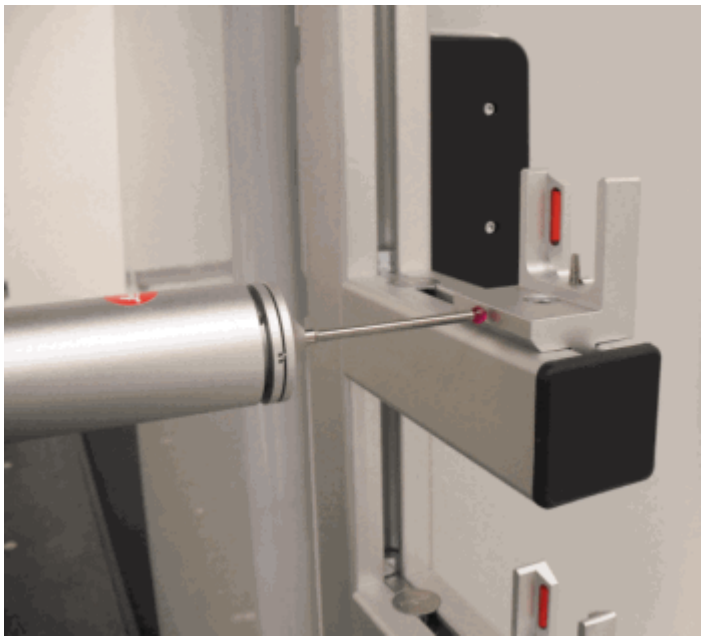
In the next step, you take the manual hits.

Step 5 - Take the Manual Hits

When you start the calibration procedure for the Travel Rack HR-X1-TR, a series of message boxes prompts you through the process of measuring the manual hits. These hits are required to fully define the orientation of this rack. Follow the prompts and take the needed hits as directed in the instructions for each hit.

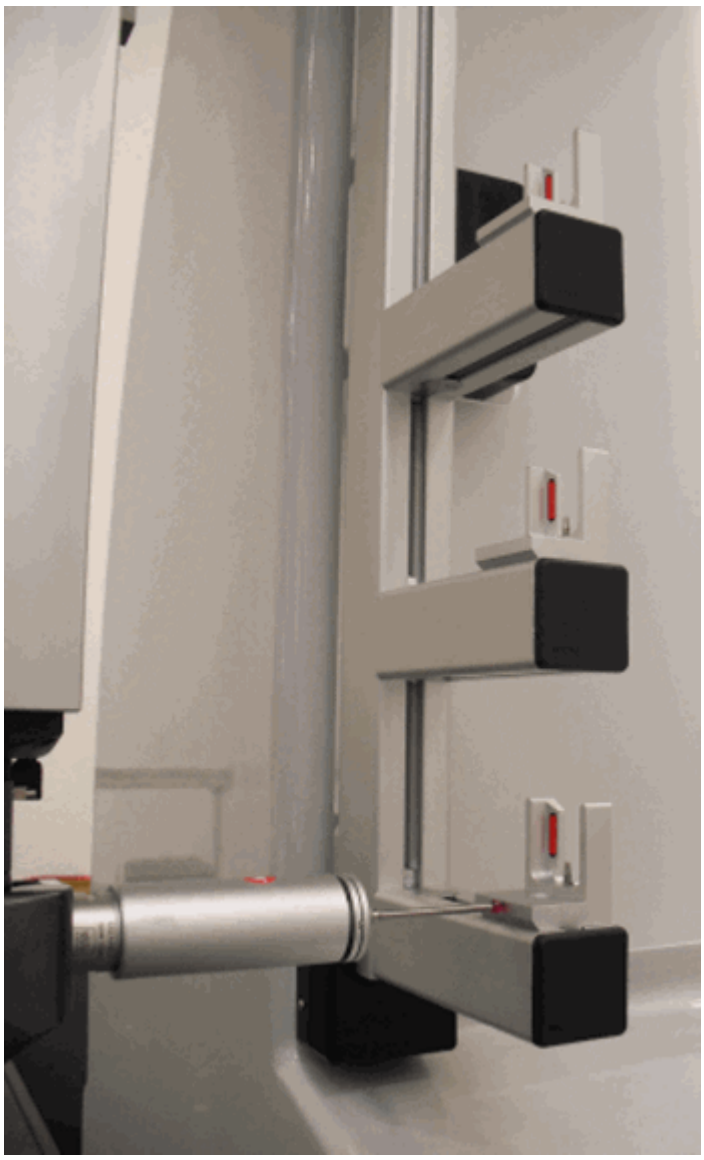
To take the manual hits:

1. Remove all of the probes that may be in any of the ports.
2. Take a horizontal hit on the flat, vertical front surface of port 1 (the top port) as shown in the view below:

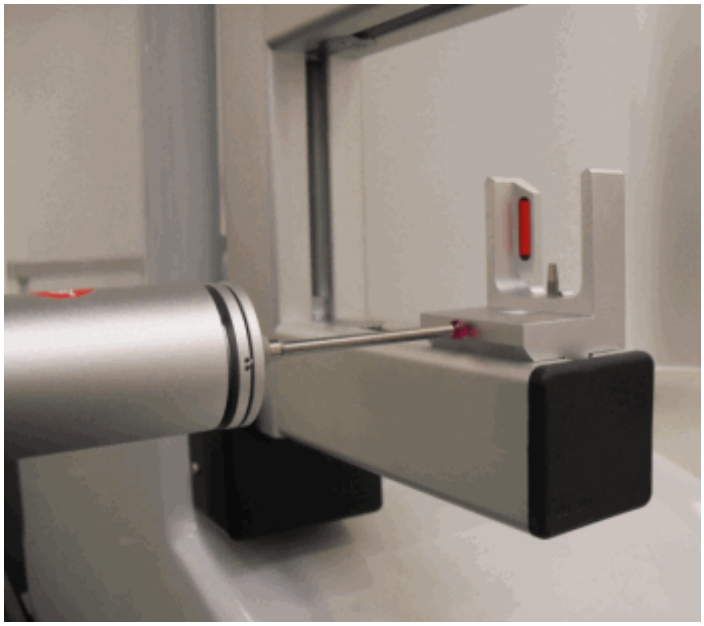


Horizontal hit on flat and vertical front surface of top port

3. Take a horizontal hit on the flat, vertical front surface of the bottom port as shown in the two views below:



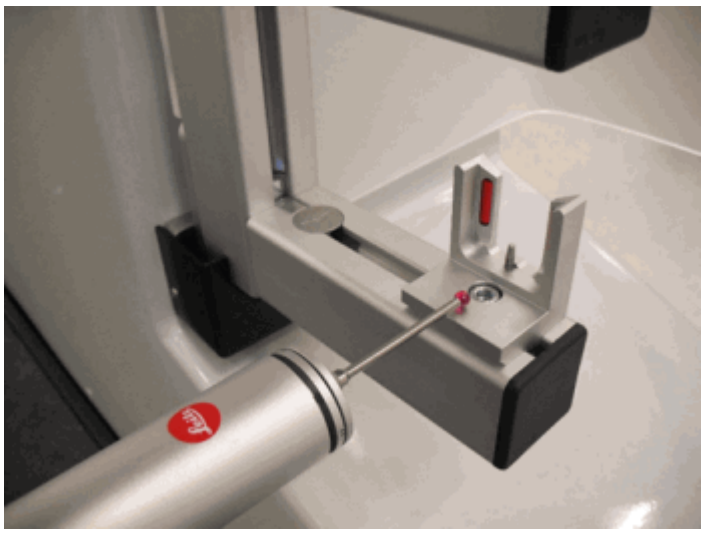
Horizontal hit on flat and vertical front surface of bottom port - view 1



Horizontal hit on flat and vertical front surface of bottom port - view 2

4. Take a vertical hit on the flat, horizontal surface of the bottom port anywhere between the edge and screw as shown in the two views below:

Vertical hit on flat and horizontal surface of bottom port - view 1



Vertical hit on flat and horizontal surface of bottom port - view 2

5. Now you are prompted to take a similar vertical hit on the flat, horizontal surface of each additional port, in the order of lower to higher, with the final hit being on the top port.

6. Use your machine's jog box to take all of the manual hits.

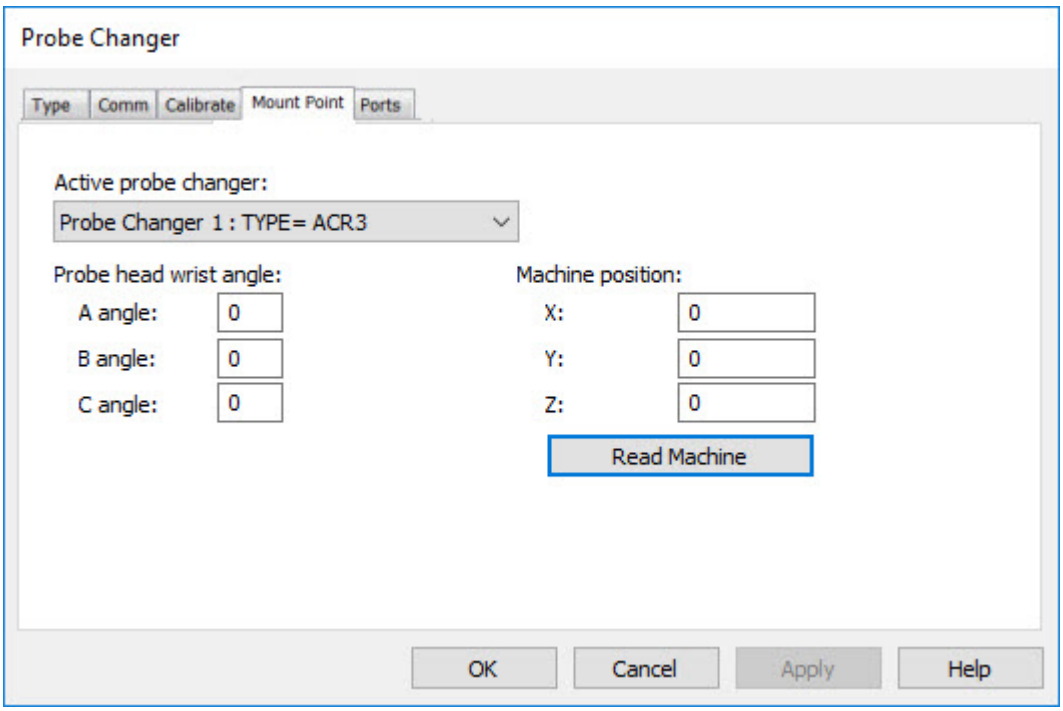
PC-DMIS performs the rest of the calibration in DCC Mode.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Mount Point Tab

You can use the **Mount Point** tab to change the values for the probe head wrist angles and define a safe position that the CMM can move to before each change cycle. You can do this for the probe changers that you defined on the **Type** tab.

To open the **Probe Changer** dialog box, select **Edit | Preferences | Probe Changer**.



Probe Changer Dialog Box - Mount Point Tab

Active probe changer

From the **Active probe changer** list, you can select the probe changer. For the probe changer, you need to define the probe head wrist angles to perform the change, and a CMM position. For more information about this list, see "Type Tab".

Probe head wrist angle

The **A angle**, **B angle**, and **C angle** boxes indicate the position of the probe head. You need these values when you slide the probe configurations in place. In the respective **Probe head wrist angle** boxes, you can type the values. If an angle is not available in the probe head, the value for that angle should be 0 (zero).

> Some probe changers do not support the C angle. The **C angle** box does not appear for those probe changers.

When you calibrate ports that hold extensions, you should define the probe head wrist angles before calibration. If you change the angles after calibration of the port, a warning message appears and informs you that recalibration is needed.

Machine position

You can use the **Machine position** area to indicate a safe location to which the CMM can move prior to each change cycle. Usually, this safe location is approximately 2 inches above and in front of the probe changer rack. However, you may find it necessary to move to some other location, depending on the type of probe changer.

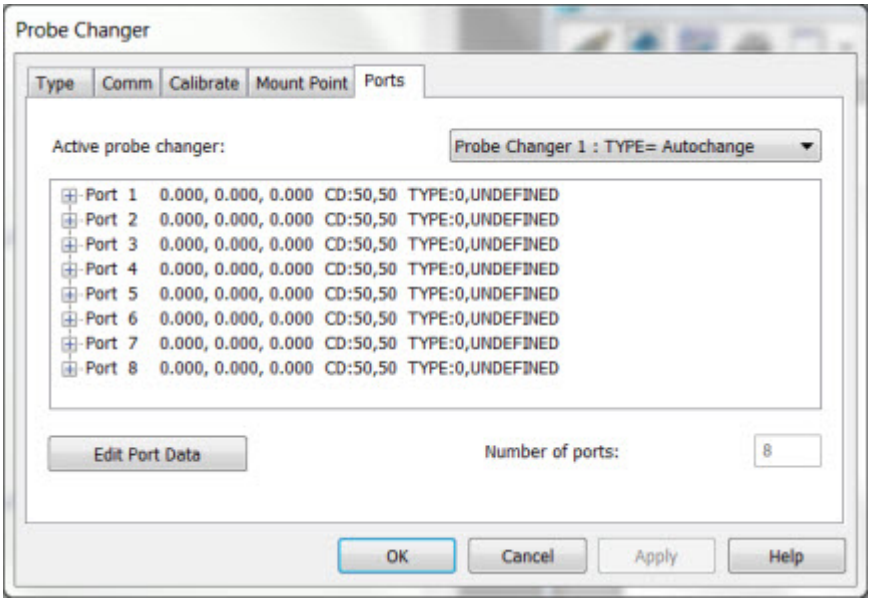
To change the values for the CMM position:

1. Select the previous value of the **X**, **Y**, or **Z** box that you want to change.

2. Type the correct values in the appropriate boxes. You can also click the **Read Machine** button to insert the current position of the CMM.

Ports Tab

To display the **Ports** tab in the **Probe Changer** dialog box, select **Edit | Preferences | Probe Changer**:



Probe Changer Dialog Box - Ports Tab

The probe changer is a rack that provides ports for different probe configurations. Use the **Ports** tab to define the probe configuration for each port that is used. The default for each port is "**(no probe)**". You can define the probe configuration for each port of the selected probe changer.

To Define the Probe Configuration for Each Port

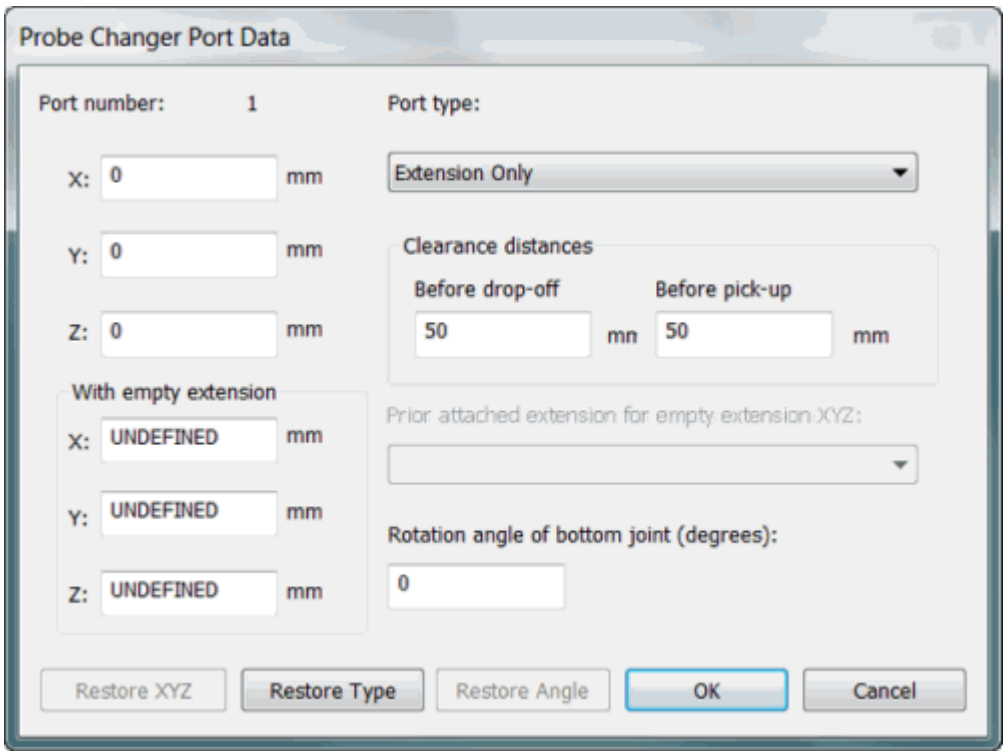
1. Select the appropriate probe changer from the **Active probe changer** list on the **Ports** tab (**Edit | Preferences | Probe Changer**).
2. Click the plus sign (**+**) to the left of the port number. PC-DMIS displays a "**(no probe)**" item as the default.
3. Double-click or right-click on the "**(no probe)**" item. A list appears.
4. Assign the appropriate probe file or extension to each port. (If you want to remove a probe file or an extension from a port, double-click it and then select the "**(no probe)**" item from the list.)
5. Select the port number and then click the **Edit Port Data** button. The **Probe Changer Port Data** dialog box appears. For help with this dialog box, see "To Manually Edit a Port's Data".

6. From the **Port type** list, specify the type of hardware that you want the ports to hold:
    - If you want it to hold a probe, select **Probe Attached.**
    - If you only want it to hold a probe extension, select **Extension Only.**
    - If the port is empty, select **UNDEFINED**.
    - For some probe changers, you can select specific inserts from the list.
7. Define the XYZ location for the port by typing the **X**, **Y**, and **Z** values for **Port number**.
8. If you are using an ACR1 or a TESASTAR-R / HR-R probe changer, and you have defined the port type as **Extension Only**, enter the value for the **Rotation angle of bottom joint (degrees)** box. You can also define the XYZ location of the bottom of the extension by defining the **X**,**Y**, and **Z** boxes in the **With empty extension** area.
9. Define the **Before drop-off** and **Before pick-up** clearance distance values as desired. For more information, see "About Port Clearance Distances".
10. Click **OK** to accept your changes.
11. Each probe configuration must be loaded into the rack using the **CMM Arm** and **Load Active Probe** option. (Do *not* insert the probe into the rack by hand.)

> **!** Extensions are not manufactured within the necessary tolerances to be interchangeable without recalibration. Therefore, if you have already configured a port for a specific extension, and you want that port to hold a different extension (even an extension of the same size), you need to recalibrate the port for the new extension.

To Manually Edit a Port's Data

1. Select the appropriate port from the **Active probe changer** list on the **Ports** tab (**Edit | Preferences | Probe Changer**).
2. Click the **Edit Port Data** button. The **Probe Changer Port Data** dialog box appears. For all probe changer types, the **X**, **Y**, and **Z** boxes should be available to edit. While you can use these boxes to manually enter a position if needed, these values are normally determined by measurements that result from running the calibration procedure for the probe changer. You can use this dialog box to change a port's type and XYZ location. For TESASTAR-R / HR-R and Autochange (ACR1) probe changers, you can also use it to edit an extension's XYZ location and edit the rotation angle of the bottom joint.



Example of the Probe Changer Port Data dialog box for an ACR1 Probe Changer.

Some probe changer types support multiple possible configurations for each port. Some example probe changer types that allow this include the Autochange (ACR1) and TESASTAR-R / HR-R.

- When you use a changer that has multiple possibilities for each port, the **Port type** list is active and filled with selections that are appropriate for that changer type.
- To specify the appropriate configuration, choose the appropriate entry from the list. For more in-depth information on the meaning of the selections available for a particular changer type, consult your hardware documentation for the probe changer.
- When port type selections are available, you should set them before you run the calibration procedure for the probe changer. If you change them later, this typically requires a recalibration of the probe changer or at least the individual ports that were changed.

Some probe changer types have port types that let you use empty extensions that do not have actual probes attached. Some example probe changer types that allow this include the Autochange (ACR1) and the TESASTAR-R / HR-R.

- When you use a probe changer that supports empty extensions, and if you have defined a port as **Extension Only**, the **With Empty Extension** area and the **Rotation angle of bottom joint (degrees)** box become active.
- While you can manually enter the XYZ location for the empty extension, this is normally determined from calibrating the probe changer.

- Some extension bars have a bottom joint that is rotated by some amount relative to the top joint. If you use one of these extension bars, you must define the amount of rotation in the **Rotation angle of bottom joint (degrees)** box before you calibrate the probe changer. Changing it later will require re-calibration of the probe changer, or at least the individual ports that were changed.

3. If desired, define separate **Before drop-off** and **Before-pick up** clearance distances in the **Clearance distances** area. For more information, see "About Port Clearance Distances".

4. After you specify the desired values, click **OK**. PC-DMIS closes the dialog box and shows a summary of the specifications for that port in the **Probe Changer** dialog box. If you changed any values, it also indicates "changes pending" for the port.

5. If you want to revert a port to its previous value, you can select that port and then click the **Edit Port Data** button. In the **Probe Changer Port Data** dialog box, click the **Restore XYZ**, **Restore Type**, or **Restore Angle** button as applicable. These buttons are only available if you made changes but have not yet clicked **OK** or **Apply** in the **Probe Changer** dialog box.

6. If you are ready to accept your changes, click **OK** or **Apply** in the **Probe Changer** dialog box.

To Change a Defined Probe Back to the "(no probe)" Default

1. Select the appropriate probe changer from the **Active Probe Changer** list on the **Ports** tab (**Edit | Preferences | Probe Changer**).

2. Double-click on the port to change. A list appears.

3. Select "(**no probe)**" from the list.

To Add or Remove Ports

You can define the number of ports for some types of probe changers.

1. Select the appropriate probe changer from the **Active probe changer** list on the **Ports** tab (**Edit | Preferences | Probe Changer**).

2. Change the value in the **Number of ports** box. (This box is unavailable if your probe changer does not allow you to change the number of ports.)

3. To immediately see the changes, select the **Apply** button.

To Drop off a Probe without Picking Up a New Probe

You may want to drop off the current probe into the probe changer without picking up a new probe from the changer. To do this:

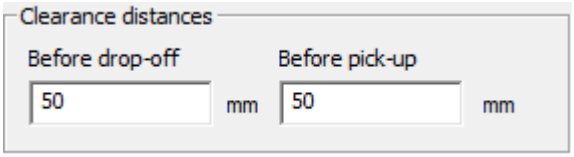
1. Create a dummy probe file in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**) for a probe that doesn't exist. Name it something like "unload".

   > The "dummy" probe used for the unload procedure must be completely defined. For details on how to define a new probe, see the "Defining Probes" topic in the "Defining Hardware" chapter.

2. Click the **Setup** button. The **Probe Setup** dialog box appears.
3. Select this probe file from the **Probe file used with probe changer for unload only** list.
4. Click **OK** to close the **Probe Setup** dialog box.
5. Open the **Ports** tab in the **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**). Ensure that the probe is NOT assigned to any port in the probe changer.
6. Click **OK** to close the **Probe Changer** dialog box.

About Port Clearance Distances

Use the **Clearance distances** area in the **Probe Changer Port Data** dialog box to define two different clearance distances from the port that your machine can use before dropping off and picking up hardware.



Clearance distances Area

You can also define clearance distances and tie them to specific probe tips. To do this:

1. From the menu, select **Insert | Hardware Definition | Probe** to display the **Probe Utilities** dialog box.
2. From the **Probe Utilities** dialog box, click the **Setup** button to display the **Probe Setup** dialog box.
3. Make your changes in the **Clearance distances** area and then click **OK**.

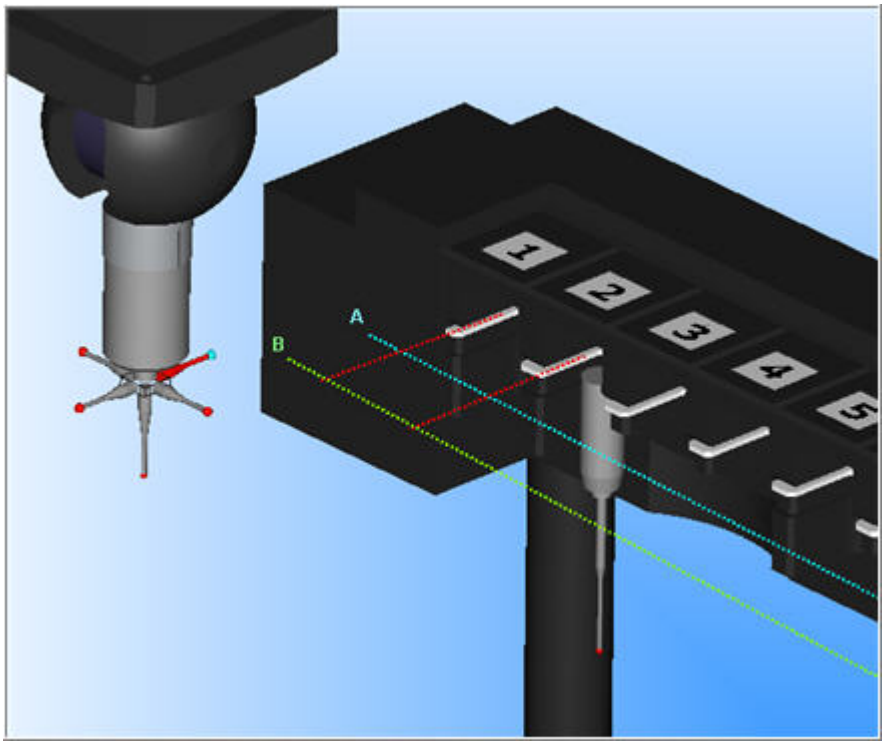
Any probe-specific values defined take precedence over those defined in the **Clearance distances** area. For more information, see "Using Probe-Specific Port Clearance Distances".

In versions prior to PC-DMIS 2009, there was only a single clearance distance that you could specify, and you had to use the PC-DMIS Settings Editor. Now you can define these different distances directly within PC-DMIS. These distances affect the motion during a probe change operation.

Before drop-off - Specifies the clearance distance PC-DMIS uses when the probe is still attached. This distance applies before dropping off a probe to the port, but it also applies after picking up a probe from this port.

Before pick-up - Specifies the clearance distance PC-DMIS uses when there is no probe attached.

The main reason for having these two distinct clearance values available to each port is to better accommodate probe configurations that have a stylus pointing back into the rack. In the example below, the star probe contains a stylus pointing towards the rack. Some example clearance lines are also provided on the image to illustrate the two separate clearance distances.



Example showing the clearance distance before pick-up (line A) and before drop-off (line B).

In these types of probe configurations, you may need an additional clearance distance to move to the clearance location without colliding with other probes stored in the rack. For example, using the sample image above, if you define a mount point in a central area of the rack (such as the area in front of port 4 or 5), then for a probe change that drops off the current star probe in port 1 and picks up the new one from port 2, when moving from the mount point to the clearance location for port 1, a collision could occur between the star probe's stylus that points toward the rack and the probe in port 2. In this case, a **Before drop-off** distance (indicated as line B) would provide the needed clearance. Yet, since that extra clearance distance is no longer needed when moving between the drop-off and pick-up ports with no probe attached, the **Before pick-up** distance (indicated as line A) is also beneficial.

Consider this Example

Again, using the image above, a typical probe change cycle, where the current probe is dropped off into port 1 and a new probe is picked up from port 2, could be described as follows:

1. The machine moves to the defined mount point for the probe changer.
2. The machine moves to the **Before drop-off** clearance location for port 1 (line B).
3. The machine moves into port 1 and drops off the current probe.
4. The machine moves back out to the **Before pick-up** clearance location for port 1 (line A).
5. It moves to the **Before pick-up** clearance location for port 2 (line A).
6. It moves into port 2 and picks up the new probe.

7. It backs out to the **Before drop-off** clearance location for port 2 (line B).
8. It moves back out to the defined mount point.

Affected Probe Changers

Although you can specify these clearance values for almost all tool changer types, it's possible that they may not have any effect in some cases. For some of the machine-specific changer types, where the change cycle is handled by the particular machine interface, it's possible that the interface has not implemented this functionality. Additionally, there may be some machine-specific changer types where the change cycle motion is handled by the machine controller itself and therefore is not under the direct control of PC-DMIS.

Initial Default Clearance Values

You can modify the appropriate entries in the PC-DMIS Settings Editor to change the initial default clearance values for specific probe changer types. These entries are in the **Option** section of the Settings Editor:

- `TCDefaultClearanceBeforeDropoff_<Changer>` - This is the before drop-off clearance distance in millimeters for the probe changer type referred to by <Changer>.
- `TCDefaultClearanceBeforePickup_<Changer>` - This is the before pick-up clearance distance in millimeters for the probe changer type referred to by <Changer>.

For example, a TP20 probe changer would have variable names of:

- `TCDefaultClearanceBeforeDropoff_TP20`
- `TCDefaultClearanceBeforePickup_TP20`

If you have not customized these new variables for a given changer type, PC-DMIS still looks for the possible existence of a customized value from the older clearance-related entry that was available for some types. If found, it applies that value to both clearances until you delete that older variable, or you save the new customized values using the above entries. You should consider these older clearance-related entries as deprecated:

- `AutochangeClearance`
- `ACR3Clearance`
- `FCR25Clearance`
- `SCP80Clearance`
- `TESASTAR-PRClearance`
- `TESASTAR-RClearance`

- `TP20Clearance`

Using Probe-Specific Port Clearance Distances

While you can use the **Probe Changer Port Data** dialog box to define a **Before drop-off** clearance distance for each port in a probe changer, this approach may not provide enough flexibility due to the specific needs of some probe and styli configurations. Sometimes, switching to a new measurement routine may require you to load a different probe configuration into a given port. Your new probe configuration may need a different clearance distance to avoid collisions when entering the port to drop it off or leaving the port after picking it up.

PC-DMIS provides you with the ability to define a clearance distance tied to the probe. This overrides any specified distance in the **Probe Changer Port Data** dialog box.

To Define a Probe-Specific Port Clearance Distance

1. Press F9 on the desired LOADPROBE command in your measurement routine to open the **Probe Utilities** dialog box.
2. Click the **Setup** button to open the **Probe Setup** dialog box.
3. Define a clearance distance in the **Clearance distance for tool changer port when this probe is loaded** box. The value uses the same measurement units as the measurement routine that is currently open.
4. Click **OK** to close the **Probe Setup** dialog box.
5. Click **OK** to close the **Probe Utilities** dialog box.

Any clearance distance that you specify in the **Probe Setup** dialog box overrides the **Before drop-off** value in the **Probe Changer Port Data** dialog box regardless of what port takes the probe. If you do not specify a clearance distance in the **Probe Setup** dialog box, then PC-DMIS uses the value from the **Probe Changer Port Data** dialog box.

With this approach, your probe changer setup can have values that may be adequate for most probes in general. This provides you with the ability to define probe-specific values for other desired probes.

Additional Notes

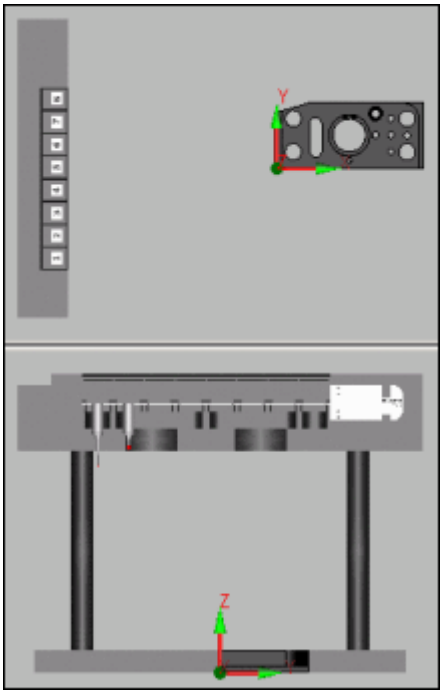
- The intent of this functionality is to avoid complications arising from particular styli configurations. Therefore, if you have a complex probe change operation where the stylus holder/configuration is kept in one port or probe changer and then extensions or other probe bodies are involved from another port or probe changer, this value only applies to the initial drop-off (when unloading) or final pickup (when loading) for the probe when the styli are attached. During intermediate portions of the overall probe change, where no styli are attached, PC-DMIS assumes that you do not need the extra clearance. During such intermediate portions, PC-DMIS uses the clearance value from the general probe changer setup instead.

- This clearance setting is used on the probe changer types that PC-DMIS handles directly (with the exception of the ACR2) in a machine interface-independent manner. For probe changer types where the changer is specific to a particular type of machine, and the change is handled within the associated machine interface, that machine interface may or may not make use of this value.

Showing an Animated Probe Changer

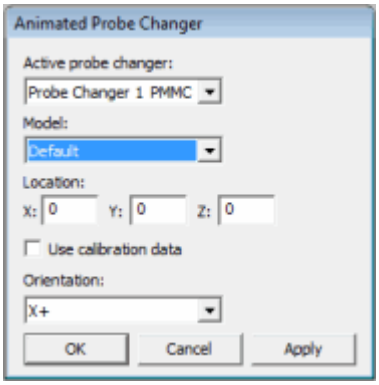
PC-DMIS provides the ability to display an animated graphical representation of a pre-defined probe changer inside the Graphic Display window.

To define a probe changer, see "Defining Probe Changers".



Example of an animated probe changer shown in the Graphic Display window in the Z+ view (top picture) and the X+ view (bottom picture)

Use the **Animated Probe Changer** dialog box (**Insert | Hardware Definition | Animated Probe Changer**) to specify the position and orientation of the probe changer.

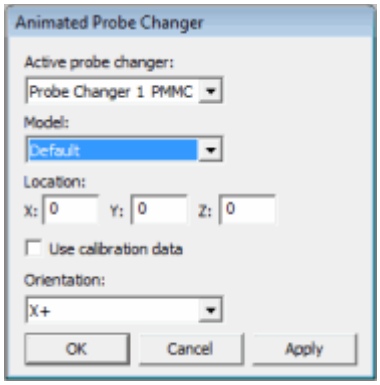


Animated Probe Changer dialog box

The items in this dialog box are discussed in the "To Display the Probe Changer in the Graphic Display Window" topic.

To Display the Probe Changer in the Graphic Display Window

1. Access the **Animated Probe Changer** dialog box (**Insert | Hardware Definition | Animated Probe Changer**).

   Animated Probe Changer Dialog Box

2. Select an existing pre-defined probe changer from the **Active probe changer** list. If you don't see any probe changers in this list, you can define a probe changer from the "Defining Probe Changers" topic in the "Defining Hardware" chapter.

3. The **Model** list allows you to animate a custom model of a PMMC probe changer. This list only becomes available for selection if you first select a PMMC probe changer from the **Active probe changer** list. The **Default** entry in the **Model** list is the standard PMMC model supplied with the PC-DMIS install. For custom items to appear in the **Model** list for selection, you need to first configure a customized PMMC model. To do this, follow the instructions in the "To Load in a Custom PMMC Probe Changer" topic.

4. Define the location and orientation of the probe changer. You can do this by either using calibration data for the location and orientation or by specifying the XYZ location and orientation directly into the dialog box.

   - To use existing calibration data select the **Use calibration data** check box. PC-DMIS fills in the location boxes with the XYZ values from the calibration.

- *To directly specify the location and orientation*, type values into the **X**, **Y**, and **Z** boxes, and then select an orientation from the **Orientation** list.

5. Click **Apply**. PC-DMIS draws the animated probe changer in the specified position and orientation in the Graphic Display window. PC-DMIS also inserts a `LOADPROBECHANGER` command into the Edit window.

6. Click **OK** when you're satisfied with the orientation and position.

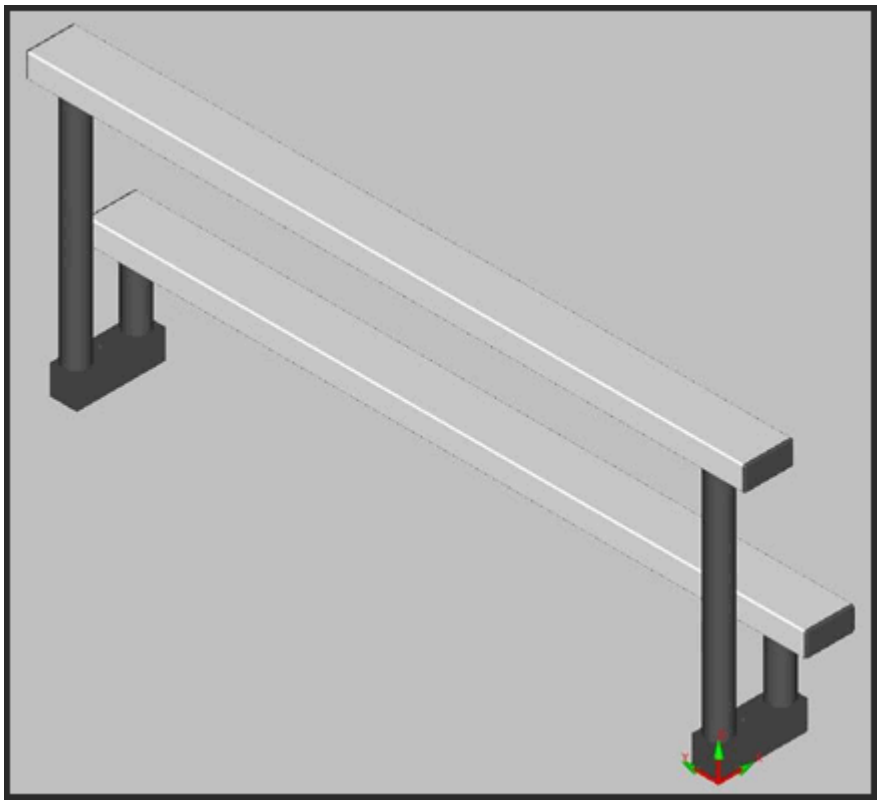
To Delete the Probe Changer from the Graphic Display Window

Access the Edit window, and delete the `LOADPROBECHANGER` command. This doesn't delete your pre-defined changer, just the animation of it inside the Graphic Display window.

To Load in a Custom PMMC Probe Changer Model

A custom PMMC probe changer model consists of two parts: the rack and the port or ports. These models have the following requirements:

- They need to be stored in a ".draw" file format.
- They need to have a consistent 0,0,0 coordinate-system location. This needs to be defined in the model's CAD creation software *before* you import the model into PC-DMIS. The trihedron in this image indicates where this position should be defined:

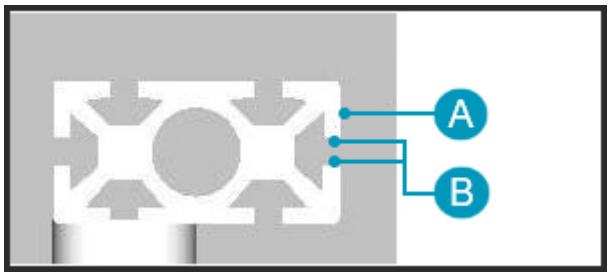


A sample Two-Tier PMMC model showing the 0,0,0 position (Red and Green trihedron).

- All the geometry must be stored in one CAD level.

You can load a custom PMMC Probe Changer model into PC-DMIS by following this procedure:

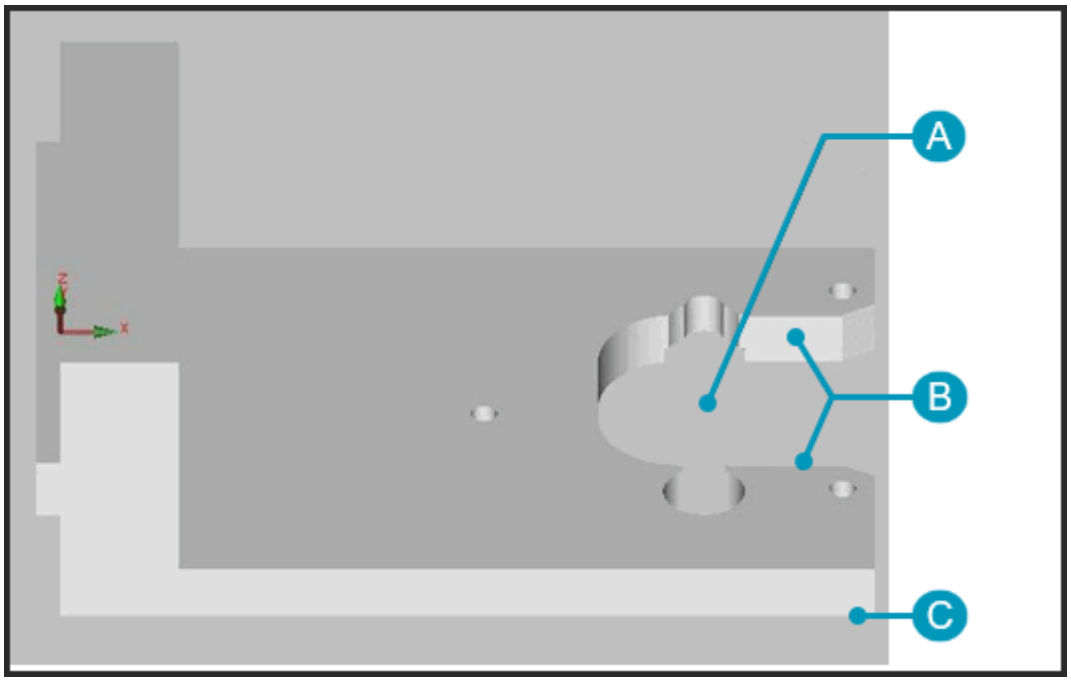
1. Create a new measurement routine.
2. Import the custom model files into PC-DMIS as you would a standard part model. PC-DMIS generates a .cad file for the model during import.
3. Find out what X and Z values should be for the rack. The Y value is not needed.



A sample model of the rack.

   - To determine the X value, measure a point on the surface, indicated by **A** in the above graphic. Make a note of the X value.
   - To determine the Z value, measure the midpoint between the two surfaces, indicated as **B** in the graphic above. Make a note of the Z value.

4. Find out what the X, Y, and Z values should be for the port.



A sample model of the port.

   - To determine the X value, measure the center point of the port opening, indicated by **A** in the above graphic. Make a note of the X value.

- To determine the Y value, measure the mid-point between the two surfaces, indicated by **B** in the above graphic. Make a note of the Y value.
- To determine the Z value, measure the bottom surface of the port, indicated by **C** in the above graphic. Record this value somewhere. Make a note of the Z value.

5. Inside Windows Explorer, rename the ".cad" extension to a ".draw" extension.
6. Copy and paste these .draw files into the same directory where the model files that ship with PC-DMIS are located. By default, these are located in the "Models\Toolchangers\" sub-folder under the PC-DMIS install directory.
7. Create an empty text file named userprobechanger.dat in the same directory as the probechanger.dat file. This should be in the "C:\ProgramData\Hexagon\PC-DMIS\2026.2" folder.
8. Open the userprobechanger.dat file inside of a text editor, and configure its contents so that a custom model entry follows the format given below. Items in angle brackets indicate where you would put your own text:

```
   ITEM:<DRAW FILE NAME> ARM
   comment lower <Rack X> 99999 <Rack Z>
   comment garage <Port X> <Port Y> <Port Z>
leitz_Ref_2-Tier_port.draw
   cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 1 <DRAW FILE
NAME>.draw
```

`<DRAW FILE NAME>` - The name of the .draw file (without the .draw filename extension).

`<Rack X>` - The X value of the rack from step 3 above.

`<Rack Z>` - The Z value of the rack from step 3 above.

`<Port X>` - The X value of the port from step 4 above.

`<Port Y>` - The Y value of the port from step 4 above.

`<Port Z>` - The Z value of the port from step 4 above.

For example, a finished entry in the userprobechanger.dat file for a single-tier rack might look something like this:

```
   ITEM:Custom_Model ARM
   comment lower 110.798 99999 394
   comment garage 82.75 4.675 -19 leitz_Ref_2-
Tier_port.draw
   cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 1 Custom_Model.draw
```

For information on defining a two-tier rack, see the sub-topic below.

9. If you followed the above procedure correctly, your custom PMMC model should now appear in the **Model** list of the **Animated Probe Changer** dialog box. See the "To Display the Probe Changer in the Graphic Display Window:" topic.

Defining a Two-Tier Rack

PC-DMIS also supports a two-tier rack (shown above in the first image of this topic). The line "comment upper <Rack X> 99999 <Rack Z>" is used to specify the upper tier on this type of rack. Therefore, a two tier model would require your entry in the .dat file to include both "comment upper" and "comment lower" lines, while a single-tier rack would need to include the "comment lower" line.

For example, the following entries show a two-tiered rack definition followed by a single-tiered rack definition:

```
ITEM:Leitz_Ref_2-Tier ARM
comment upper 82.5 99999 447.7
comment lower 182.5 99999 162.7
comment garage 82.75 4.675 -19 leitz_Ref_2-Tier_port.draw
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 2 Leitz_Ref_2-Tier.draw
ITEM:Reference_Frank ARM
comment lower 110.798 99999 394
comment garage 82.75 4.675 -19 leitz_Ref_2-Tier_port.draw
cadgeom 0 0 0 1 0 0 0 1 0 0 0 1 1 Reference_Frank.draw
```

Protecting Probes from Probe Changer Collisions

Depending on a variety of circumstances, including user errors, there’s the possibility that a physical crash happens during a probe change cycle. PC-DMIS provides multiple levels of protection against that event.

To avoid a collision in this situation, you can add this line of code to your measurement routine to enable the probe changer safety feature option:

```
ASSIGN/RESERVED_TOOLCHANGE_SAFETY_MODE=1
```

When you enable this entry and you have the above line of code in your measurement routine, if PC-DMIS detects that the wrong probe is loaded in the pickup slot of the probe changer than what is expected, it issues a warning message similar to this:

Click **OK** to continue without making any changes.

If you decide not to use this collision avoidance feature and PC-DMIS performs the probe change as described in the above scenario, it may cause a collision and result in damage to your equipment.

Click **Cancel** to end the execution of the measurement routine. With the execution of the measurement routine ended, you can fix the issue and then restart the measurement routine.

Working with Rotary Tables

Except where specified, the following Rotary Table commands were developed to support the PC-DMIS/NC (Numerical Control) for use on CNC machine tools. However, you can use them in regular CMM measurement routines as well. For more information on how to use CNC machines with PC-DMIS, see the PC-DMIS/NC Help file.

Ignore Rotary Table

When you have a rotary table defined for use, PC-DMIS usually assumes that you place the parts to measure on the rotary table. Essentially, PC-DMIS expects you to use the rotary table.

With the `IGNOREROTAB` command, you can determine whether the measurement routine uses the rotary table.

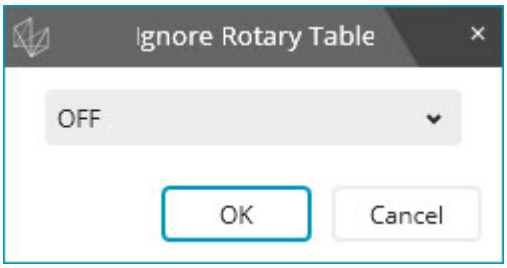
To insert an `IGNOREROTAB` command into the measurement routine, select **Insert | Parameter Change | Probe | Ignore Rotary Table**.

You can also insert an `IGNOREROTAB` command:

- When you type `IGNOREROTAB` and you are in Command Mode.
- When you are in Summary Mode, right-click and select **Add Command** to open the **Add Command** dialog box. From the dialog box, select **Ignore Rotary Table** and click **OK**.

Modify Ignore Rotary Table command in Summary Mode and Command Mode

You can press F9 on the `IGNOREROTAB` command to open the **Ignore Rotary Table** dialog box.



Ignore Rotary Table Dialog Box

From the list, select either **OFF**, **ON**, or **CALIBRATION**.

The options are as follows:

IGNOREROTAB/OFF

This command means PC-DMIS does not ignore the rotary table. In this state, PC-DMIS adjusts any measured data collected from the machine by using the data from the rotary table's calibration. This is the default state. If you want to measure parts with the use of a rotary table, you do not need to add this command.

```
IGNOREROTAB/OFF
```

IGNOREROTAB/ON

This command means PC-DMIS ignores the rotary table. Thus, the measured data that PC-DMIS collects does not have a rotary table adjustment applied to it. This might be used when a part that you want to measure is placed on a CMM table and not on a rotary table. You want to measure the part without using the rotary table even though your table exists.

```
IGNOREROTAB/ON
```

IGNOREROTAB/CALIBRATION

This command means PC-DMIS ignores the rotary table. You can use this command when you want to calibrate the rotary table.

```
IGNOREROTAB/CALIBRATION
```

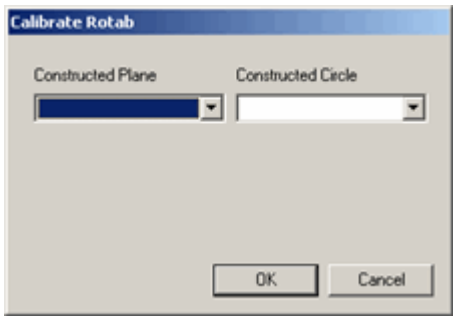
How to calibrate the rotary table using measurement routine

To calibrate the rotary table, follow these steps:

1. Attach a proper calibration artifact (a sphere) to the rotary table.
2. Add the `IGNOREROTAB/CALIBRATION` command to your measurement routine.
3. Measure the sphere at three or more positions of a rotary table.
4. Construct a plane through the resulting sphere centers.
5. Create an alignment to level this constructed plane to the axis of rotation of the rotary table.
6. Construct a circle through the resulting sphere centers.
7. Once you have constructed the plane and circle, you can use them as inputs to the `CALIBRATEROTAB` command.
8. Insert the `CALIBRATEROTAB` command. To insert the command, select the **Insert | Calibrate | Rotary Table from Features** menu item.

   ```
   CALIBRATEROTAB/PLANE=TABLE_PLN, CIRCLE=TABLE_CIR,
   MEAS/XYZ=0,0,0, MEAS/IJK=0,0,0
   ```

   This command tells a suitable measurement routine to calibrate the rotary table during measurement routine execution.
9. Press F9 on the command to display the **Calibrate Rotab** dialog box.



Calibrate Rotab Dialog Box

10. From the **Constructed Plane** list, select the constructed plane in step 4. From the **Constructed Circle** list, select the constructed circle in step 6.
11. Click **OK**. PC-DMIS updates the command to use the constructed features.
12. When you execute the measurement routine, PC-DMIS uses the constructed features to update the rotary table calibration data. The `MEAS/XYZ` and `MEAS/IJK` portions in the command block show the result for the center point and plane of rotation for the table.

For information on how to set up your rotary table, see the "Defining the Rotary Table" topic in the "Setting Your Preferences" chapter.

Rotate the Table to the Active Angle (Not Supported in PC-DMIS NC)

The actual rotary table angle on the machine and the active angle specified in your measurement routine don't always match. The **Operation | Rotate Table to Active** menu item automatically causes the actual table to rotate until its angle matches the active angle in the measurement routine at your cursor's location.

Configuring Advanced Mouse Devices

Most PC-DMIS systems use a standard mouse and keyboard configuration. Your setup may have more advanced hardware, such as a ZMouse or a 3D Mouse or motion controller. To configure those devices, see the topics below:

Configuring a Z-Rail Mouse

Certain systems offer a track ball style of mouse referred to as a ZMouse. This enables you to control the movement of the mouse pointer from the Z rail.

The last line of the parameter file (DOWNLOAD) should read:

N1000 ZMOUSE2
or
N500 ZMOUSE2

To enable the ZMouse, verify that the last line of the DOWNLOAD file reflects the above statement. Modify the line, if necessary.

Configuring 3Dconnexion 3D Mouse Devices

3Dconnexion 3D Mouse devices communicate with PC-DMIS through the 3DxWare interface. Information can be found in the following topics.

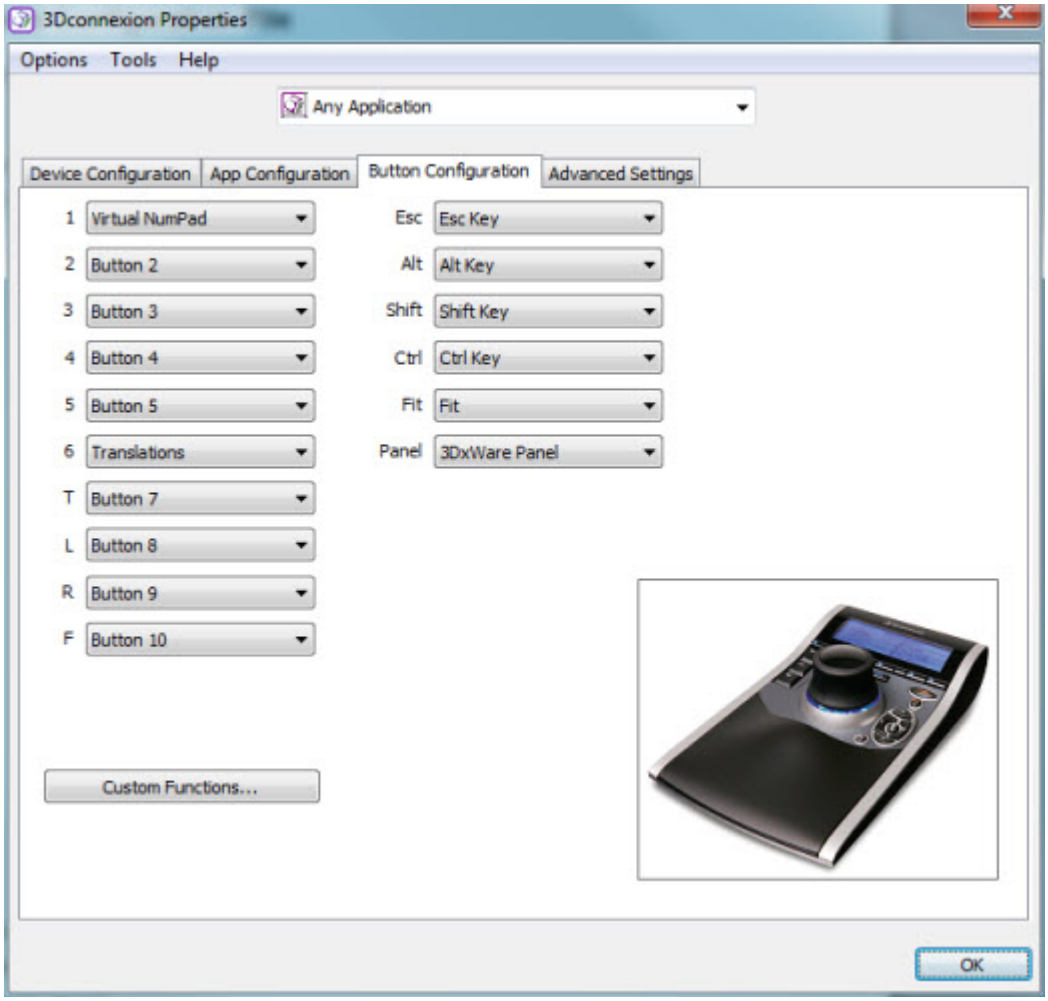
Contact a 3DConnexion representative for support on these devices.

Please contact Hexagon Technical Support for any 3rd-party files you may need for your hardware.

Editing the 3DxWare Configuration File

1. Start the PC-DMIS application.

2. Double-click the 3DxWare icon on your system tray to open the **3Dconnexion Properties** dialog box.



3Dconnexion Properties Dialog Box

   a. See if the **Application** list in the top-center of the **3Dconnexion Properties** dialog box contains an entry for something like "WAI Applications".

   b. If "WAI Applications" is available, then select it as the current configuration, and follow the information in the topic "Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices". Otherwise, continue with this procedure.

   c. If "WAI Applications" is not available, and you only see "Any Application", select **File | Save As** and save the name of the "Any Application" configuration to "My PC-DMIS", "WAI Applications", or whatever you like. PC-DMIS uses this configuration to define the rotation speed and other settings.

   d. Close the **3Dconnexion Properties** dialog box.

3. Close your PC-DMIS application.

4. Navigate to this directory, where <user> is your Windows user name:

   C:\Users\<user>\AppData\Roaming\3Dconnexion\3DxWare\profiles

5. Use a text editor (such as Microsoft WordPad or Notepad) to edit the most recent "user??.scg" file (for example, user00.scg).
6. In the "[IDENTIFICATION]" section of this file, compare the "APPLICATION_NAME" to what you entered in step 2c above to verify this is the file you just saved. Note that a comma might be preceded by a backslash (\).

   ```
   [IDENTIFICATION]

   APPLICATION_NAME = "My PC-DMIS"
   ```

7. In the "[CONFIGURATION]" section, edit the "APP_CONTROLS_BUTTONS" and "EXECUTABLES" values to enable them for use with the PC-DMIS executable.

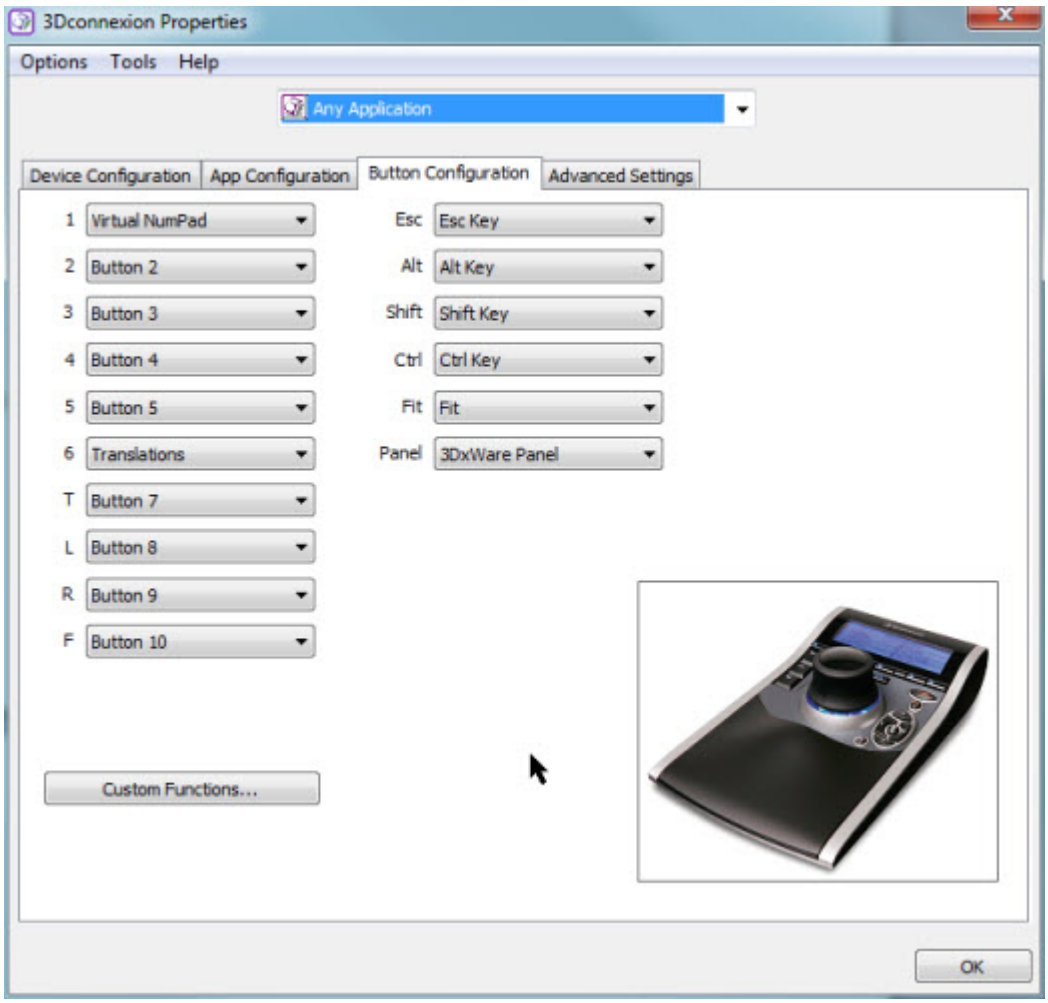
   ```
   [CONFIGURATION]

   APP_CONTROLS_BUTTONS = TRUE

   EXECUTABLES = { "PCDLRN.exe"}
   ```

8. As you can see, the above "EXECUTABLES" indicates that this configuration file is used by PC-DMIS.
9. Save and close the file.
10. Follow the information in the "Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices" topic to map custom functions to your device's buttons.

Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices

This topic describes how to map PC-DMIS or PC-DMIS Planner functionality to the available buttons on 3Dconnexion 3D Mouse devices by using the **3Dconnexion Properties** dialog box.

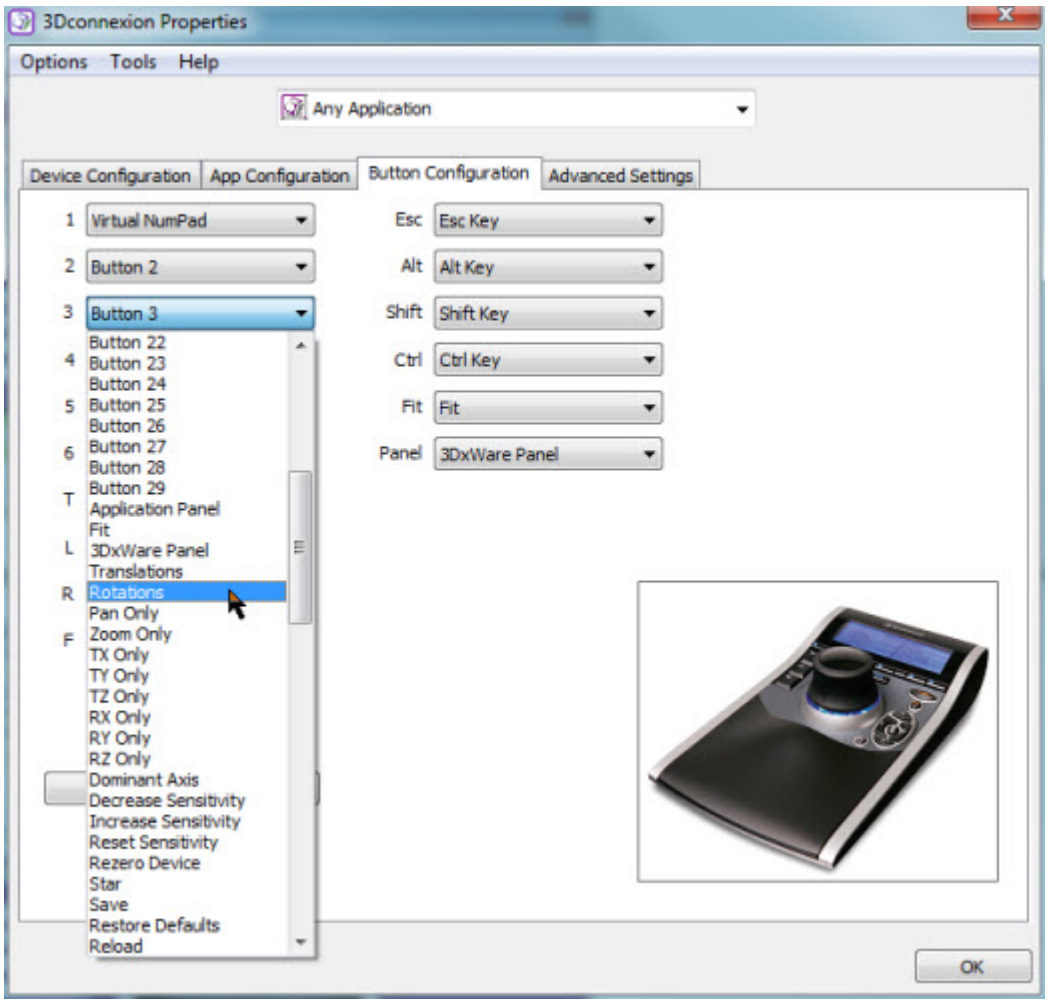
1. Start PC-DMIS or PC-DMIS Planner.
2. Load a measurement routine.
3. Double click the 3DxWare icon located in your System Tray. The **3Dconnexion Properties** dialog box opens showing a graphical depiction of your motion device.



3Dconnexion Properties dialog box showing a SpacePilot Device

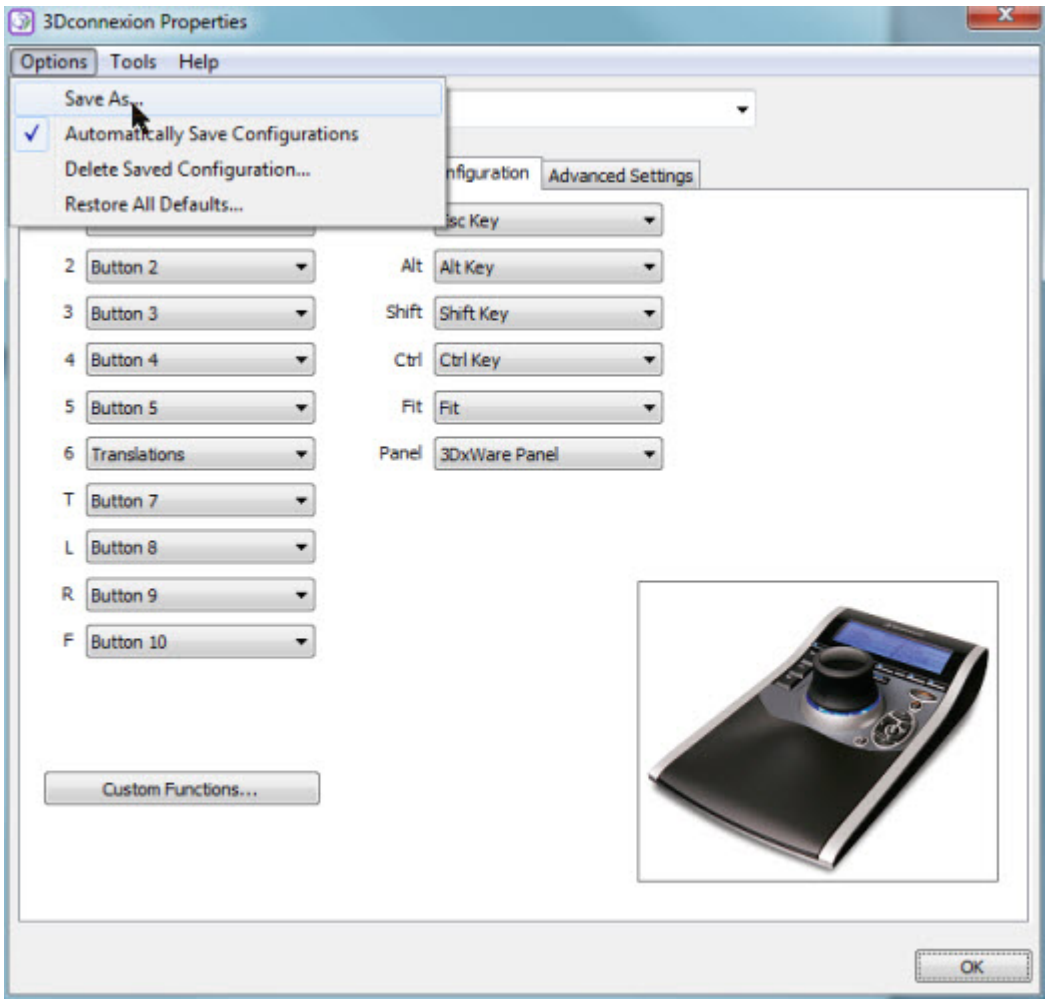
4. Select the correct configuration name from the **Application** list in the top-center of the **3Dconnexion Properties** dialog box. If a relevant name doesn't appear in the list, you probably need to add it to the 3DxWare configuration file. See "Editing the 3DxWare Configuration File".

5. To change map functionality to a button, select the button's list, click the down arrow icon to its right, and select the new function from the list.



3Dconnexion Properties dialog box for SpacePilot Showing Mapping of a Button

6. If the **Configuration | Automatically Save Configurations** menu item is checked, your changes will be saved in your current configuration file.



3Dconnexion Properties dialog box with the Automatically Save Configuration Menu Checked

7. Make any other changes as needed.
8. Click **OK** to accept your configuration and close the dialog box.

3Dconnexion 3D Mouse Device Modes

These devices are always in a certain mode. PC-DMIS displays a symbol in the lower right corner of the Graphic Display window to indicate the current mode for CAD PZR, PC-DMIS pan, or PC-DMIS Zoom.

Standard Modes

- Show cursor
- CAD PZR
- PC-DMIS rotate
- PC-DMIS pan
- PC-DMIS zoom

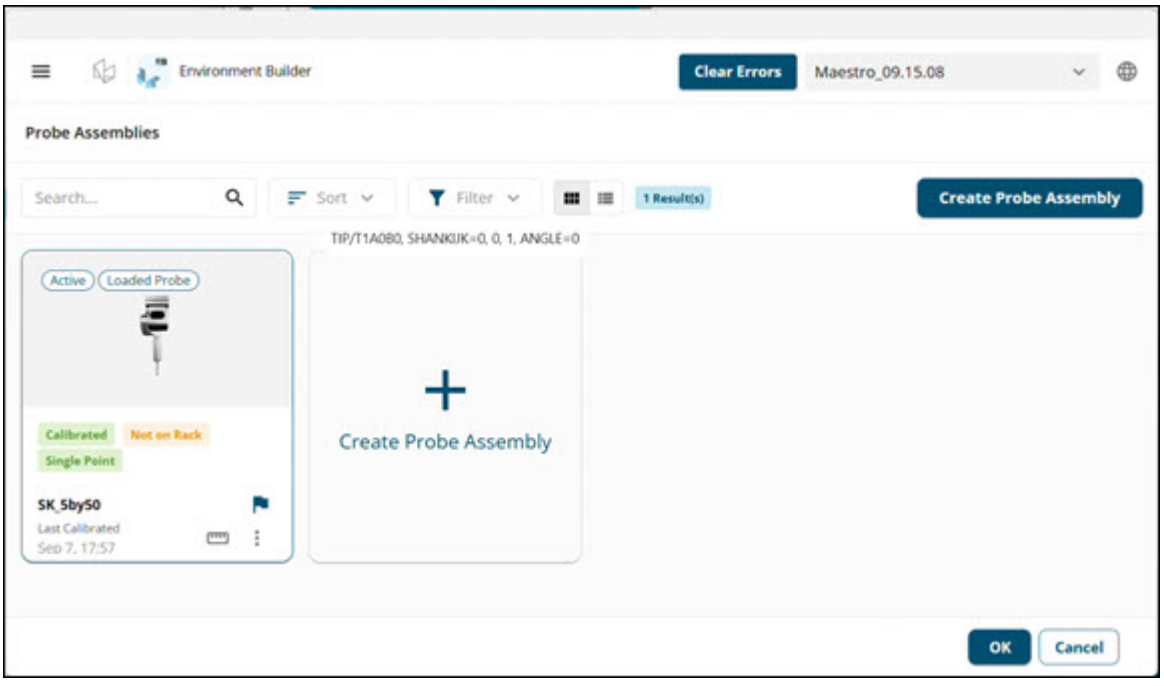
- CAD PZR dominant - The image in the lower-right corner of the Graphic Display window displays a "1" to signify that you can only use one of the three functions (Pan, Zoom, or Rotate) at a time.

Demo Mode

These devices can also place the Graphic Display window into a Demo mode. This mode causes the Graphic Display window to rotate automatically several times a second. To enter or exit the Demo mode, press the Ctrl key, and then press the programmed **Scale to Fit** button on the device.

Working with Environment Builder

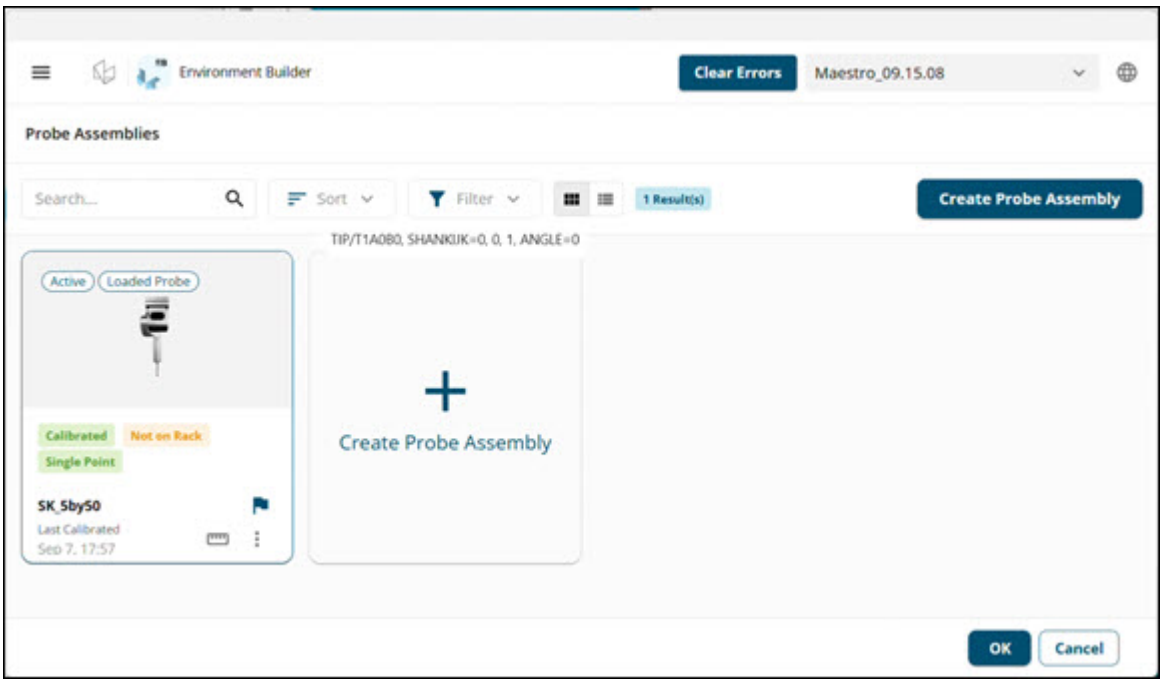
This section of the PC-DMIS documentation explains how to use the Environment Builder tool to define, build, and edit probe assemblies, probe changers, and machines for Maestro CMM users.



Environment Builder Window

Environment Builder - Getting Started

If you are a Maestro CMM user, your system must be properly configured to use the Environment Builder tool. An Applications Engineer should have already configured your system. If this is not the case, reach out to Hexagon Manufacturing Intelligence Customer Support.



Environment Builder Dialog Box

When your system is configured for a Maestro CMM, you can open the Environment Builder tool in these ways:

- Each time you create a new PC-DMIS measurement routine, the **Environment Builder** tool opens automatically.
- When your cursor is on the `LOADPROBE` command in the Edit window, press the F9 key to open the Environment Builder tool.
- You can press the keyboard combination of Ctrl + Alt + P to open the Environment Builder tool anywhere within the PC-DMIS environment.
- You can select the PC-DMIS **Insert | Hardware Definition | Probe** menu option to open the Environment Builder tool.
- You can select the PC-DMIS **Operation | Calibrate/Edit | Probe Changer** menu option to open the Environment Builder tool.

To prevent loss of data, before you open the Environment Builder tool, make sure you save your PC-DMIS measurement routine.

Be aware that when you modify probe assembly settings, it can affect measurement accuracy and efficiency of your measurement routine.

Tips for Efficiency

Familiarize yourself with the keyboard shortcuts (such as the F9 key and the Ctrl + Alt + P key combination) so you can quickly access the Environment Builder tool.

Use the calibration and editing features of the Environment Builder tool to maintain optimal performance of your probe assemblies.

The Environment Builder Menu

To open the slide-out menu panel, select the menu button ( ) in the upper-left corner of the window. Use the panel to open any page of Environment Builder:

- **Probe Assemblies** – This page shows the probe assemblies defined for the current machine. For information on how to create probe assemblies, see "Creating Probe Assemblies".
- **Results** – This page shows the results of your probe assembly calibrations. For information on how to review calibration results, see "Reviewing Calibration Results".

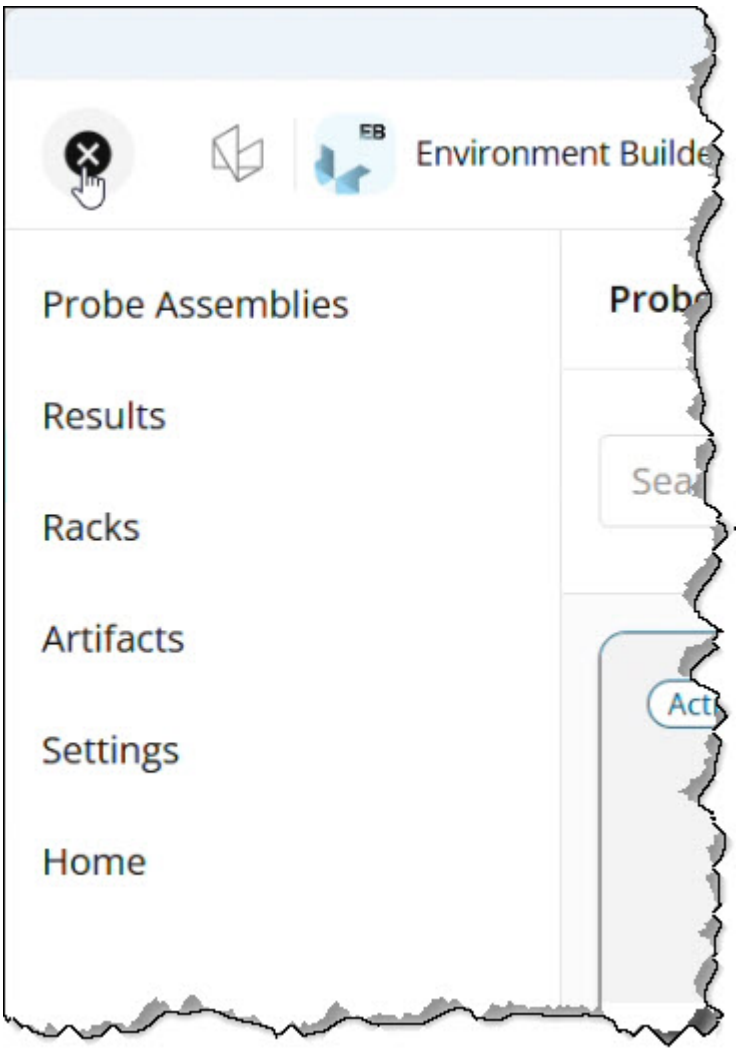
  This option is not available in offline mode.

- **Racks** – This page shows the probe changers on the CMM table and the probe assemblies assigned to them. For information on probe changers, see "Probe Changers".
- **Artifacts** – This page shows the calibration artifacts on the CMM table. For information on how to add calibration artifacts, see "Adding Calibration Artifacts in Environment Builder".
- **Settings** – Use this page to configure the calibration limits, clearances, and interface preferences that Environment Builder applies. For information on how to configure Environment Builder settings, see "Configuring Environment Builder Settings".

  This option is not available in offline mode.

- **Home** – Use this page to set up the machine and probe head that your environment is built around. For information on how to set up your machine, see "Setting Up Your Machine".

To hide the panel, select the **Close** button ( ). You cannot open the menu while a calibration is in progress.

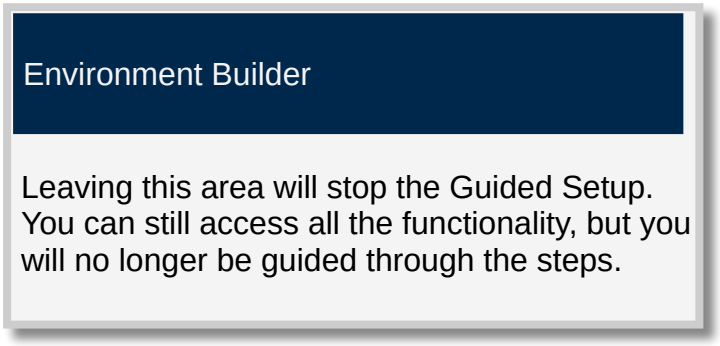


Using the Guided Setup

The first time you use Environment Builder, it offers a Guided Setup that guides you through the minimum steps required to set up a working environment. The Guided Setup includes these steps, in order:

- **Machine Setup** – In this step, you select the machine and probe head.
- **Reference Probe** – In this step, you build the probe assembly that Environment Builder uses as the reference probe for calibration.
- **Choose Artifact** – In this step, you select and place a calibration artifact.
- **Measure Artifact** – In this step, you measure the artifact you placed.
- **Choose Rack** – In this step, you select and place a probe changer.
- **Measure Rack** – In this step, you align and calibrate the probe changer.

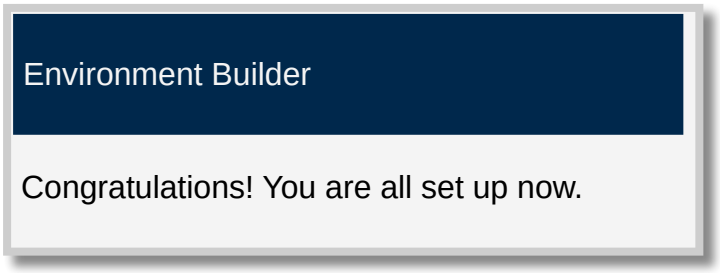
Select **Continue to Rack Setup** to move from the artifact steps to the rack steps. Environment Builder keeps you in the Guided Setup until you complete it. If you navigate away from the setup, Environment Builder displays the following confirmation message:



Select **Yes, leave** to exit the Guided Setup, or select **Cancel** to remain on the page.

When Environment Builder runs in offline mode, the Guided Setup skips the calibration steps. If the connection to the machine changes while you are using the Guided Setup, Environment Builder notifies you that the setup steps have changed and retains the data you already entered.

When you complete all the steps, Environment Builder displays the following message:



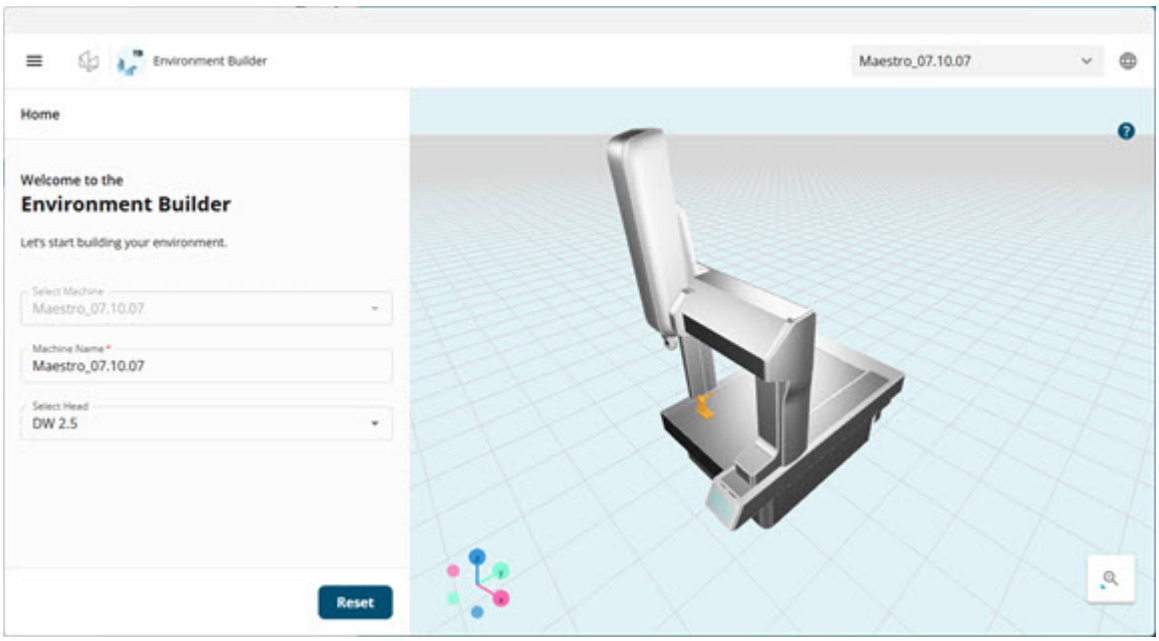
Select **Let's Get Started** to begin working.

Setting Up Your Machine

The **Home** page is where you select the CMM and the probe head that Environment Builder uses to build your environment. Environment Builder shows the **Home** page the first time you start the tool, and you can return to it at any time.

To set up your machine:

1. Open the slide-out menu panel and select the **Home** menu option.
2. From the **Select Machine** list, select your CMM. If Environment Builder detects your machine, it displays the message "Machine is auto-detected." and selects the machine automatically.
3. In the **Machine Name** box, type a nickname that identifies the machine.
4. From the **Select Head** list, select the probe head installed on the machine.
5. Select the **Save Machine** button.



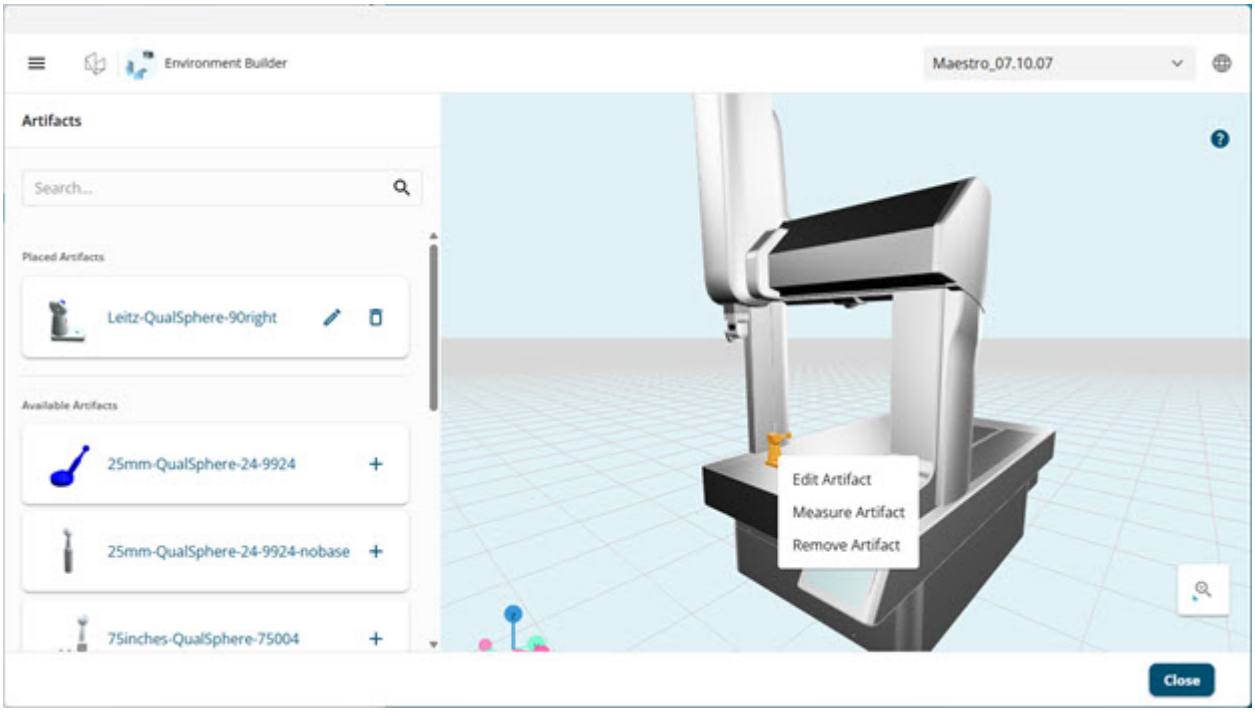
Machines that you have already defined appear under **Available Machines**. Select **New Machine** to define another machine, or select **Delete Machine** to remove a machine. If you delete the machine that you are currently using, Environment Builder resets the environment.

- Changing the probe head on a saved machine affects the probe assemblies. If the new head has the same connection type as the current head, Environment Builder updates the probe assemblies to use the new probe head. Otherwise, Environment Builder deletes the probe assemblies. In both cases, Environment Builder also deletes all probe calibrations and rack assignments.
- After you complete the initial setup, Environment Builder locks the machine selection. Select the **Reset** button to clear the selection.
- Your license limits the number of machines that you can define. When you reach this limit, Environment Builder displays the following message: "Upgrade your license to create more machines."
- Machine selection is unavailable when Environment Builder runs in the read-only user role. For information on the read-only user role and offline mode, see "Working in a Read-Only Role or Offline".

To configure how Environment Builder connects to the machine, select the **Settings** menu option, open the **Machine Configuration** tab, and then configure the **Controller Type**, **Firmware Type**, **CMM IP Address**, and **CMM IP Port**.

Navigating and Working in the 3D Scene

Environment Builder shows your machine, probe changers, calibration artifacts, and probe assemblies in an interactive 3D scene.



Example of a context menu for a selected calibration artifact in the 3D scene.

Help - To view the mouse controls currently in effect, hover over the question mark icon. Environment Builder supports two sets of mouse controls. To change the mouse control, open the slide-out menu panel, select the **Settings** menu option, and then select the **Mouse Controls** tab:

- **PCDMIS FUSION** – Drag with the left mouse button to orbit, drag with the right mouse button to pan, and scroll the mouse wheel to zoom in or out. This is the default option.
- **PCDMIS** – Drag with the middle mouse button (mouse wheel) to orbit, drag with the right mouse button to pan, and scroll the mouse wheel to zoom in or out. In this mode, the wheel zooms in the opposite direction to match PC-DMIS.

In both modes, you can hold the Ctrl key and drag with the right mouse button to orbit instead of pan. On a touch device, drag with one finger to orbit, drag with two fingers to zoom, and hold the spacebar while you drag with one finger to pan.

Camera Focus - Select **Camera Focus** and then select the object you want to center in the 3D scene.

To open a context menu, right-click in the 3D scene. The contents of the menu depend on the object you right-click:

- The CMM table – **Add Artifact**, **Add Rack**, and **Create Probe Assembly**.
- A calibration artifact – **Edit Artifact**, **Measure Artifact**, and **Remove Artifact**.
- A probe changer – **Edit Rack**, **Measure Rack**, **Place Probe Assembly**, **Clear Rack**, **Remove Rack**, and **Switch On** or **Switch Off**.
- A module on a probe changer – **Edit Module** and **Remove Module**.
- A probe assembly on a probe changer – **View Probe Assembly** and **Clear Port**.

Select **Hide 3D Scene** to collapse the scene and provide more space for the rest of the window. Select **Show 3D Scene** to display the scene again.

When you position racks and artifacts, Environment Builder indicates whether snapping is enabled or disabled by displaying **Snapping Enabled** or **Snapping Disabled**.

Configuring Environment Builder Settings

To configure how Environment Builder behaves, open the slide-out menu panel, and then select the **Settings** menu option. Environment Builder groups the settings into these tabs:

- **Machine Configuration** – Use this tab to define the **Controller Type**, **Firmware Type**, **CMM IP Address**, and **CMM IP Port** that Environment Builder uses to communicate with your machine.
- **Calibration Limits** – Use this tab to define the **Sphere Standard Deviation Limit** and **Tip Diameter Error Limit**. When a calibration exceeds either limit, Environment Builder displays a notification for the corresponding deviation type.
- **Clearance Distance** – Use this tab to define the clearances that Environment Builder applies during qualification: along the qualification tool shank vector for non-continuous wrists, along the qualification tool shank vector for continuous wrists, in Z when the qualification tool shank is perpendicular to the Z axis, along the tip shank vector during qualification, and for the tool changer port when a probe is loaded.
- **Calibration Validity** – Use this tab to define the **Calibration Interval** after which a calibration is no longer valid.

  You can override the global calibration interval for a single probe assembly by using **Set Overdue Frequency**. For information on the probe assembly options, see "Probe Assembly Options".
- **Mouse Controls** – Use this tab to define the mouse control scheme for the 3D scene. For information on how to navigate and work in the 3D scene, see "Navigating and Working in the 3D Scene".
- **Precision** – Use this tab to define the number of decimal places that Environment Builder displays for numeric values.

Working in a Read-Only Role or Offline

Environment Builder can run in a read-only user role. In this role, you can inspect your environment but cannot change it. The menu and tile options display **View Probe Assembly**, **View Angles**, and **View Probe Assembly Settings** instead of the corresponding edit and calibration options. Environment Builder disables machine selection, and the boxes in the **Edit Artifact** and **Edit Rack** dialog boxes are read-only.

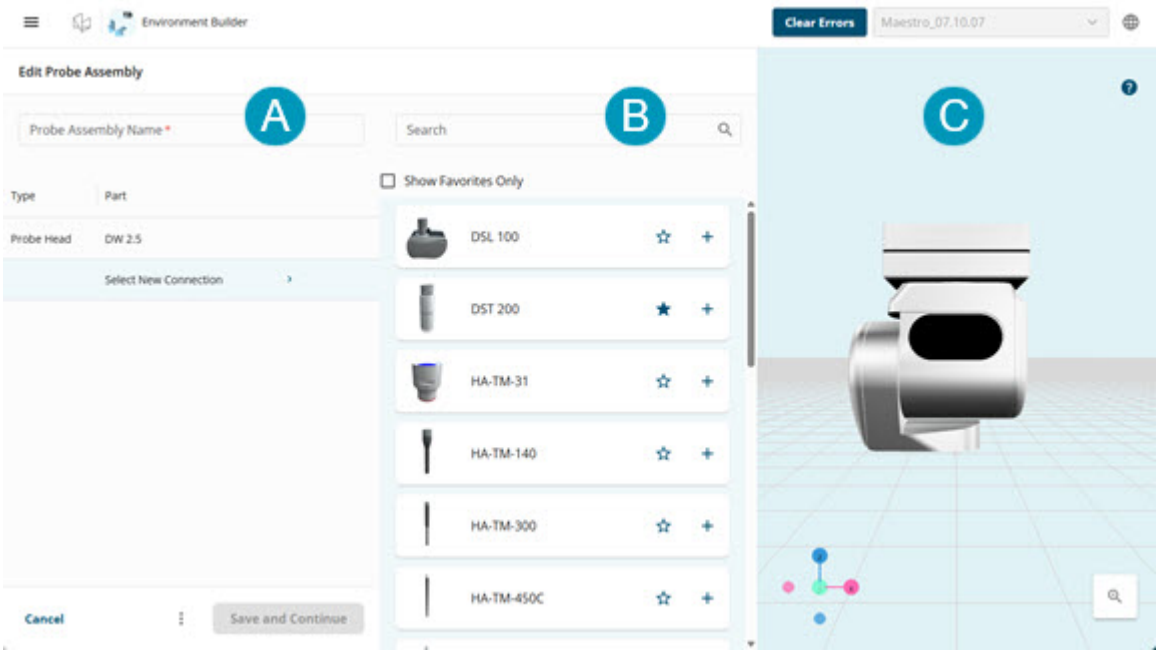
Environment Builder also runs when the connection to the machine is unavailable. In offline mode:

- Environment Builder displays an **Offline Mode** label and notifies you when the connection to the machine is lost or re-established.
- Calibration is unavailable. Calibration options display the message "Calibration is unavailable in offline mode."
- Environment Builder hides the **Results** and **Settings** menu options.
- The Guided Setup skips the calibration steps.

Creating Probe Assemblies

Environment Builder provides you with an intuitive workflow to design and create your probe assemblies.

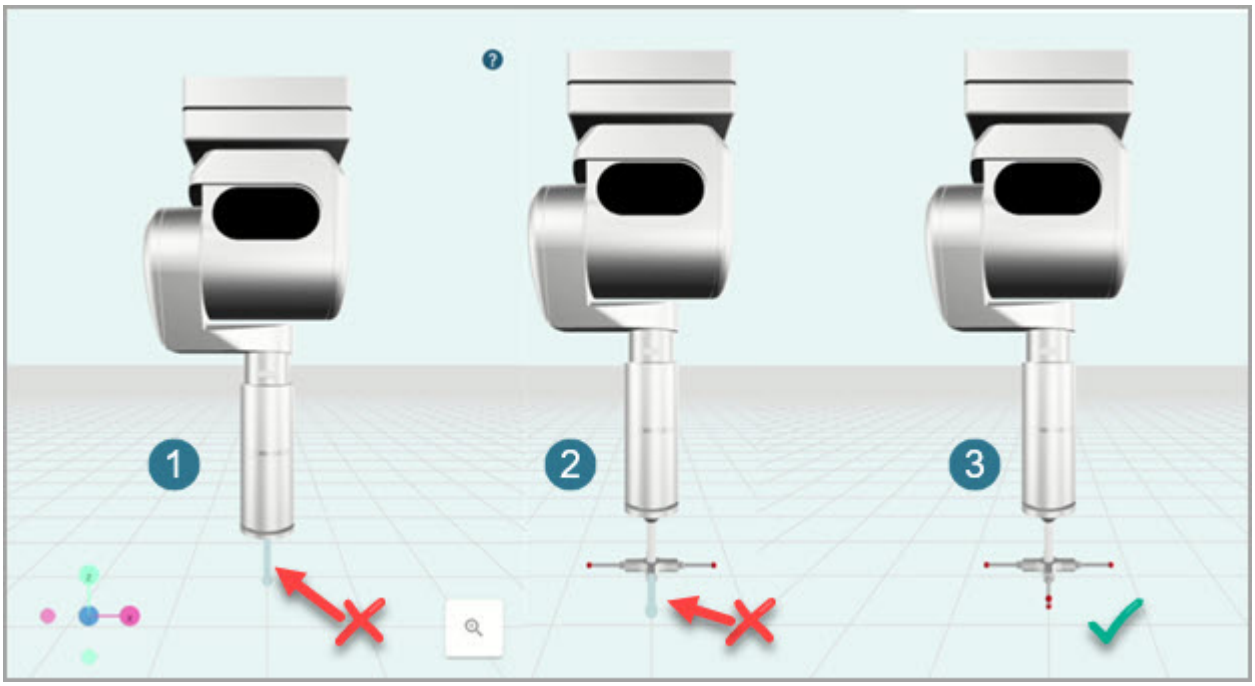
When Environment Builder starts, it typically displays the **Probe Assemblies** page that you can use to edit an existing probe assembly or create a new probe assembly. Select the **Create Probe Assembly** tile to show a blank **Edit Probe Assembly** window with a pre-selected probe head based on your machine configuration. An example is shown below:



Edit Probe Assembly Window

A - Probe Assembly details area
B - Probe Assembly component listing
C - Probe Assembly View area

Environment Builder shows the status of your probe assembly in the Probe Assembly View area as you build it. Below, you can see the stages of building a Star probe assembly. Environment Builder prompts you for the next component in a light blue shade when it detects that the probe assembly is incomplete.



Stages of building a Star probe assembly in Environment Builder

1. *Incomplete probe assembly with a DST-200 sensor component selected*
2. *Incomplete probe assembly with the 5-Way DST module selected*

3. *Completed Star probe assembly*

To change the view of your probe assembly:

- Right-click and drag in the Probe Assembly View area to move the probe assembly left-right, up-down, and diagonally.
- Rotate the wheel on your mouse to zoom in and out.
- Press and hold the wheel button and drag to change the probe assembly perspective.

You can also use the Trihedron to change the view of your probe assembly to the selected coordinate view. To do this, click one of the colored nodes on the Trihedron to rotate to the selected perspective.

Environment Builder Trihedron

X – Shown in pink, click on the X node or its opposite to rotate to the +X axis view or the -X axis view.

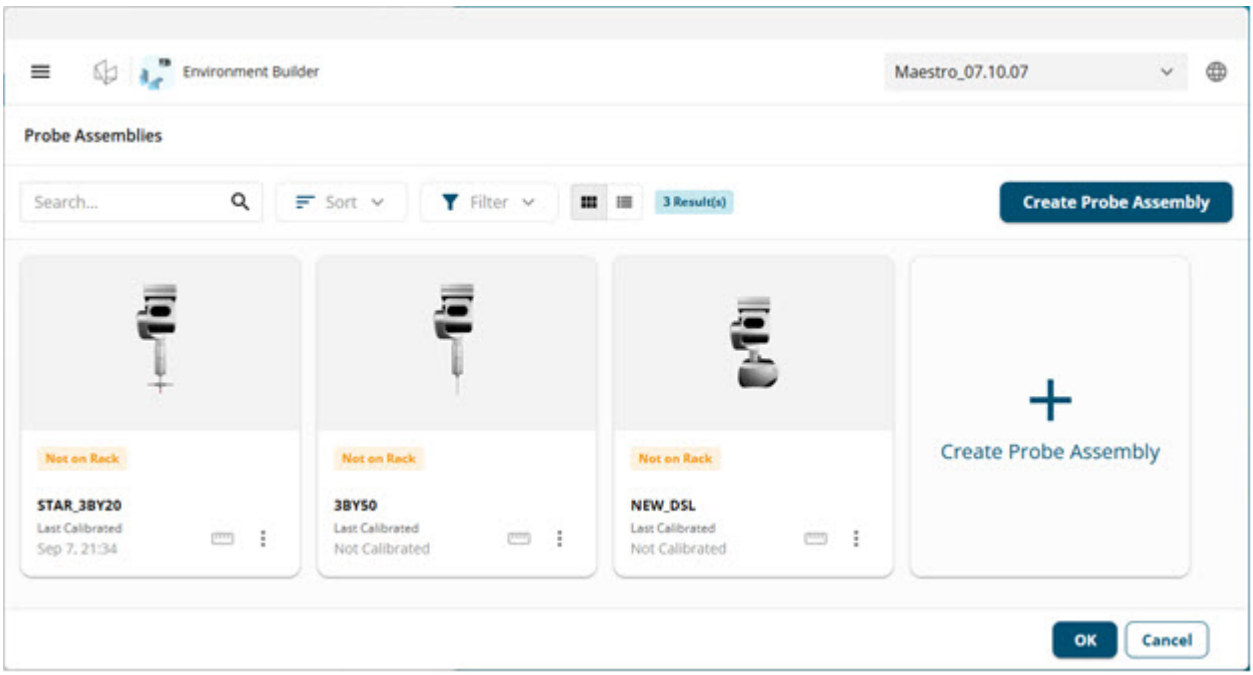
Y – Shown in blue, click on the Y node or its opposite to rotate to the +Y axis view or the -Y axis view.

Z – Shown in light cyan, click on the Z node or its opposite to rotate to the +Z axis view or the -Z axis view.

This section shows you how to create various probe assemblies that you can use as examples to design and create your own probe assemblies for your specific needs.

Managing the Probe Assemblies List

The **Probe Assemblies** page lists all probe assemblies defined for the current machine.



Search - Type all or part of a probe assembly name to filter the list to matching assemblies.

Sort - Select to change the order in which probe assemblies appear.

Filter - Select to narrow the list to probe assemblies that match the criteria you choose.

Card View - Select to show the probe assemblies as tiles.

List View - Select to show the probe assemblies as rows in a list.

Each probe assembly shows its calibration status:

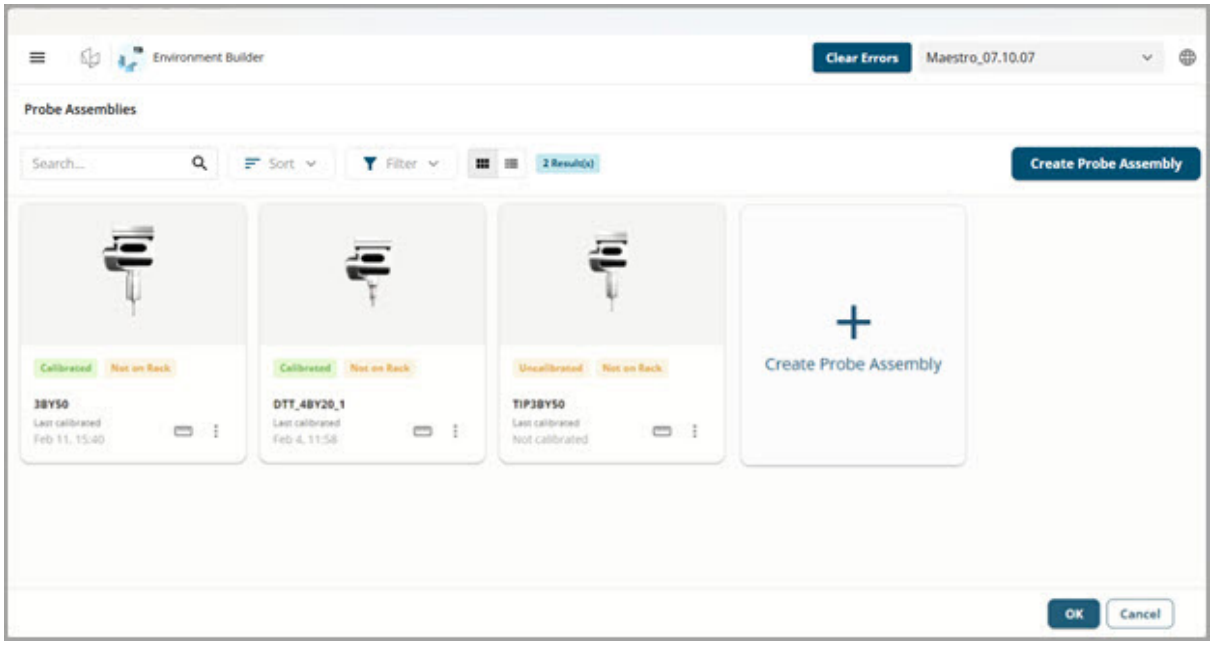
- **Calibrated** (green) – All selected angles are calibrated and the calibration is still valid.
- **Fast Calibrated** – Environment Builder calibrated the assembly using FastCal.
- **Partially Calibrated** – Some, but not all, angles are calibrated.
- **Uncalibrated** (orange) – The assembly has not been calibrated.
- **Calibration Overdue** – The calibration is older than the interval set in the Environment Builder settings or older than the assembly's own overdue frequency.
- **Calibration Error** – The last calibration failed.
- **Calibration in Progress** – A calibration is running for the assembly.

The tile also shows when the assembly was last calibrated, whether it is assigned as a reference probe, whether it is the loaded or active probe, and where its parts are on a probe changer: **Placed on Rack**, **Partially on Rack**, or **Not on Rack**.

Creating a DSL Probe Assembly

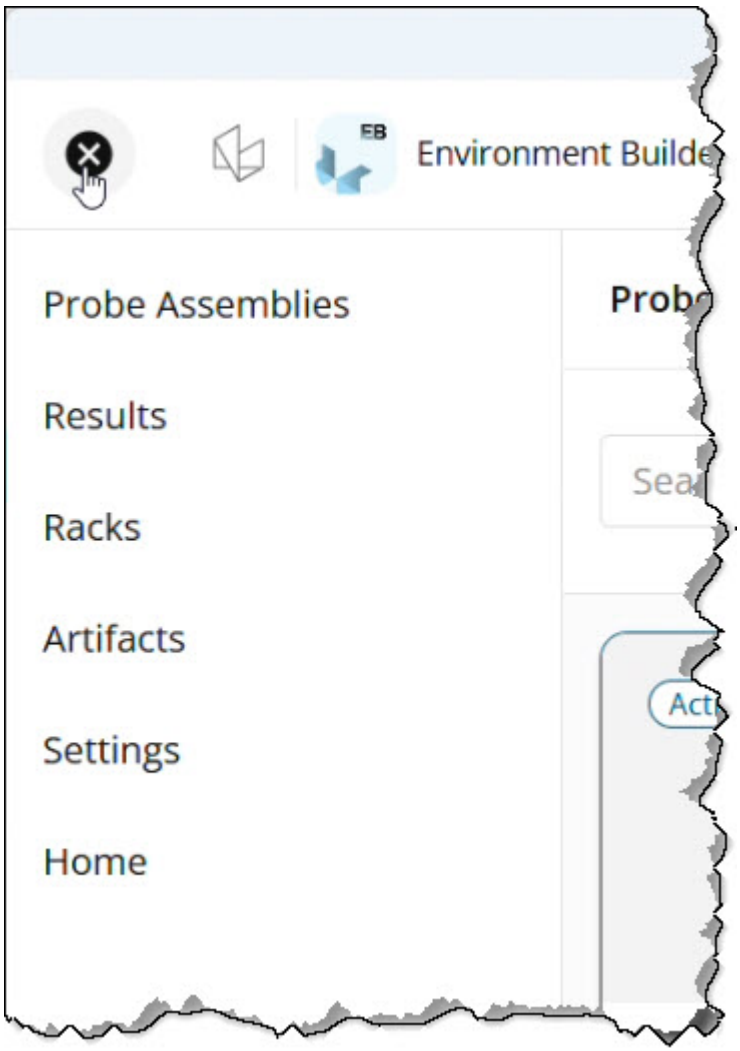
To create a Digital Scanning Laser (DSL) probe assembly with Environment Builder:

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

If you do not see the **Probe Assemblies** window, click the menu button ( ) located in the upper-left of the window to show the slide-out menu panel and select the **Probe Assemblies** menu option.

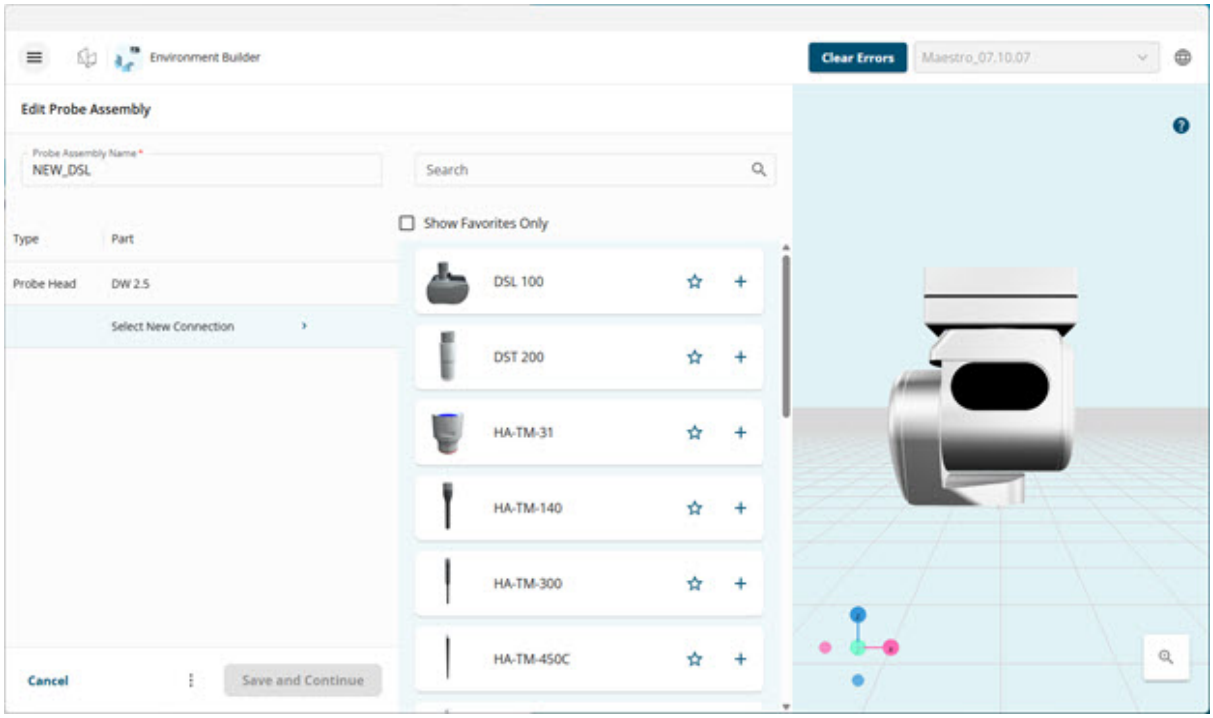


Click the **Close** button ( ) to hide the slide-out menu panel.

2. Select the **Create Probe Assembly** tile or button, and then confirm that the DW-2.5 probe head is selected for the machine configuration.

> As you select new components for your probe assembly, Environment Builder updates the probe component listing to only show components that are compatible with the last component you added.

3. For this example, from the list of sensors that are compatible with the selected DW-2.5 probe head, select the DSL-100 sensor.



Environment Builder dynamically updates the probe assembly with the selected components.

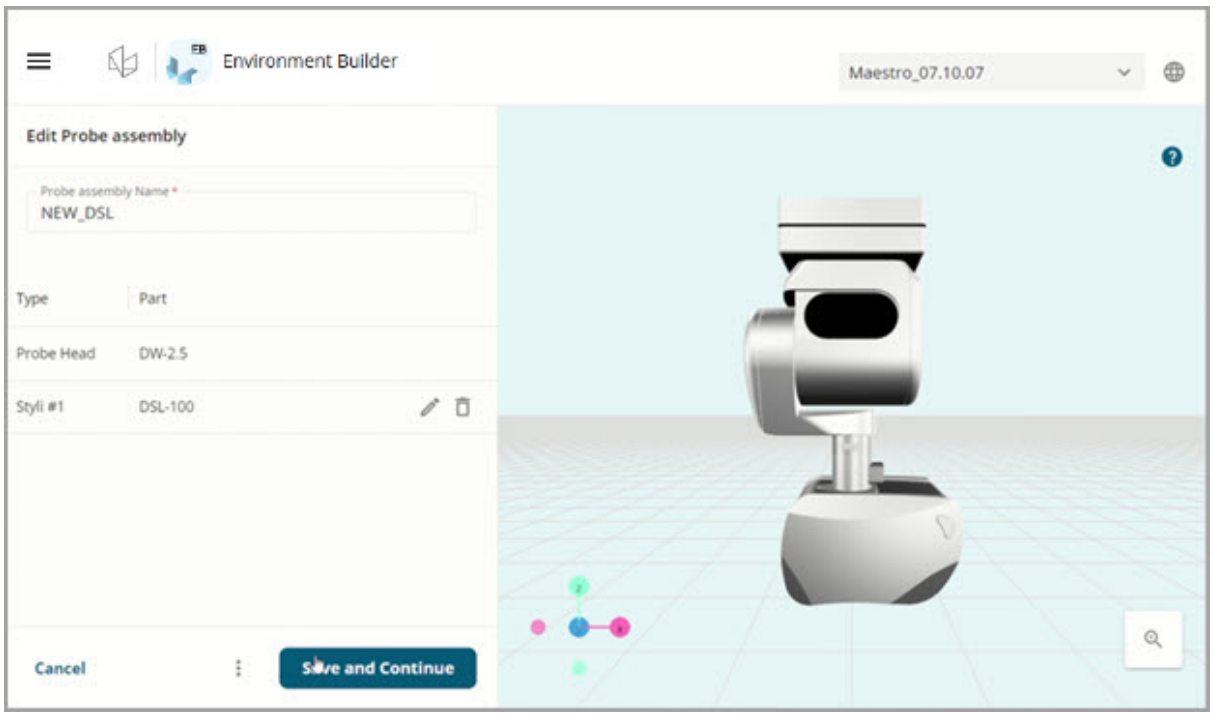
To make changes to any probe assembly component, you can use the associated **Edit** button ( ).

To remove any probe assembly component, you can use the associated **Delete** button ( ).

Be aware that you cannot save an incomplete probe assembly. When you delete a component from your assembly, you must replace it with another compatible component.

4. In the **Probe Assembly Name** box, type a unique name for your new probe assembly, and then do one of the following:
   - To save your changes and open the **Edit Angles** dialog box, select **Save and Continue**.
   - To save your changes and return to the **Probe Assemblies** page, select **More Options** ( ) and then select **Save and Close** .

You must type a unique name. Environment Builder prompts you to re-type the name if it detects that the probe assembly name is already in use.



5. Verify that the new probe assembly is available in the list of probe assemblies on the Environment Builder main window. This confirms that the creation of the new probe assembly was successful, and you can add it into your PC-DMIS measurement routines.

Tips for Efficiency

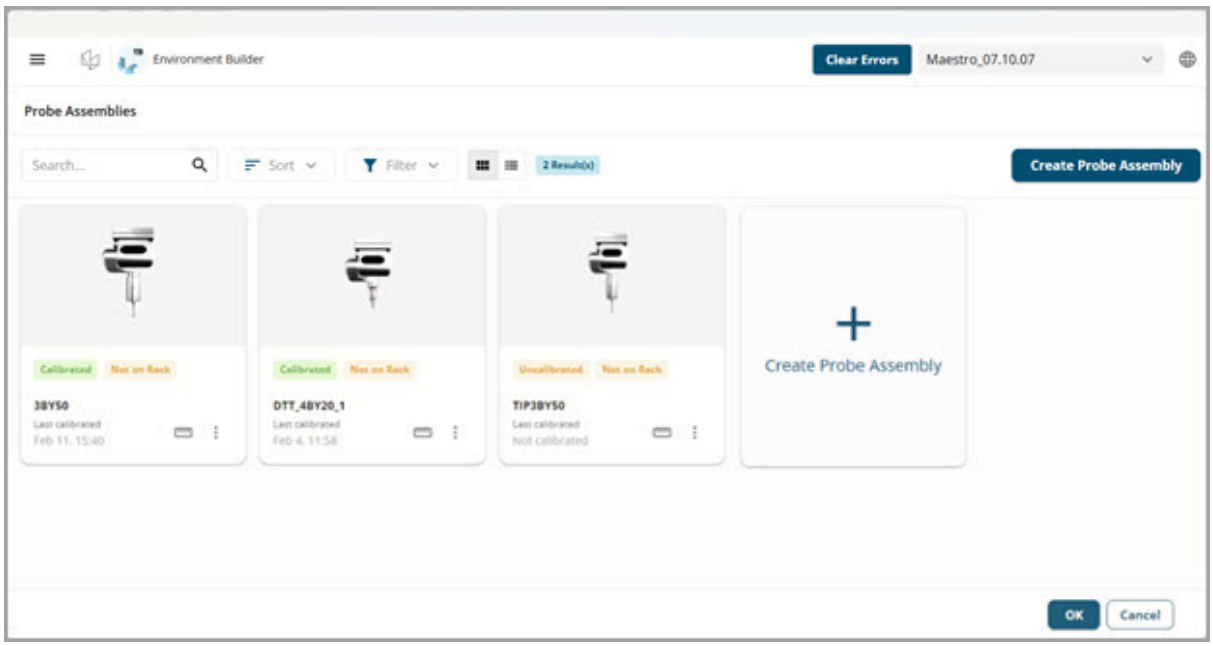
To speed up the probe assembly build process, familiarize yourself with the available components in Environment Builder.

Follow a naming convention to maintain consistency with identifiers for ease of retrieval.

Creating a DST Probe Assembly

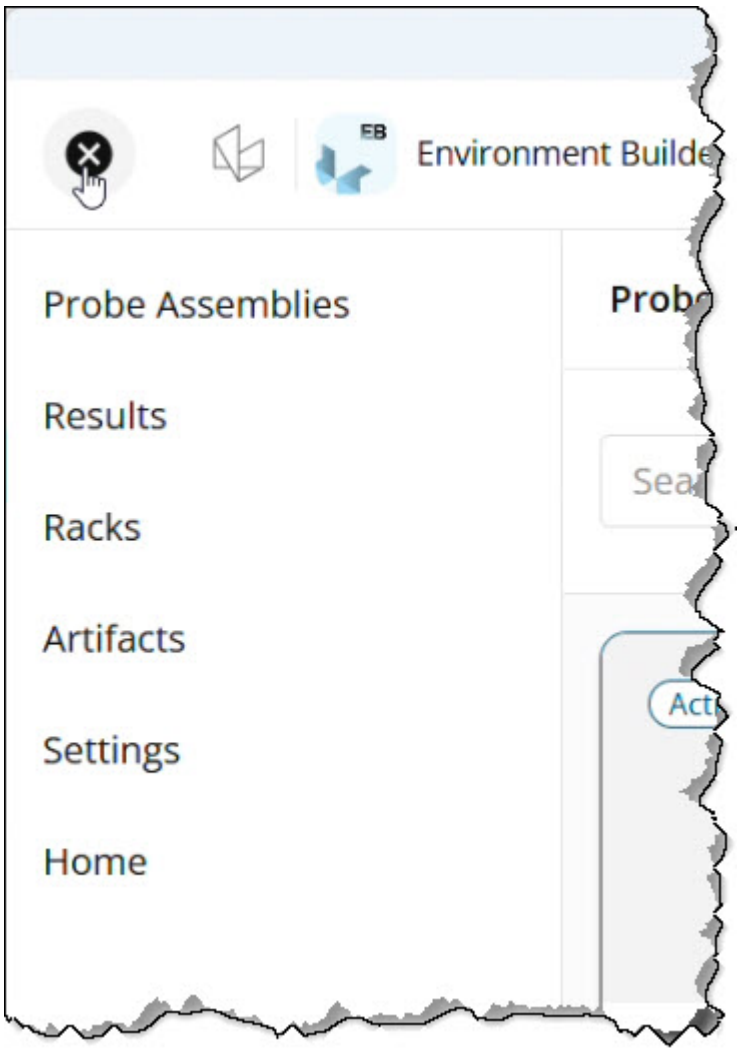
To create a Digital Scanning Tactile (DST) probe assembly with Environment Builder:

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

If you do not see the **Probe Assemblies** window, click the menu button ( ) located in the upper-left of the window to show the slide-out menu panel and select the **Probe Assemblies** menu option.

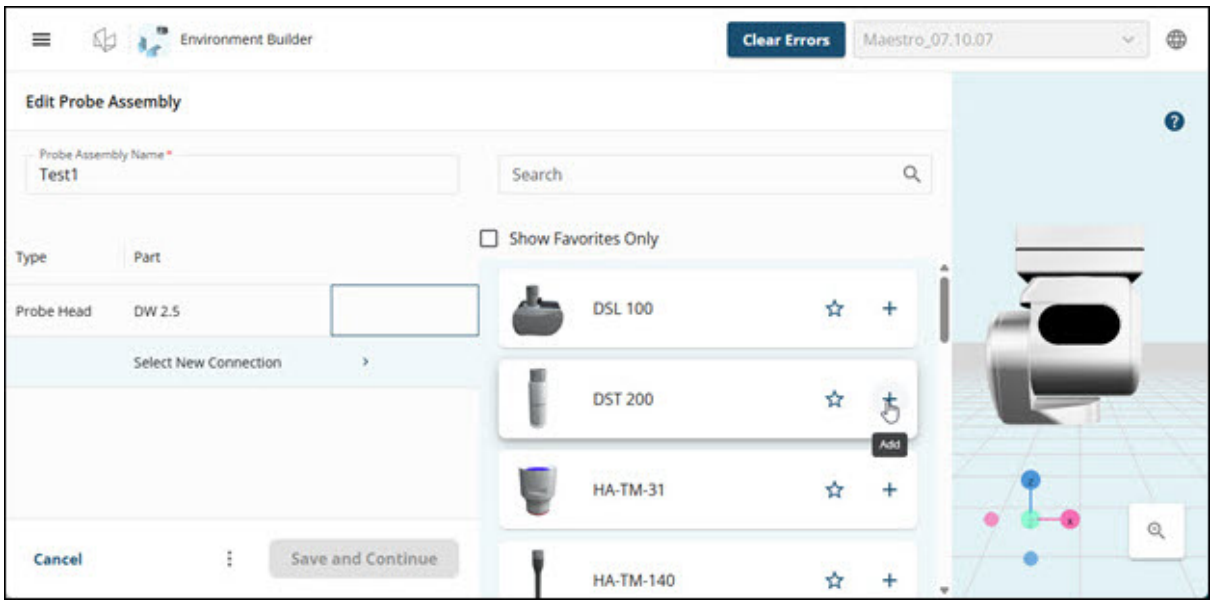


Click the **Close** button ( ) to hide the slide-out menu panel.

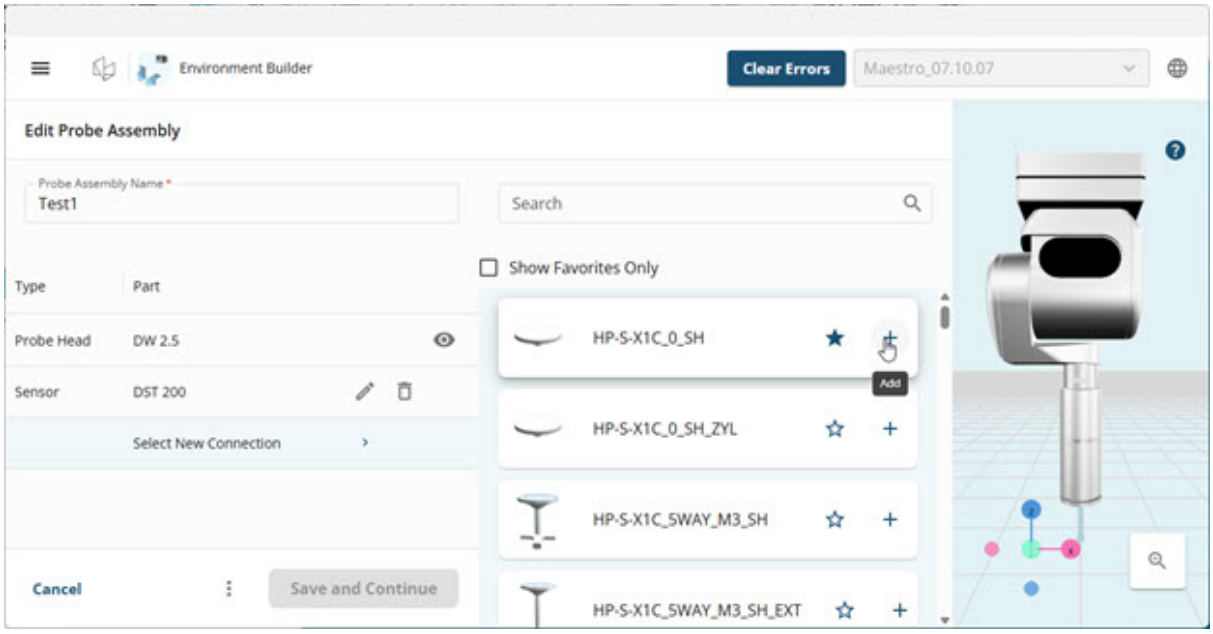
2. Select the **Create Probe Assembly** tile or button, and then confirm that the DW-2.5 probe head is selected for the machine configuration.

> As you select new components for your probe assembly, Environment Builder updates the probe component listing to only show components that are compatible with the last component you added.

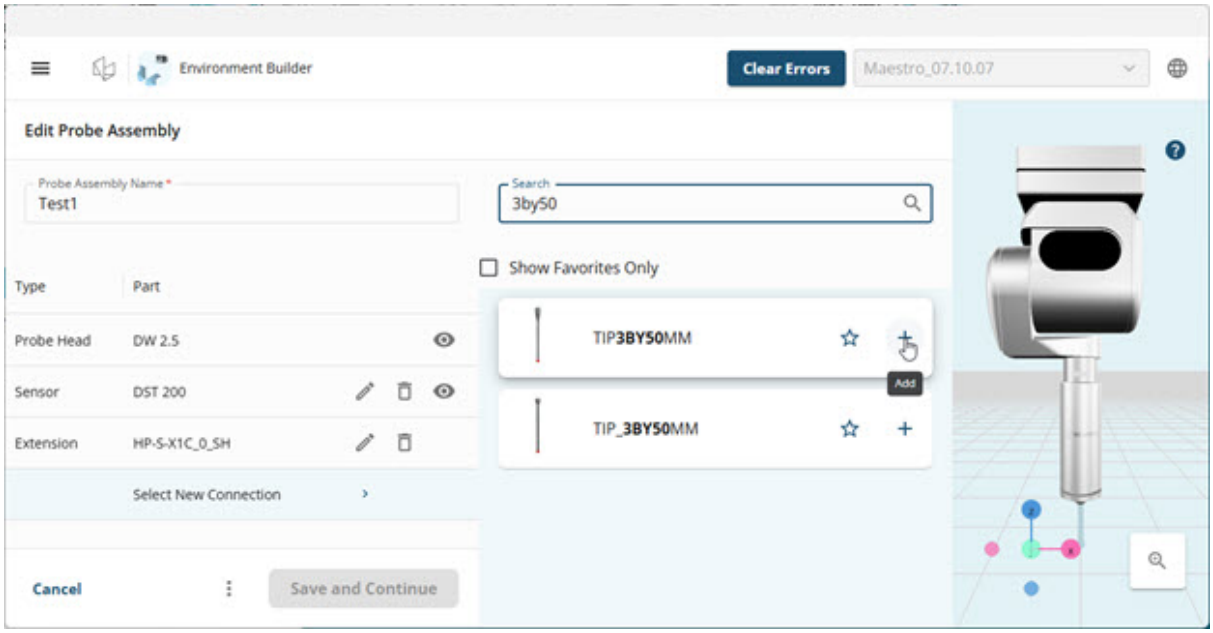
3. For this example, from the list of sensors compatible with the selected DW-2.5 probe head, select the DST-200 sensor.



4. The next component that you need to add to the probe assembly is the DST module. Select the HP-S-X1C_0_SH module from the list.



5. The final component that you need to add to the probe assembly is the stylus. Select the 3x50 stylus from the list.



Make sure it is the correct fit for your probe assembly.

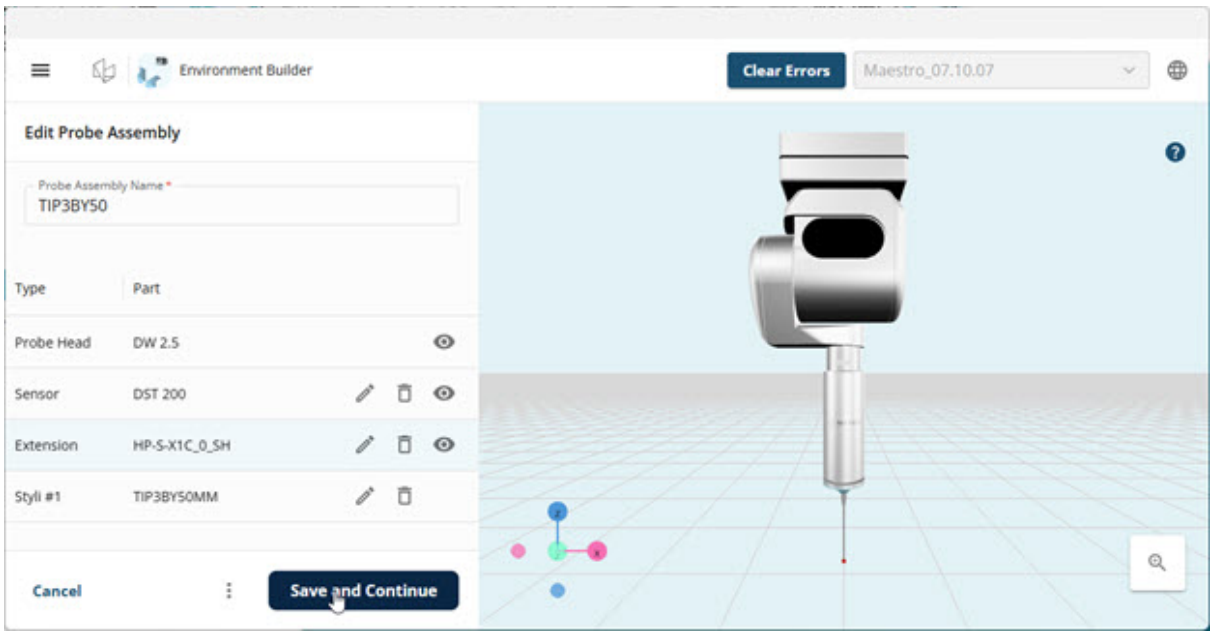
To make changes to any probe assembly component, you can use the associated **Edit** button ( ).

To remove any probe assembly component, you can use the associated **Delete** button ( ).

> Be aware that you cannot save an incomplete probe assembly. When you delete a component from your assembly, you must replace it with another compatible component.

6. In the **Probe Assembly Name** box, type a unique name for your new probe assembly, and then do one of the following:
   - To save your changes and open the **Edit Angles** dialog box, select **Save and Continue**.
   - To save your changes and return to the **Probe Assemblies** page, select **More Options** ( ) and then select **Save and Close** .

> You must type a unique name. Environment Builder prompts you to re-type the name if it detects that the probe assembly name is already in use.



7. Verify that the new probe assembly is available in the list of probe assemblies on the Environment Builder main window. This confirms that the creation of the new probe assembly was successful, and you can add it into your PC-DMIS measurement routines.

Tips for Efficiency

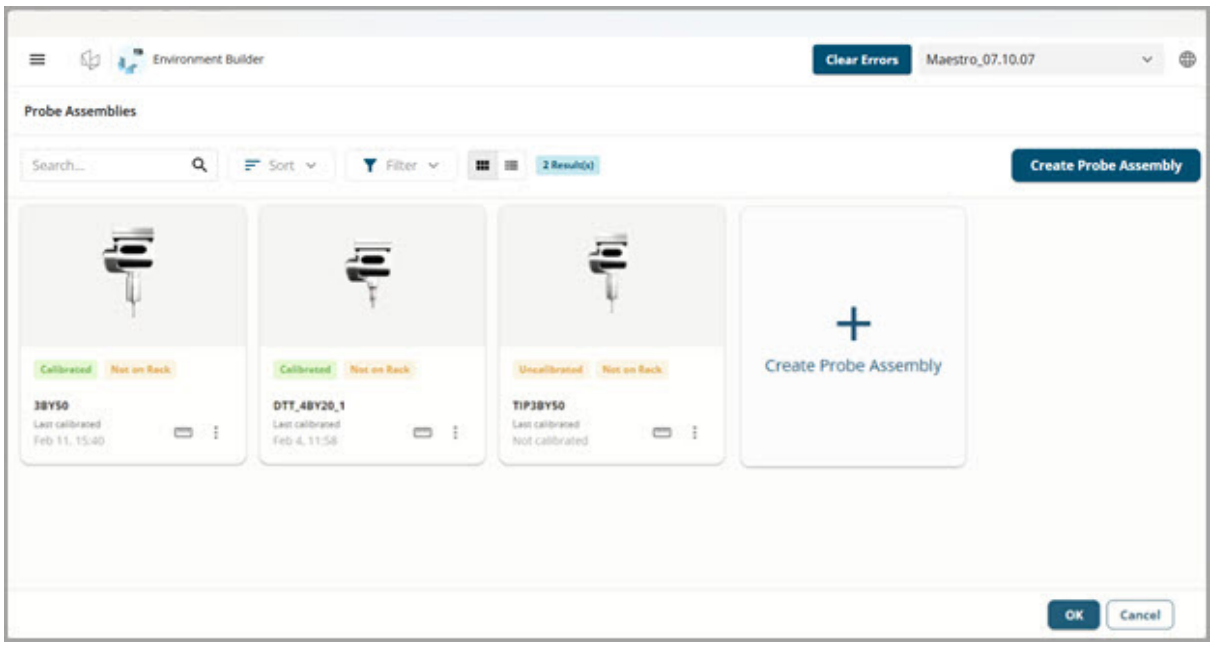
To speed up the probe assembly build process, familiarize yourself with the available components in Environment Builder.

Follow a naming convention to maintain consistency with identifiers for ease of retrieval.

Creating a DTT Probe Assembly

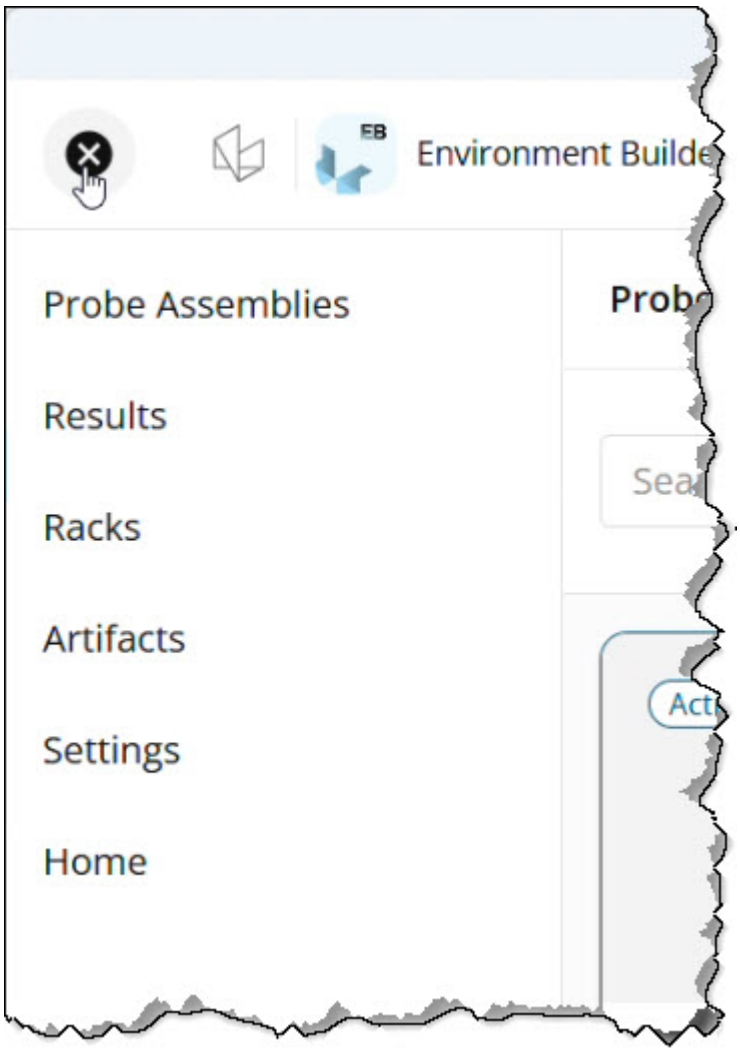
To create a Digital Touch Trigger (DTT) probe assembly with Environment Builder:

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

If you do not see the **Probe Assemblies** window, click the menu button ( ) located in the upper-left of the window to show the slide-out menu panel and select the **Probe Assemblies** menu option.

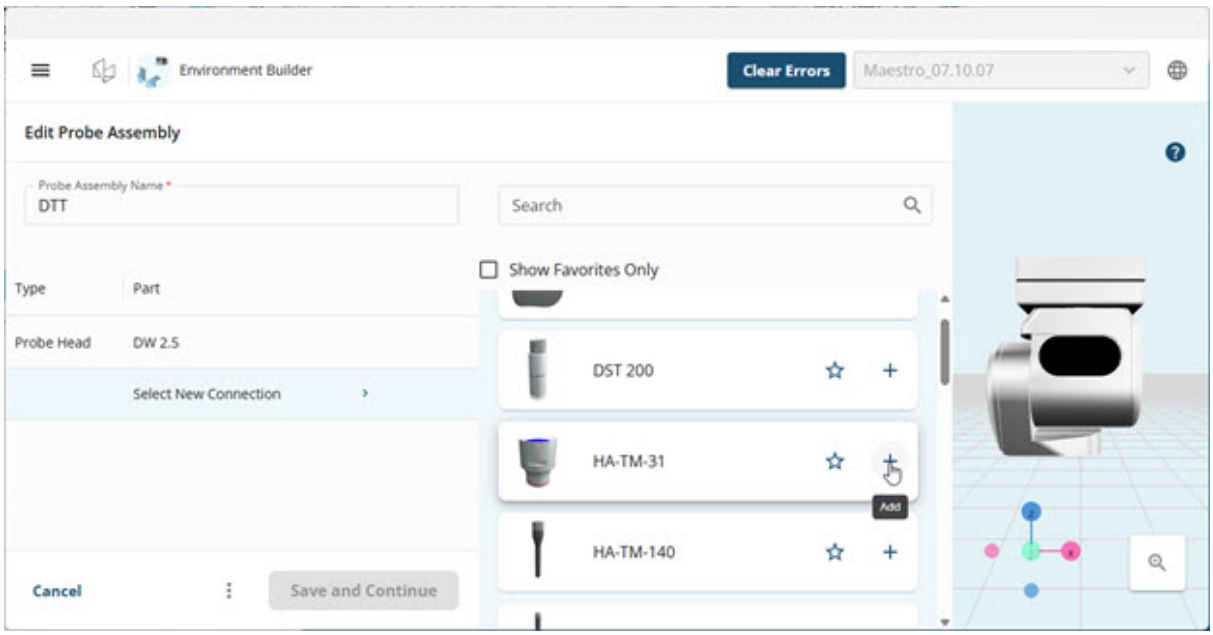


Click the **Close** button ( ) to hide the slide-out menu panel.

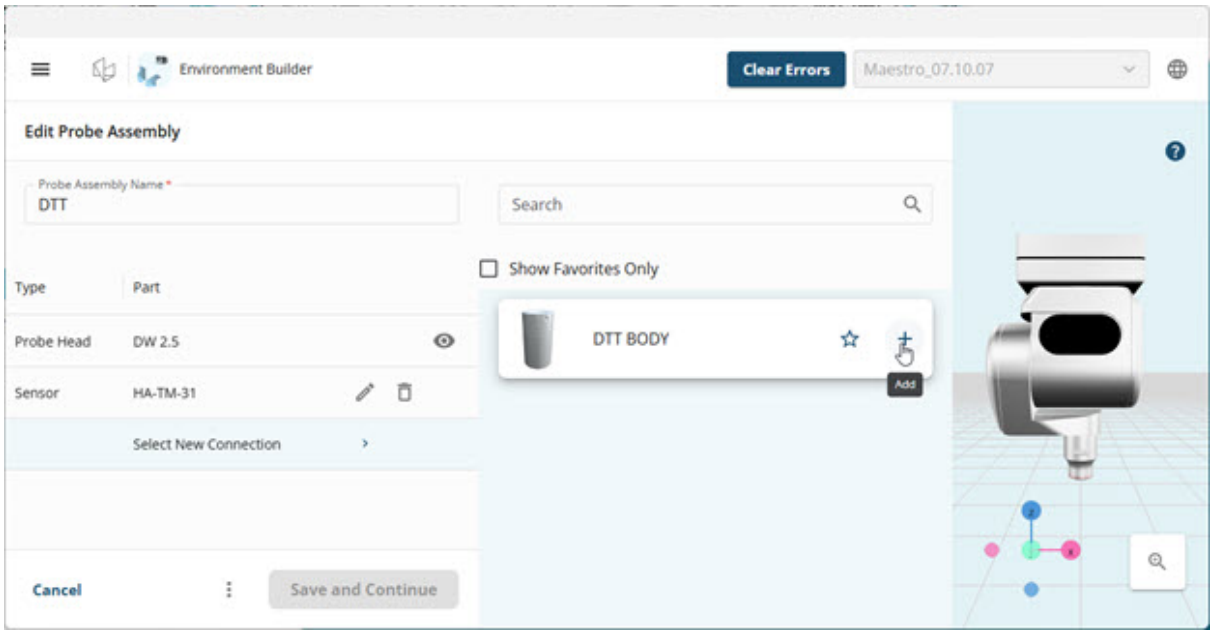
2. Select the **Create Probe Assembly** tile or button, and then confirm that the DW-2.5 probe head is selected for the machine configuration.

> As you select new components for your probe assembly, Environment Builder updates the probe component listing to only show components that are compatible with the last component you added.

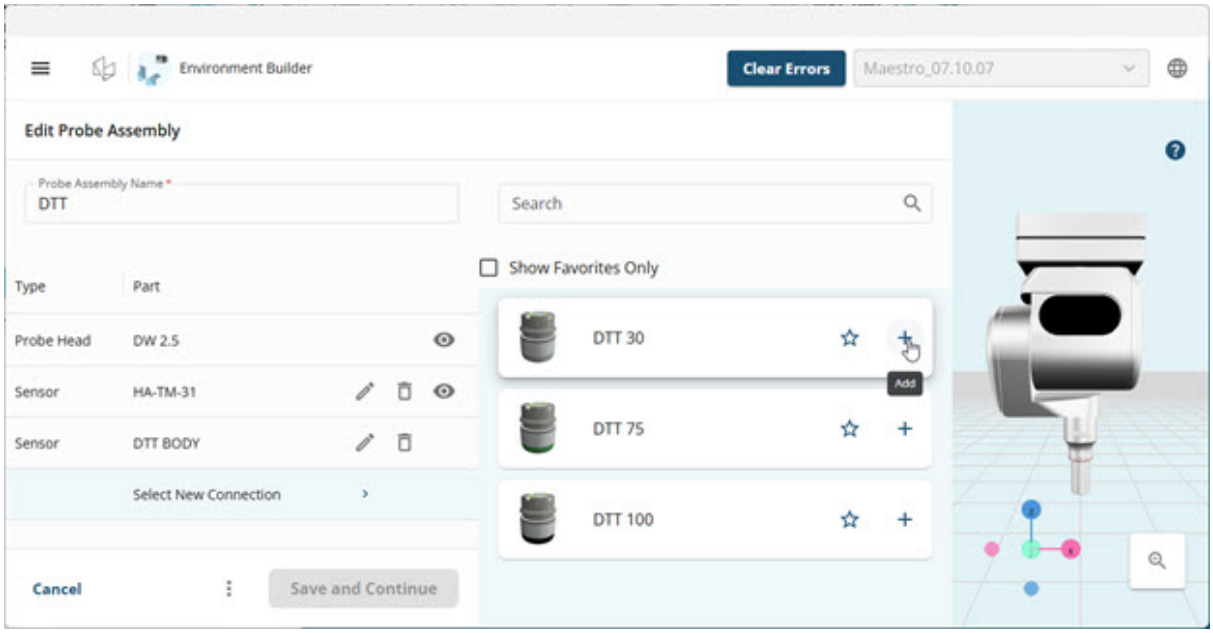
3. For this example, from the list of adapters compatible with the selected DW-2.5 probe head, select the HA-TM-31 adapter.



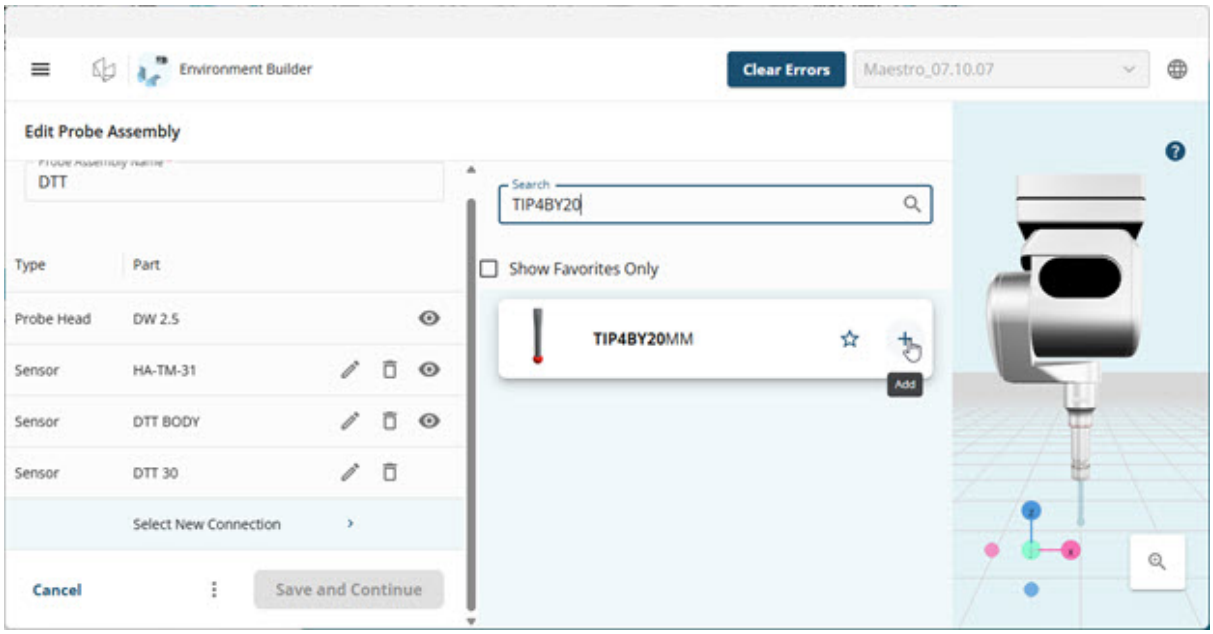
4. The next component that you need to add to the probe assembly is the digital touch trigger body. Select the DTT_BODY sensor option from the list.



5. The next component that you need to add to the probe assembly is the digital touch trigger module. Select the DTT-30 module option from the list.



6. Finally, select the 4x20 stylus option from the list.



Make sure it is the correct fit for your probe assembly.

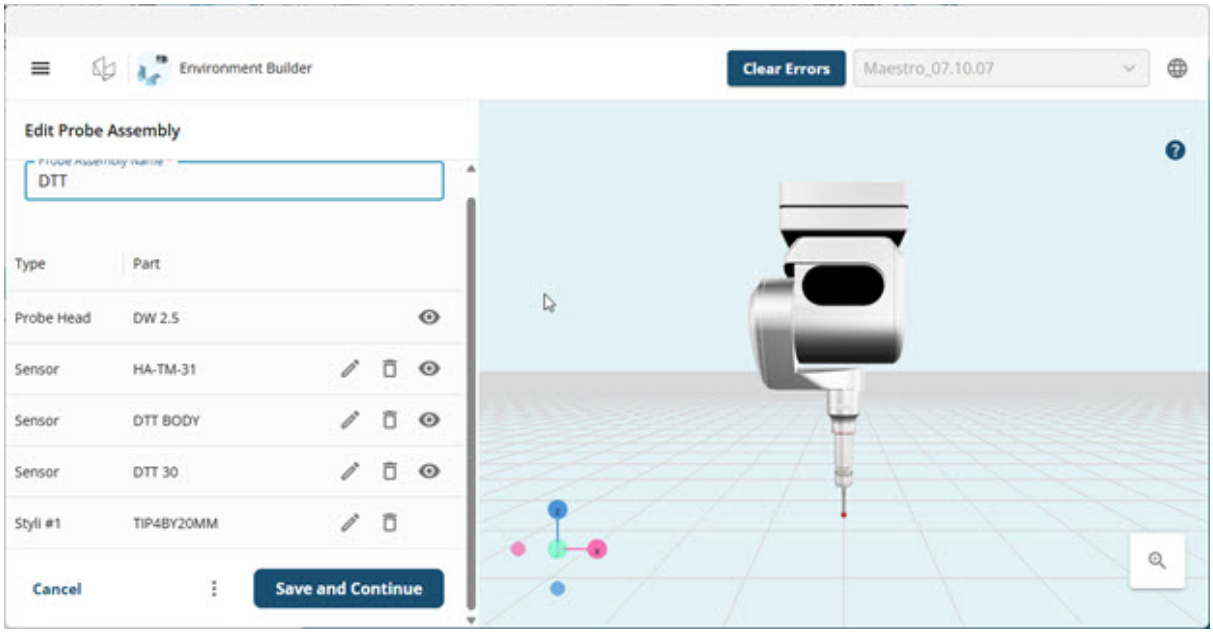
To make changes to any probe assembly component, you can use the associated **Edit** button ( ).

To remove any probe assembly component, you can use the associated **Delete** button ( ).

> Be aware that you cannot save an incomplete probe assembly. When you delete a component from your assembly, you must replace it with another compatible component.

7. In the **Probe Assembly Name** box, type a unique name for your new probe assembly, and then do one of the following:
   - To save your changes and open the **Edit Angles** dialog box, select **Save and Continue**.
   - To save your changes and return to the **Probe Assemblies** page, select **More Options** ( ) and then select **Save and Close** .

> You must type a unique name. Environment Builder prompts you to re-type the name if it detects that the probe assembly name is already in use.



8. Verify that the new probe assembly is available in the list of probe assemblies on the Environment Builder main window. This confirms that the creation of the new probe assembly was successful, and you can add it into your PC-DMIS measurement routines.

Tips for Efficiency

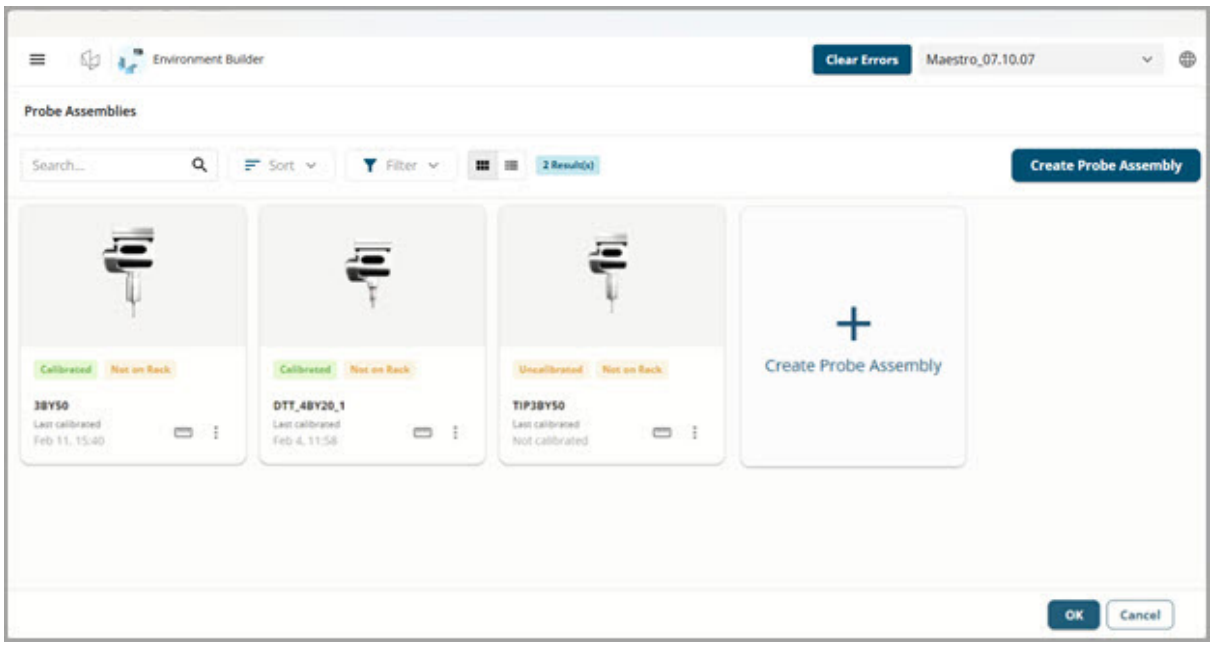
To speed up the probe assembly build process, familiarize yourself with the available components in Environment Builder.

Follow a naming convention to maintain consistency with identifiers for ease of retrieval.

Creating a Star Probe Assembly

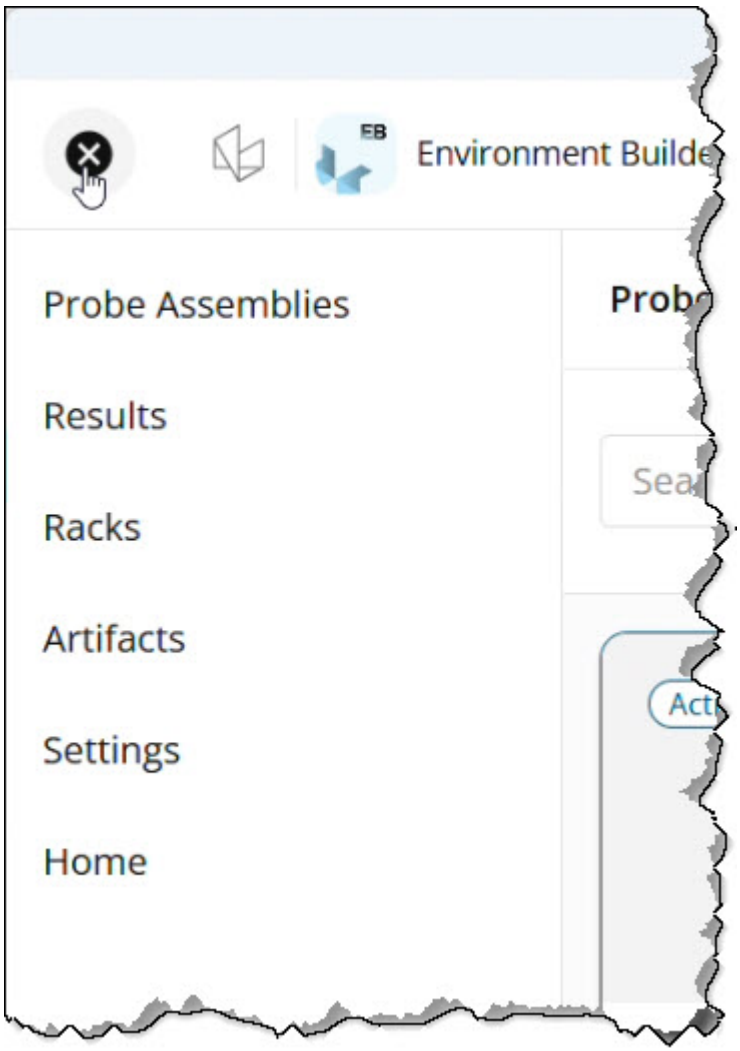
To create a Star probe assembly with Environment Builder:

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

If you do not see the **Probe Assemblies** window, click the menu button ( ) located in the upper-left of the window to show the slide-out menu panel and select the **Probe Assemblies** menu option.

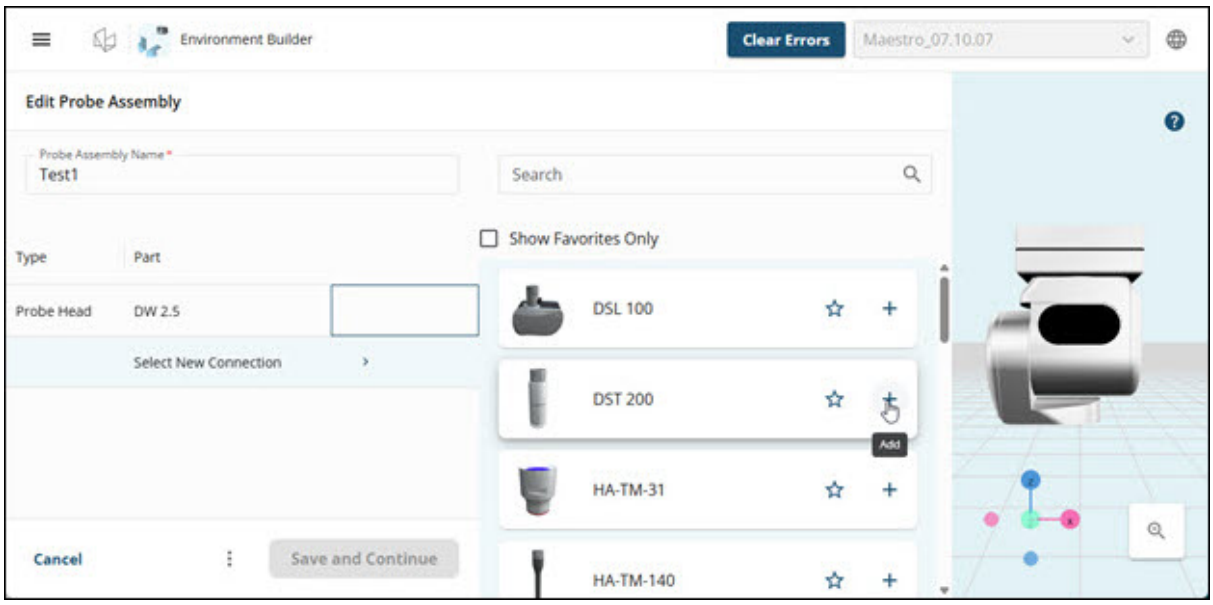


Click the **Close** button ( ) to hide the slide-out menu panel.

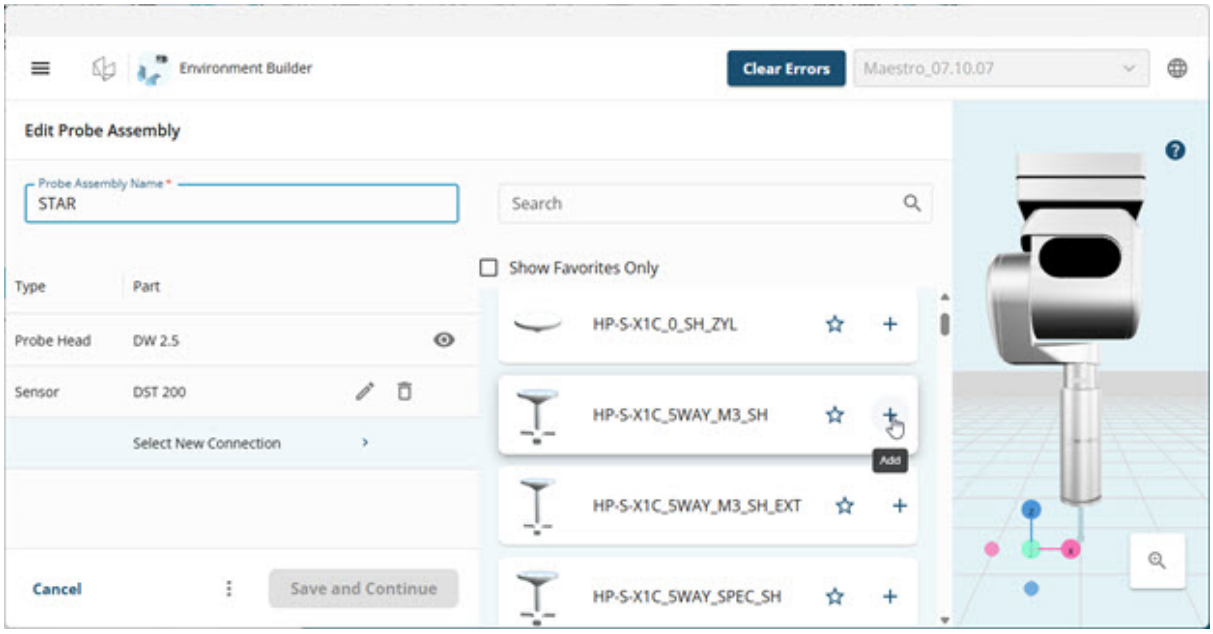
2. Select the **Create Probe Assembly** tile or button, and then confirm that the DW-2.5 probe head is selected for the machine configuration.

> As you select new components for your probe assembly, Environment Builder updates the probe component listing to only show components that are compatible with the last component you added.

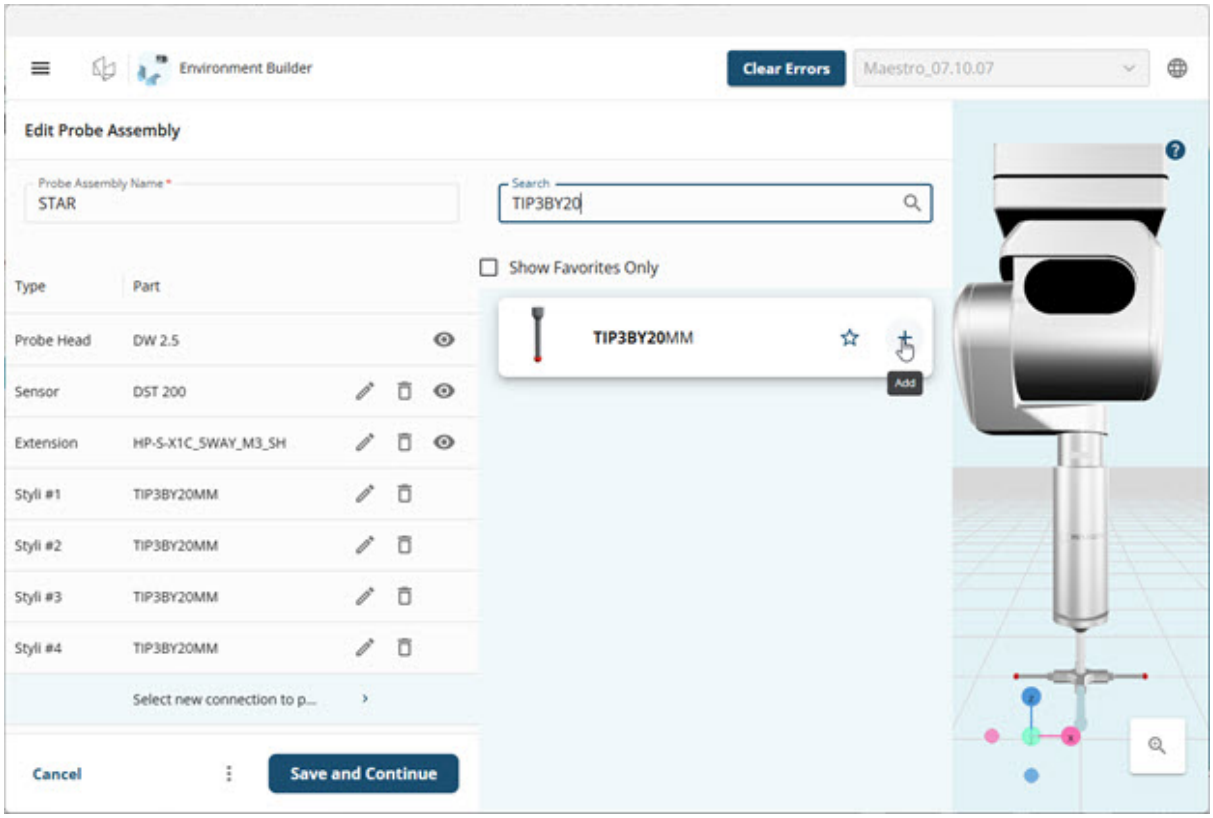
3. For this example, from the list of sensors compatible with the selected DW-2.5 probe head, select the DST-200 sensor.



4. The next component that you need to add to the probe assembly is the HP-S-X1C_5WAY_M3_SH module. From the list of components, select that module.



There are five ports on the module that you can outfit with compatible styli. Environment Builder automatically selects the first port which is the bottom port. After you add the stylus to Port 1, Environment Builder moves in a counterclockwise rotation to each of the remaining ports, starting with Port 2 in the X+ direction.



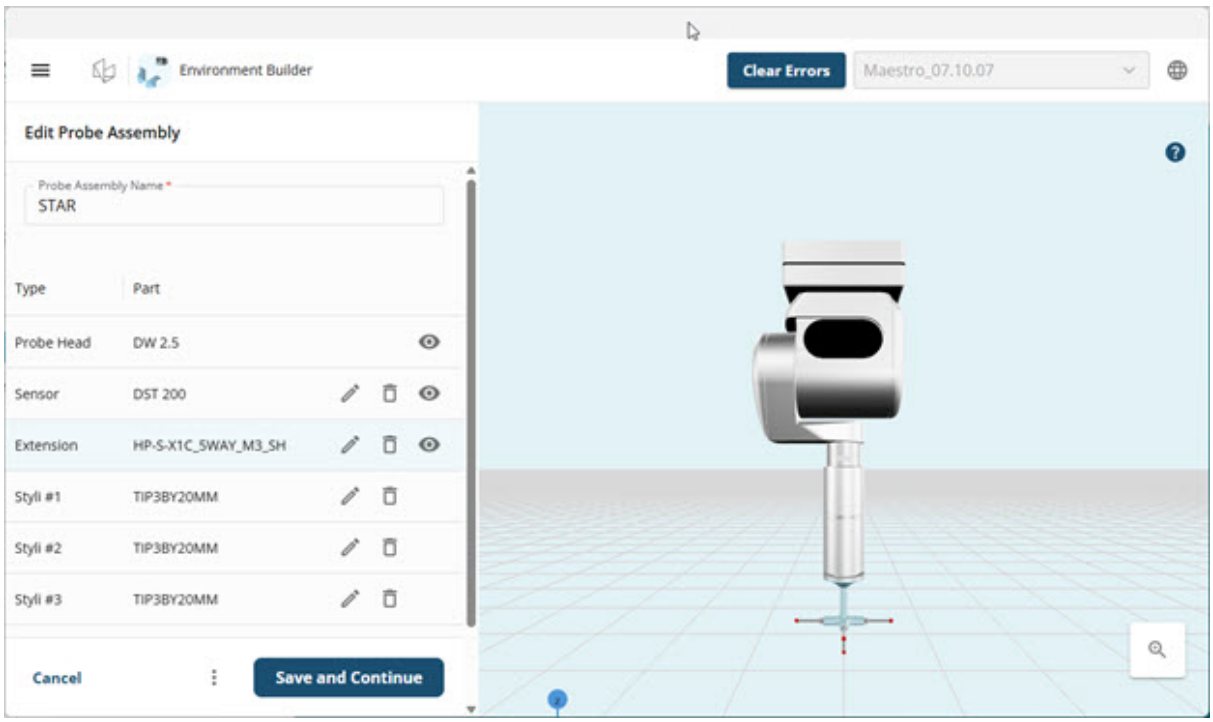
To make changes to any probe assembly component, you can use the associated **Edit** button ( ).

To remove any probe assembly component, you can use the associated **Delete** button ( ).

> Be aware that you cannot save an incomplete probe assembly. When you delete a component from your assembly, you must replace it with another compatible component.

5. In the **Probe Assembly Name** box, type a unique name for your new probe assembly, and then do one of the following:
   - To save your changes and open the **Edit Angles** dialog box, select **Save and Continue**.
   - To save your changes and return to the **Probe Assemblies** page, select **More Options** ( ) and then select **Save and Close** .

> You must type a unique name. Environment Builder prompts you to re-type the name if it detects that the probe assembly name is already in use.



6. Verify that the new probe assembly is available in the list of probe assemblies on the Environment Builder main window. This confirms that the creation of the new probe assembly was successful, and you can add it into your PC-DMIS measurement routines.

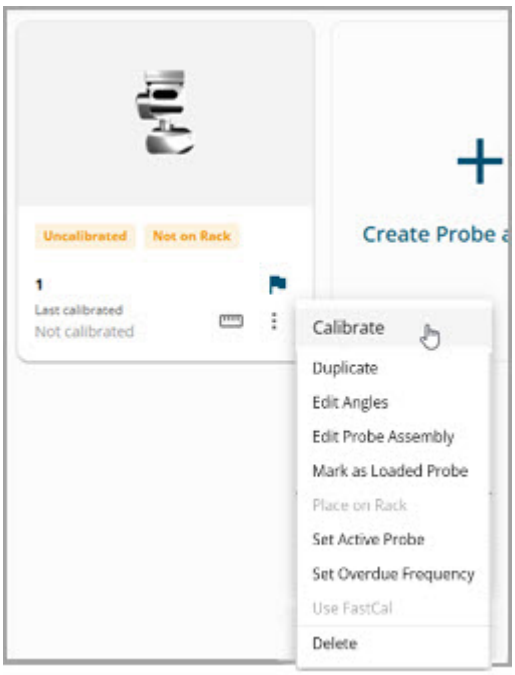
Tips for Efficiency

To speed up the probe assembly build process, familiarize yourself with the available components in Environment Builder.

Follow a naming convention to maintain consistency with identifiers for ease of retrieval.

Probe Assembly Options

To open a probe assembly's menu, select **More Options** ( ) in the lower-right corner of its tile. The menu contains the following options:



- **Edit Probe Assembly** – Change the components in the assembly.
- **Edit Angles** – Add, edit, or remove angles for the assembly. For information on adding angles to probe assemblies, see "Adding Angles to Probe Assemblies in Environment Builder".
- **Calibrate** – Calibrate the assembly. For information on calibrating a probe assembly, see "Probe Assembly Calibration Procedure".
- **View Calibration Result** – Display the results of the last calibration.
- **Place on Rack** – Assign the assembly, or part of it, to a port on a probe changer. For information on placing probe assemblies on a probe changer, see "Placing Probe Assemblies on a Probe Changer".
- **Set Active Probe** – Set the probe assembly that Environment Builder returns to PC-DMIS when you close it.
- **Mark as Loaded Probe** – Specify which probe assembly is physically mounted on the machine.
- **Use FastCal** / **Unset FastCal** – Turn FastCal on or off for the assembly. While FastCal is active, the angle options are unavailable and Environment Builder displays the message "These options are unavailable while FastCal is active."
- **Set Overdue Frequency** – Set the duration, in days or hours, after which this assembly's calibration becomes overdue, independent of the interval set in the Environment Builder settings.
- **Duplicate** – Create a copy of the assembly that you can modify.

- **Delete** – Remove the assembly. If one or more artifacts use the assembly as their reference probe, Environment Builder warns you that deleting the assembly might affect dependent operations.

In the read-only user role, **Edit Probe Assembly** and **Edit Angles** become **View Probe Assembly** and **View Angles**, respectively. Options that modify the environment are unavailable. For information on the read-only user role and offline mode, see "Working in a Read-Only Role or Offline".

Showing and Hiding Assembly Components

While editing a probe assembly, you can hide individual components to focus on the parts of the assembly you are working on. In the probe assembly component list, select the **Show/Hide** button next to a component to show or hide it.

Environment Builder saves the visibility settings with the probe assembly, so the same components stay hidden the next time you open it.

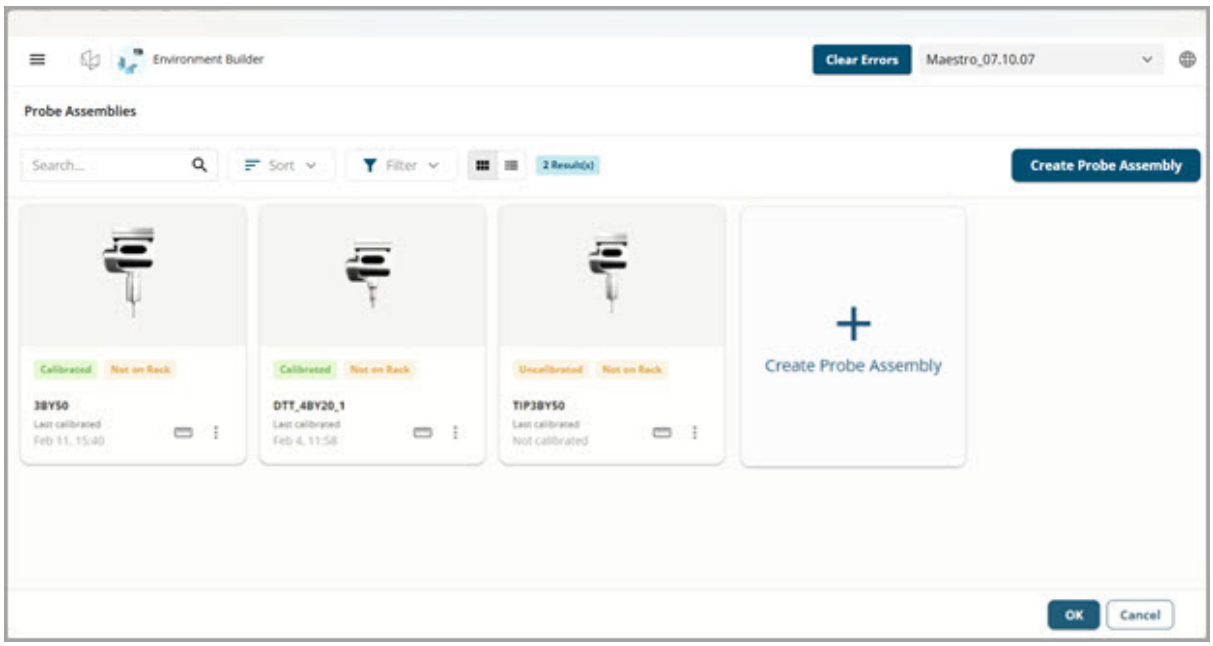
Calibrating Probe Assemblies

This section provides you with the workflow that you can use to calibrate your probe assembly in Environment Builder.

Adding Calibration Artifacts in Environment Builder

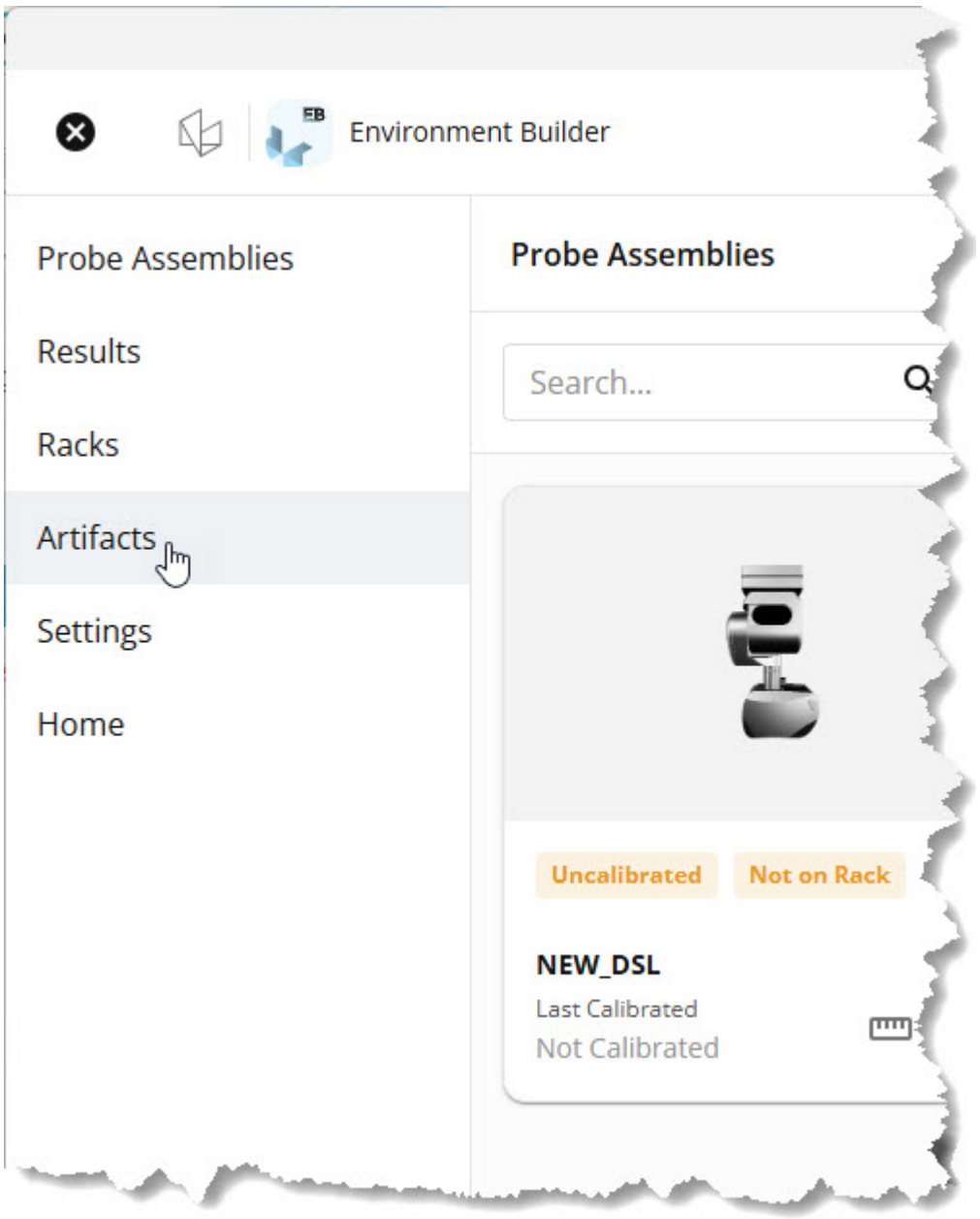
Follow this procedure to add and configure new calibration artifacts in Environment Builder that you can use to calibrate your probe assemblies.

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.

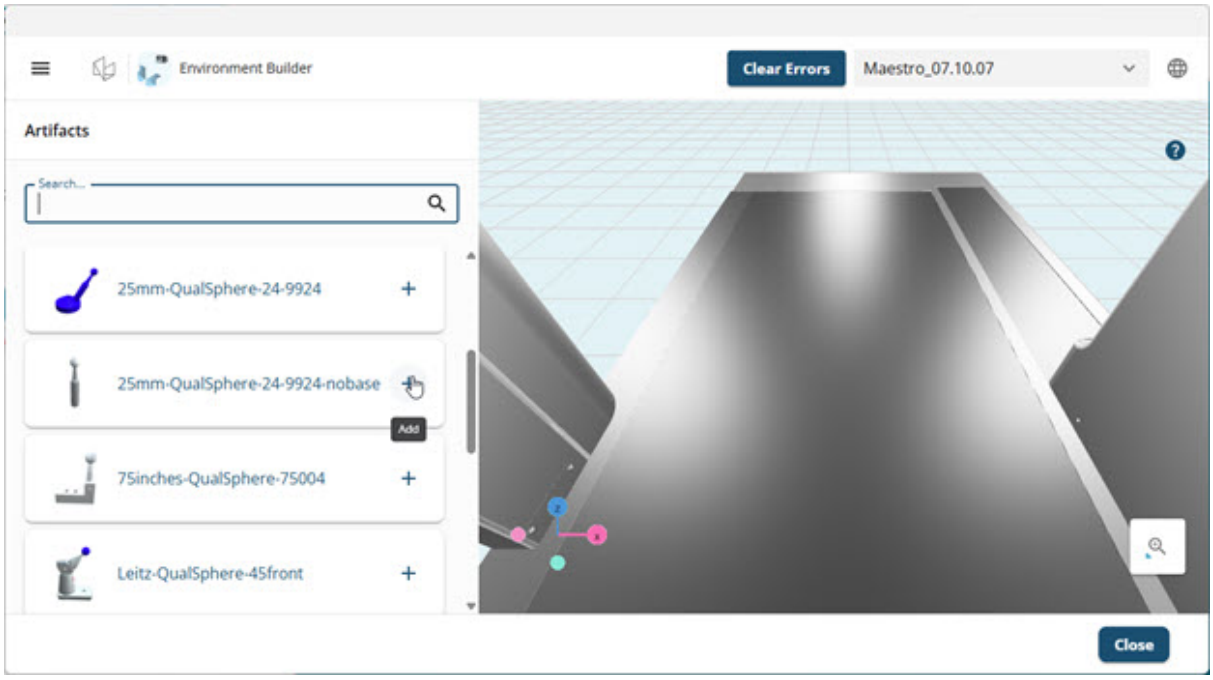


To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

2. Select the menu button ( ) in the upper-left corner of the window to show the slide-out menu panel and select the **Artifacts** menu option.

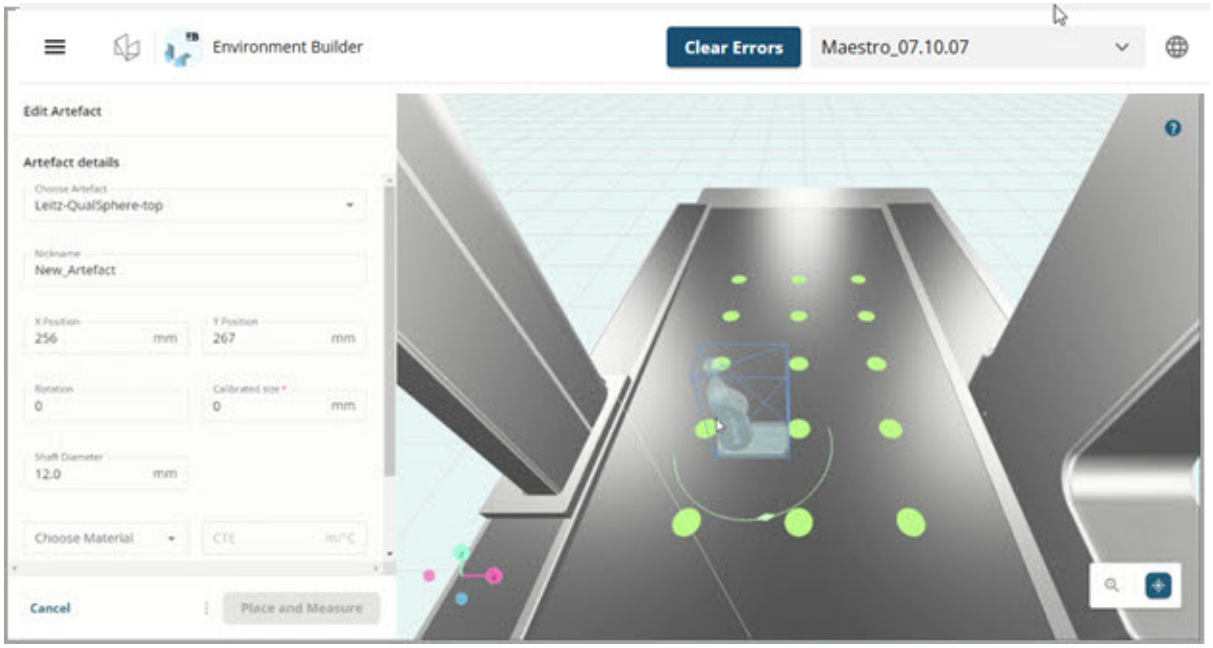


3. Scroll through the list of artifacts and select the one that matches your CMM.



4. Select the **+** button on the right side of the artifact tile to add the artifact to the CMM table and open the **Edit Artifact** dialog box.

5. In the **Nickname** box, type a new name to replace the factory default nickname for easier identification in the future.



6. On the CMM table in the 3D scene, drag and drop the artifact on top of a green dot to define its rough position.

7. Click and drag the green arc near the artifact to define its rough rotation.

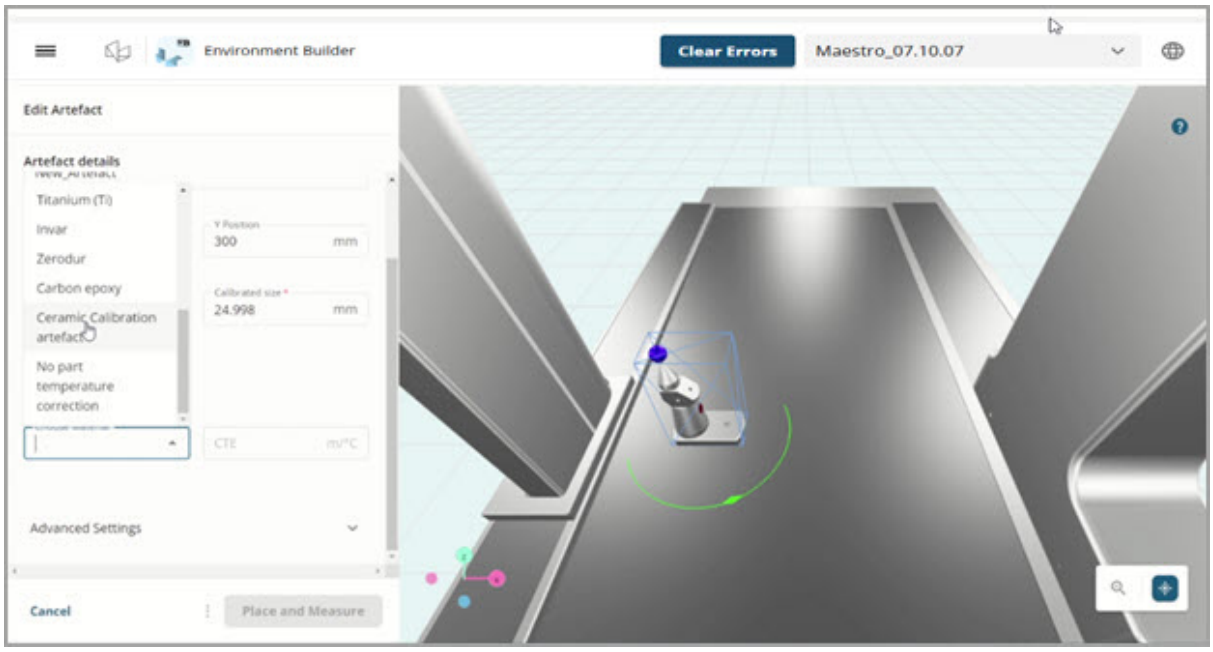
8. You can use the **Edit Artifact** dialog box to refine the artifact's position. To do this, type the value into the **X Position** and **Y Position** boxes (or click in each box and

use the Up and Down arrow keys on your keyboard to increment or decrement the current value).

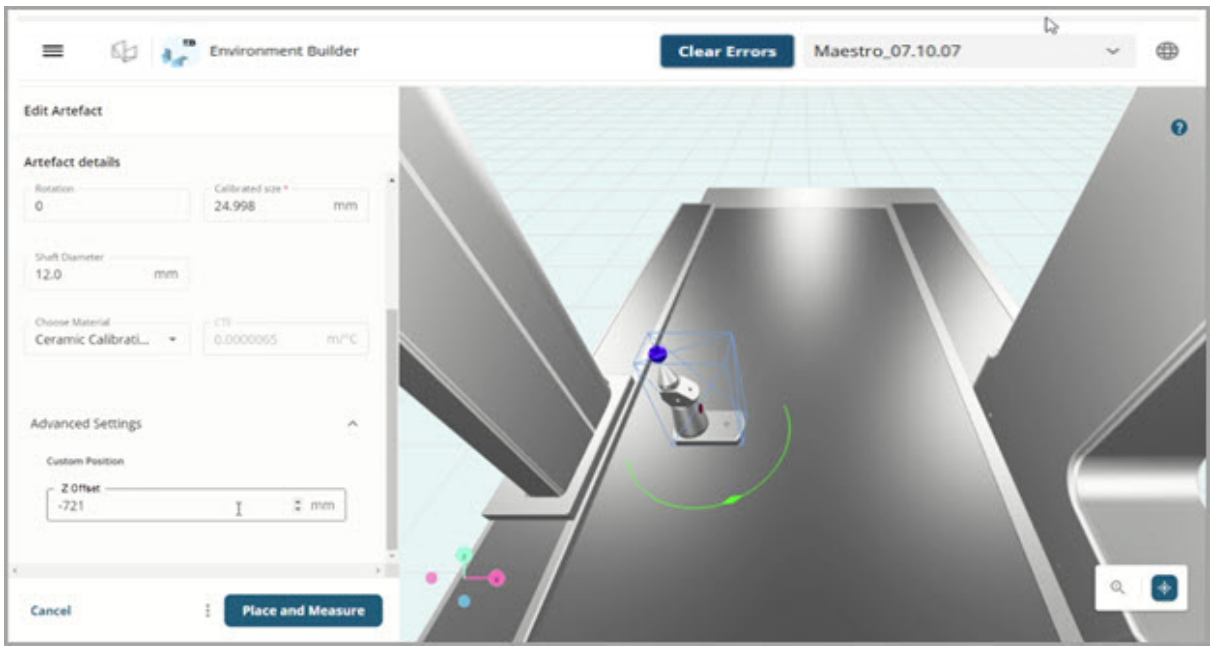
9. From the **Edit Artifact** dialog box, you can define a finer rotation. Type the artifact's angle into the **Rotation** box.

10. Environment Builder inserts a default artifact size value. Type the calibrated size of your artifact into the **Calibrated Size** box. It is stamped somewhere on your artifact's body.

11. From the **Choose Material** list, select the artifact's material type.



12. To define the Z Rise value, in the **Advanced Settings** section of the **Edit Artifact** dialog box, type it into the **Z Rise** box (or use the up and down arrows in the box). This is necessary if the artifact is mounted on a table fixture or has an extension.



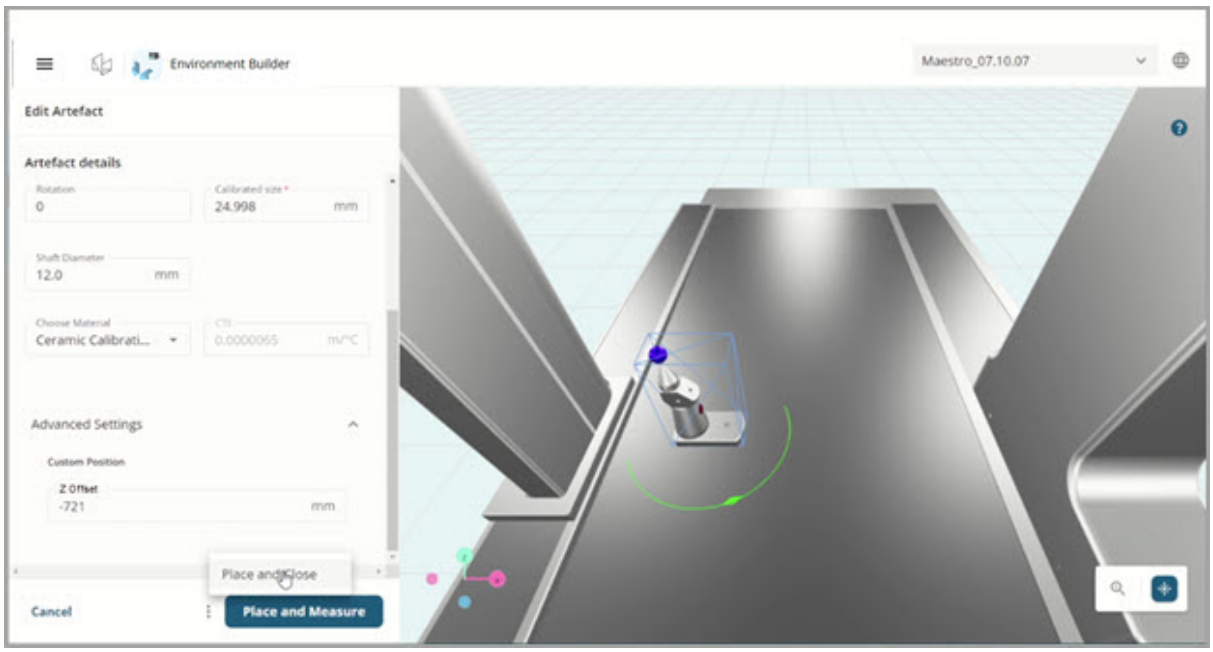
Make sure that all setup options match your physical CMM configuration.

13. Do one of the following:

    - To place the artifact on the virtual CMM for use in probe calibrations, select **Place and Measure**.
    - To save your changes and open the **Artifacts** page, select **More Options** ( ), and then select **Place and Close**.

When you edit an artifact that is already placed, these buttons read **Update and Measure** and **Update and Close**.

> **!** Make sure that all artifact configuration settings are correct before you use it in your measurement routines.



To avoid calibration errors, make sure that the artifact size and material match the artifact's physical specifications.

For accurate probe readings, make sure that you position the artifact correctly.

Tips for Efficiency

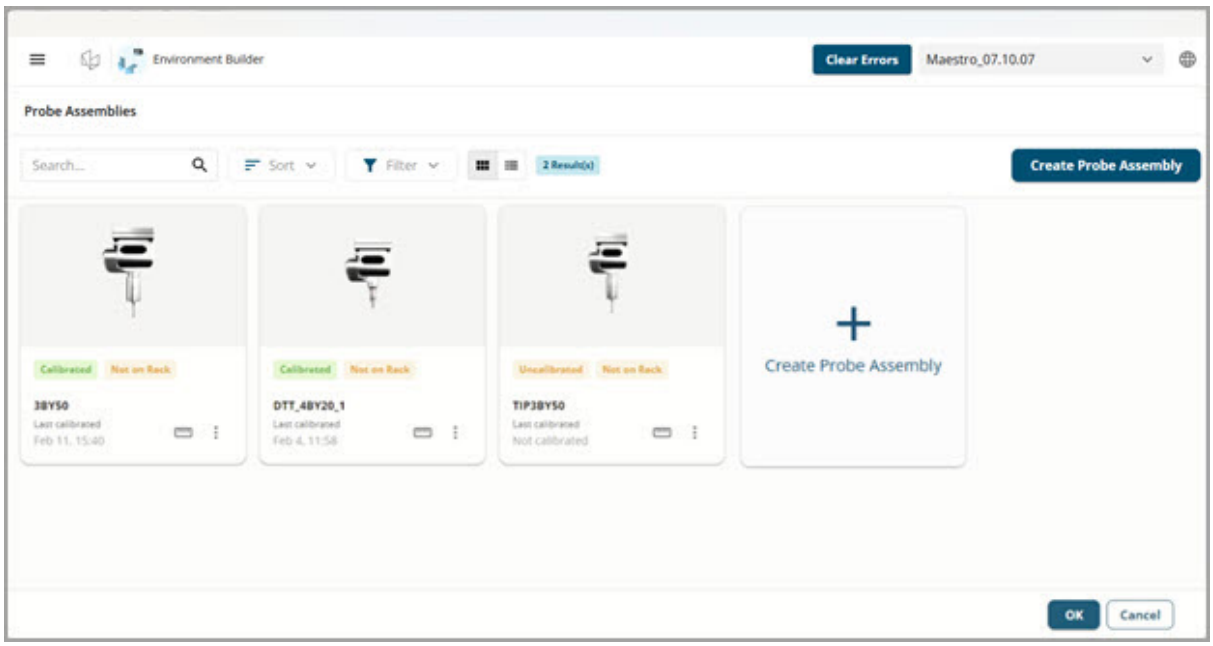
To speed up the setup and calibration processes, familiarize yourself with the Environment Builder's interface.

To reduce future setup time, create a template for common artifacts.

Adding Angles to Probe Assemblies in Environment Builder

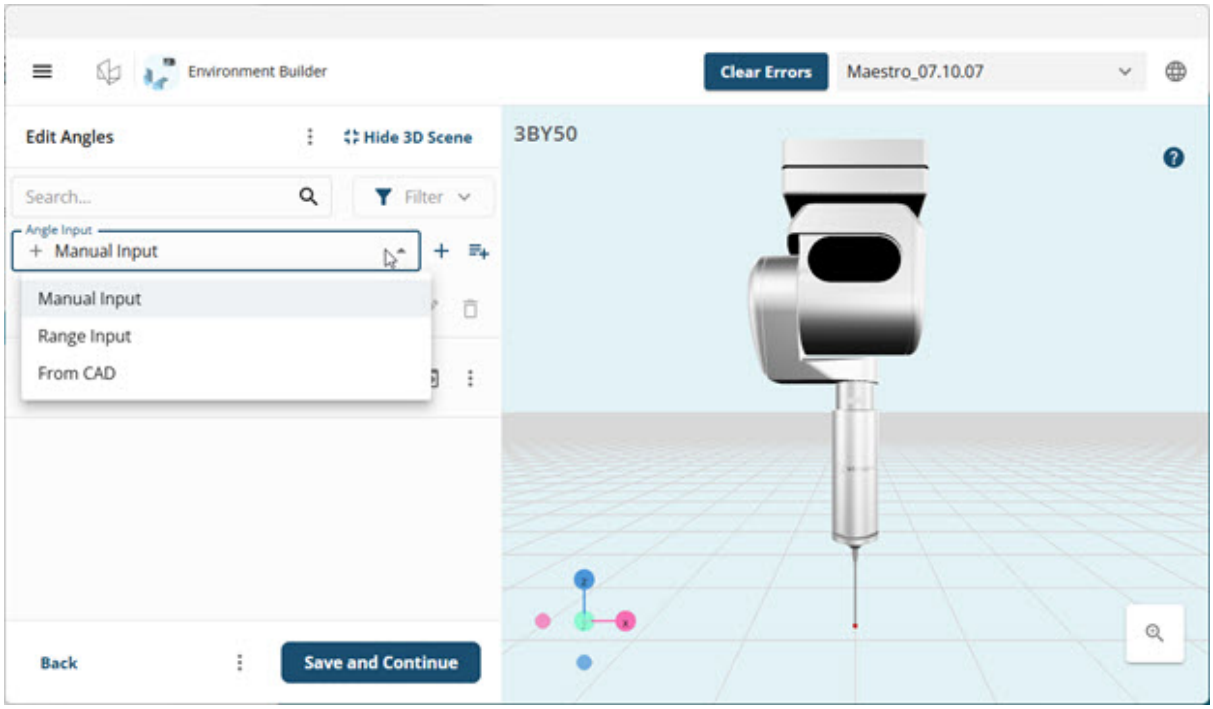
Follow this procedure to add and manage angles in existing Environment Builder probe assemblies.

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

2. In the **Probe Assemblies** page, on the tile for the probe assembly that you want to add an angle to, select **More Options** ( ) and then select **Edit Angles** to open the **Edit Angles** dialog box.

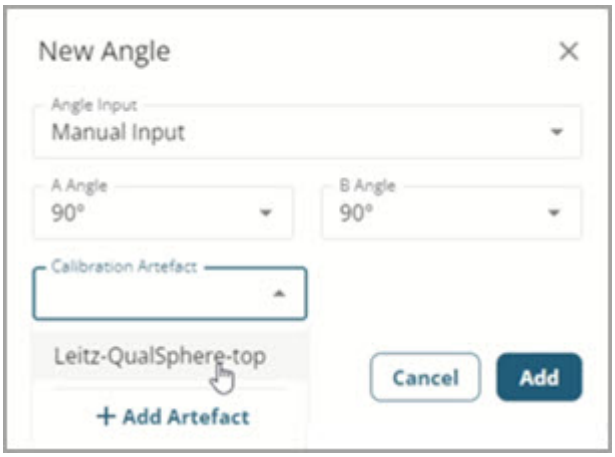


3. In the **Edit Angles** dialog box, add angles using one of these input methods:

If the input method is already selected, select the **+** (**New Angle**) button beside the **Angle Input** list to open the **New Angle** dialog box.

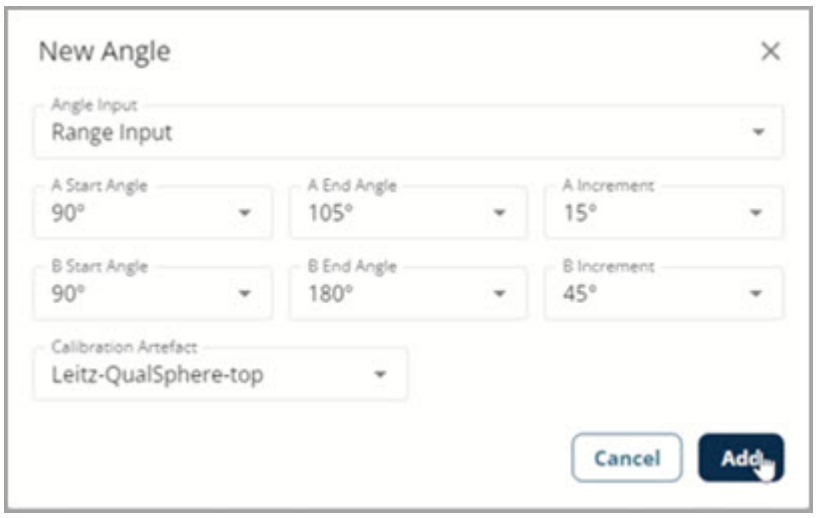
- **Manual Input**

  1. From the **Angle Input** list, select **Manual Input** to open the **New Angle** dialog box.
  2. In the **A Angle** and **B Angle** boxes, type or select the angle values.
  3. From the **Calibration Artifact** list, select the artifact to use, and then select **Add**.



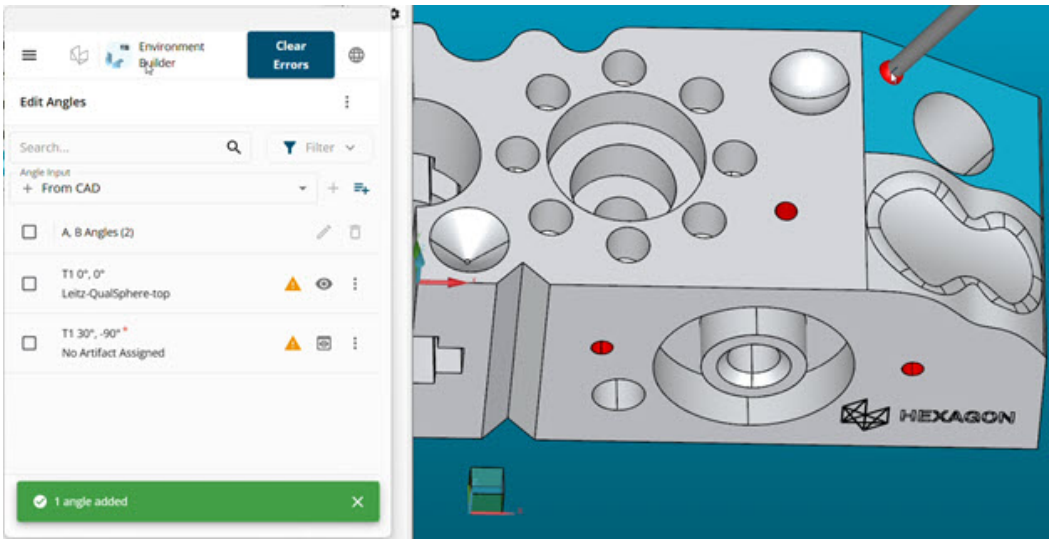
- **Range Input**

  1. From the **Angle Input** list, select **Range Input** to open the **New Angle** dialog box.
  2. In the **A Start Angle**, **A End Angle**, and **A Increment** boxes, type or select the values to add a range of angles.
  3. In the **B Start Angle**, **B End Angle**, and **B Increment** boxes, type or select the values to add a range of angles.
  4. From the **Calibration Artifact** list, select the artifact to use, and then select **Add**.



- **From CAD**

  1. From the **Angle Input** list, select **From CAD**, and then in the 3D scene, select the angle (or angles) directly on the CAD model. Environment Builder adds each selected angle to the list. Unlike **Manual Input** and **Range Input**, an angle that you add from CAD is created without a calibration artifact. It is marked with an asterisk and shows **No Artifact Assigned**.

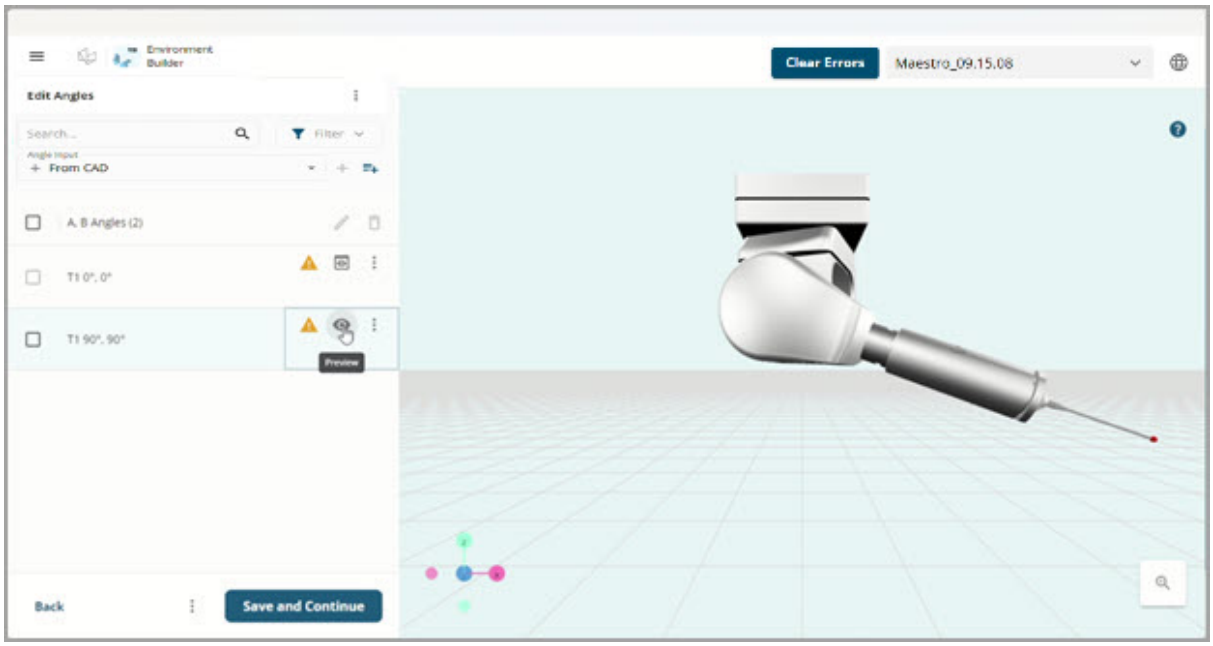


  2. To assign a calibration artifact, select the angle (or select the **A, B Angles** group), select **Edit** ( ), and then, in the **Select Artifact** dialog box, select the **Calibration Artifact** and select **Update**. Environment Builder assigns the artifact to the selected angles.

You must assign a calibration artifact to every angle before you can select **Save and Continue** or **Save and Close**.

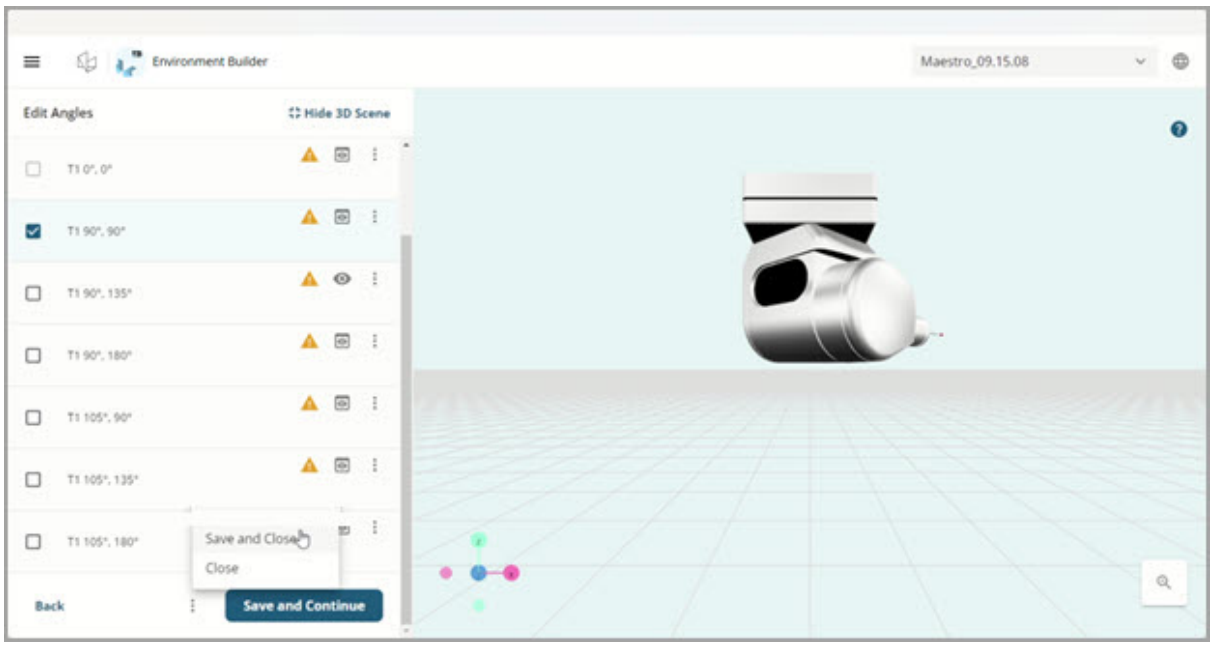
- **Add and select angles from routine** - Select this option to add angles from measurement routines.
  - **This Routine** - Search the current measurement routine for AB angles.
  - **All Routines** - Search for tips used in other measurement routines by the currently-active probe file.

4. To preview an angle, in the **Edit Angles** dialog box, in the row for the angle, select the **Preview** button ( ) to view a simulation of the probe assembly at that angle.



> Be aware of **Caution** icons ( ) next to the tips. These indicate uncalibrated tips. You must calibrate tips before you can use them in your measurement routines. For details, follow the calibration procedures described in the "Calibrating Probe Assemblies" section of the PC-DMIS documentation.

5. Select **More Options** ( ) to show the **Edit**, **Edit Artifact**, and **Delete** options.
6. Review the added angles in the list, and then do one of the following:
   - To save your changes and start calibrating the probe assembly, select the **Save and Continue** button.
   - To save your changes and return to the **Probe Assemblies** page, select **More Options** ( ), and then select **Save and Close**.



To avoid calibration errors, make sure that the angles are calibrated correctly.

Be cautious when you edit or remove angles, as this may cause the angle values to reset to their original, default values.

Tips for Efficiency

Preview angles regularly to confirm their positions before you finalize them.

Selecting a Qualification Algorithm

When you start a calibration, Environment Builder asks you to select an algorithm from the **Select Qualification Algorithm** list. The available algorithms depend on the sensor or styli in your probe assembly:

- **Ball Stylus** – The available algorithms are **Tactile TRAX Qualification**, **Tactile Dynamic Qualification**, and **FastCal Qualification**.
- **Cylinder Stylus** – The available algorithm is **Cylinder Qualification**.
- **Disc Stylus** – The available algorithm is **Disc Qualification**.
- **Line Laser** – The available algorithm is **Line Laser Qualification**.

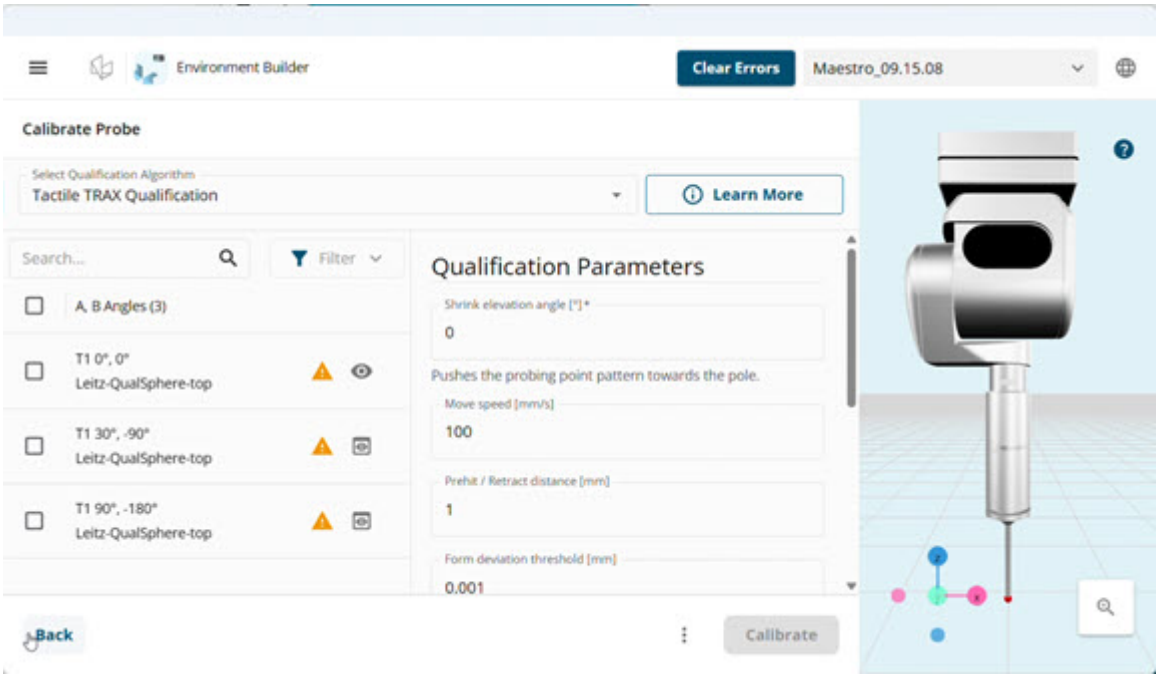
To view detailed information about an algorithm, select **Learn More** next to it.

You can also enable FastCal for a probe assembly without starting a calibration. On the assembly's tile, select **More Options** ( ) and then select **Use FastCal**. Environment Builder displays the confirmation message "Fast Calibration set successfully." To disable FastCal, select **Unset FastCal**. While FastCal is active for an assembly, you cannot edit its angles.

Defining Calibration Parameters

After you select an algorithm, Environment Builder displays the **Parameters** panel, where you select the angles to calibrate and define how the calibration runs. The parameter groups that Environment Builder displays depend on the algorithm you selected.

Use the **Load or create a parameter set** list to reuse a set of parameters. Select **Save Parameter Set** to save the current values with a name. Environment Builder displays the confirmation message "Parameter set <name> saved successfully." Select **Default Parameters** to restore the factory values. If you previously calibrated the assembly, Environment Builder restores your previous selections and displays the message "Angles selection and qualification parameters have been restored from the previous calibration."



The parameter groups are:

- **General Information** – Number of Hits, Prehit/Retract, Move Speed, and Touch Speed.
- **Type of Operation** – Calibrate Tips, Calibrate the Unit, Qualification Check, Home the Unit, and Calibrate ScanRDV.
- **Shank** – Shank Qual, Number of Shank Hits, and Shank Offset.

- **Calibration Mode** – Default Mode or User Defined. When you select User Defined, set the Number of Levels, Start Angle, End Angle, and Touch Speed.
- **Wrist Calibration** – Axis Start, Axis End, and Increments for each wrist axis, as well as Create New Maps or Replace Closest Map, and Tool Mounted on Rotary Table.
- **Fast Calibration** – User Cal Check, Recalibrate, and Reload Factory Map. Environment Builder displays this group only when you select the FastCal algorithm.
- **Tips to use if none are explicitly selected** – All or Used in Routine, as well as Abort Execution and Reset tips to Theo at start of calibration.
- **Calibration Order** – The order in which Environment Builder calibrates the selected angles, from top to bottom.

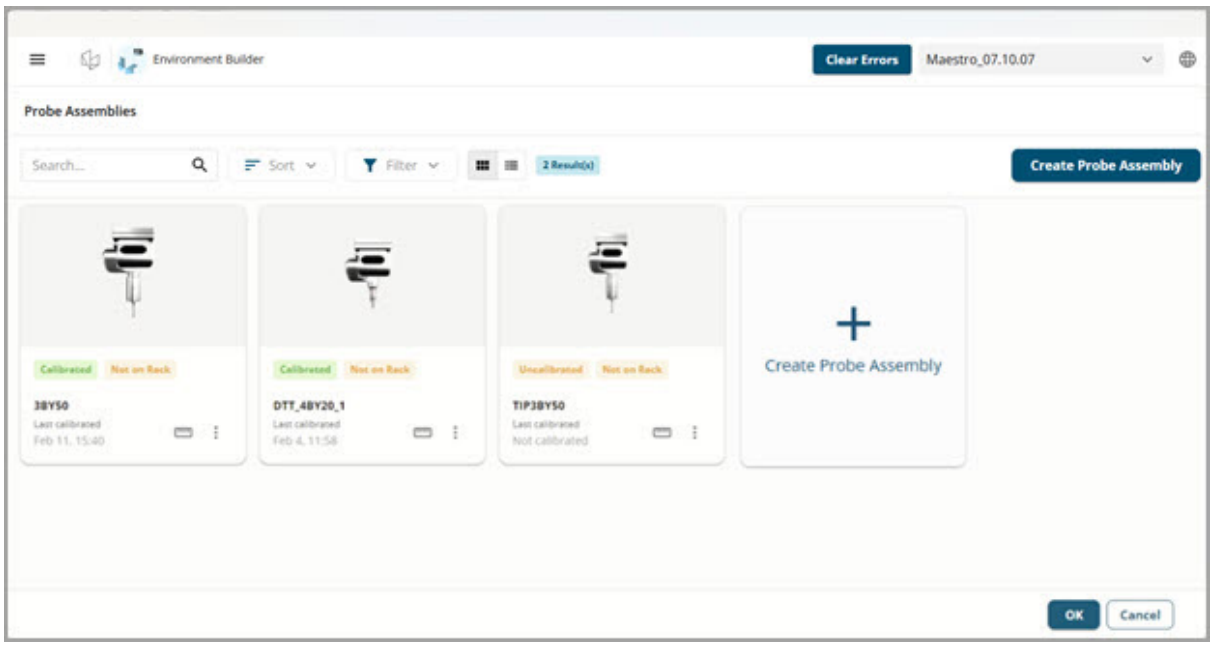
Calibration Order matters when the probe path between two angles could cause a collision. Select **User defined** and drag the angles into the order you want. Environment Builder preserves that order and does not sort the angles automatically. If you have not selected any angles, the group displays "No angles selected."

When you qualify star styli, be aware of possible collisions with the sphere support, depending on the wrist orientation and stylus length. To prevent collisions, adjust the elevation angle or the sphere orientation.

Probe Assembly Calibration Procedure

Follow this procedure to calibrate a probe assembly.

1. Open the Environment Builder main window. You can see a list of probe assemblies created in Environment Builder.

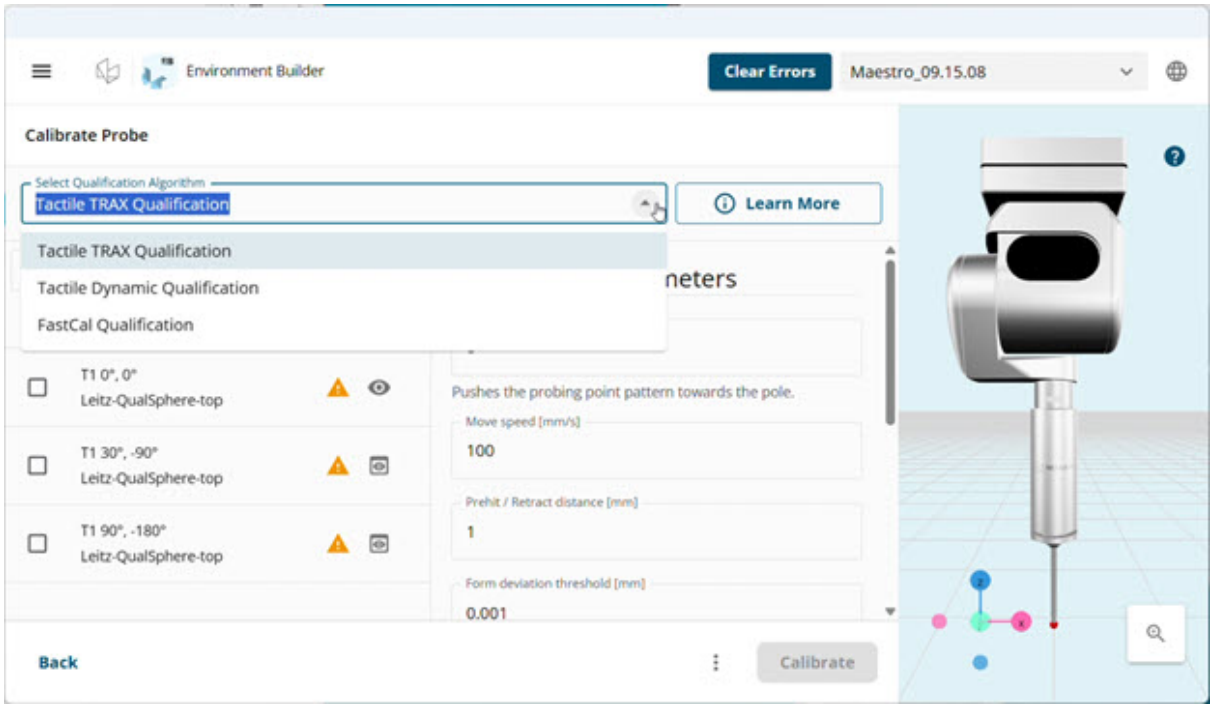


To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

- Probe assemblies that you have successfully calibrated have a green "Calibrated" message in their tiles.
- Probe assemblies that you have not yet calibrated show an orange "Uncalibrated" message in their tiles.

For information on the different calibration states, see "Managing the Probe Assemblies List".

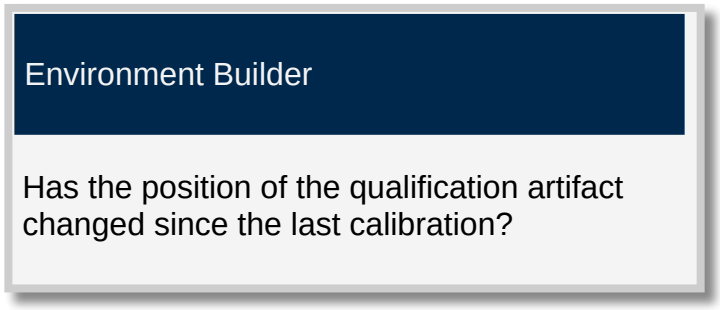
2. From the **Probe Assemblies** page, identify the probe assembly that you want to calibrate. Select **Calibrate** ( ) or select **More Options** ( ) and then **Calibrate** to open the **Calibrate Probe** dialog box to select an algorithm and define the qualification parameters.
3. From the **Select Qualification Algorithm** list, choose a qualification algorithm. For more information, see "Selecting a Qualification Algorithm".



4. On the left side of the **Calibrate Probe** dialog box, select the angles you want to calibrate.

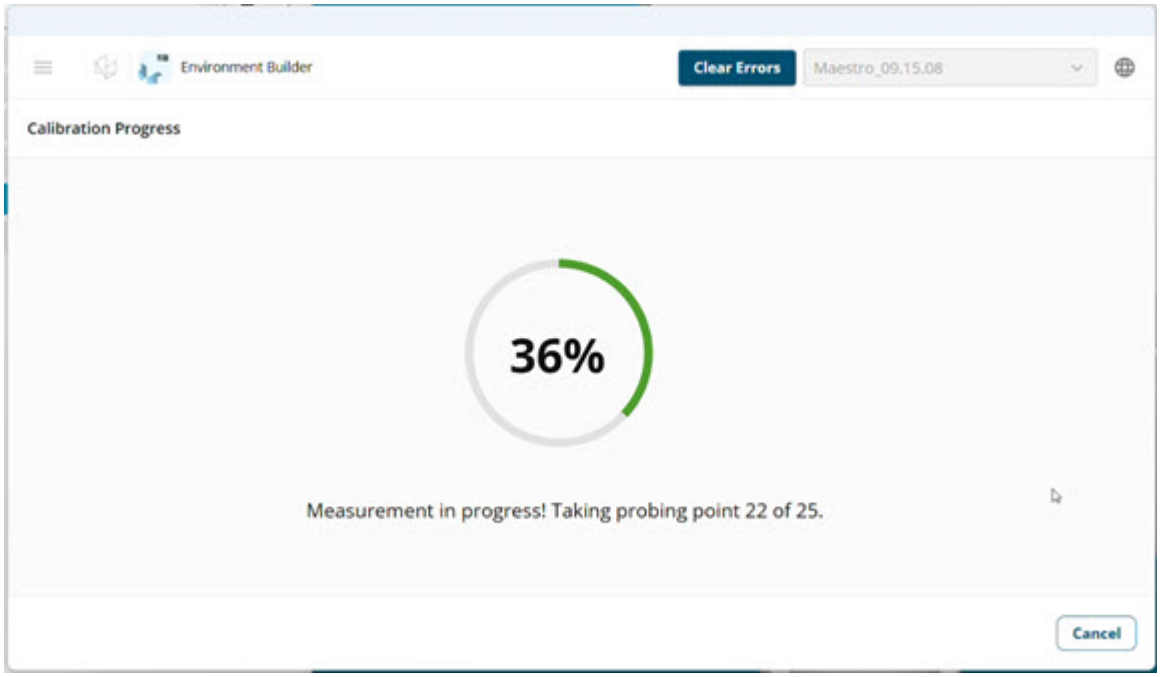
5. On the right side, under the **Qualification Parameters** area, define the qualification parameters. The parameters Environment Builder offers depend on the algorithm you choose.

6. Select **Calibrate** and Environment Builder displays the following message.



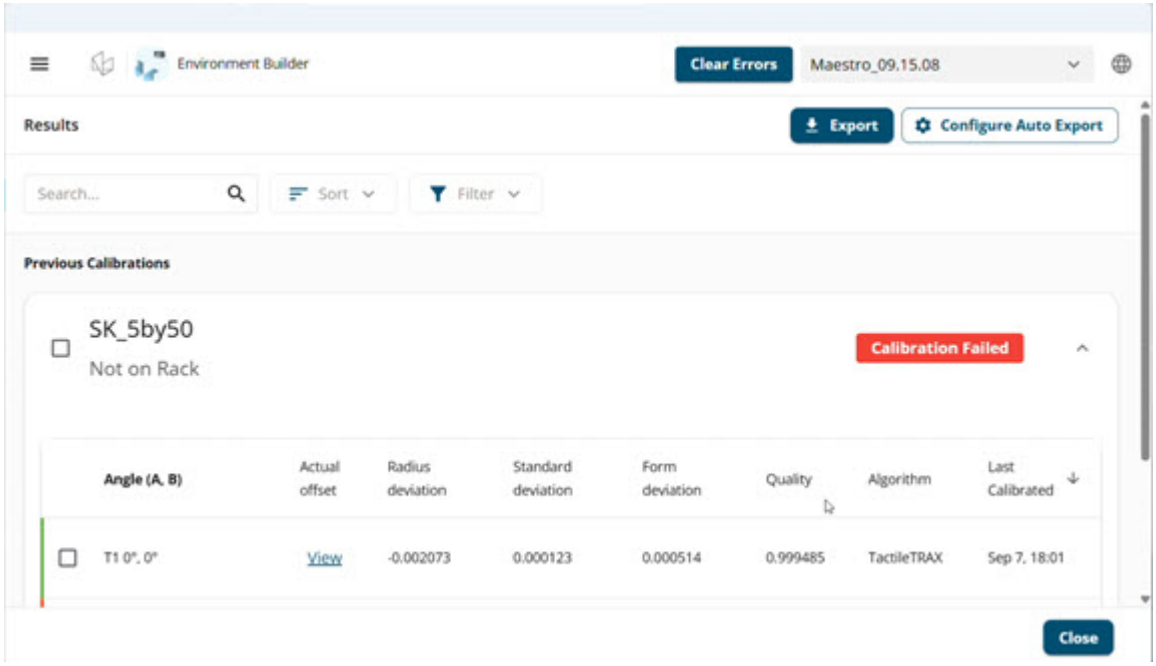
   - If you select **No**, Environment Builder opens the **Calibration Setup** dialog box where you need to select the qualification algorithm and define the qualification parameters and then select **Continue**.
   - If you select **Yes - DCC Hits** or **Yes - Manual Hit**, Environment Builder opens the **Calibration Progress** window.

7. Follow the on-screen instructions to complete the calibration process. A progress bar shows the status of the calibration process.



Environment Builder shows images associated with the probe calibration as it progresses through the calibration process.

8. When the calibration finishes, Environment Builder shows the calibration's **Results** dialog box.



9. Review the calibration results to make sure they meet your specifications. If the results do not meet your specifications, review all setup and parameter settings, make changes as needed, and then repeat this procedure.

Locating a Calibration Artifact

You must locate an artifact with the reference probe that was originally used to qualify it. This probe might not be the probe you selected for the calibration you are about to run. If the probes differ, Environment Builder tells you which reference probe the artifact requires and applies the parameters you set to locate the artifact.

Environment Builder also asks you to confirm whether the artifact has moved. When it displays the message "Has the position of the qualification artifact changed since the last calibration?", select one of the following:

- **No** – Uses the artifact's position from the last calibration.
- **Yes - Manual Hit** – Relocates the artifact by taking a manual hit.
- **Yes - DCC Hits** – Relocates the artifact automatically.

Before moving the wrist, Environment Builder reminds you to make sure that the probe is in a safe position for wrist movement.

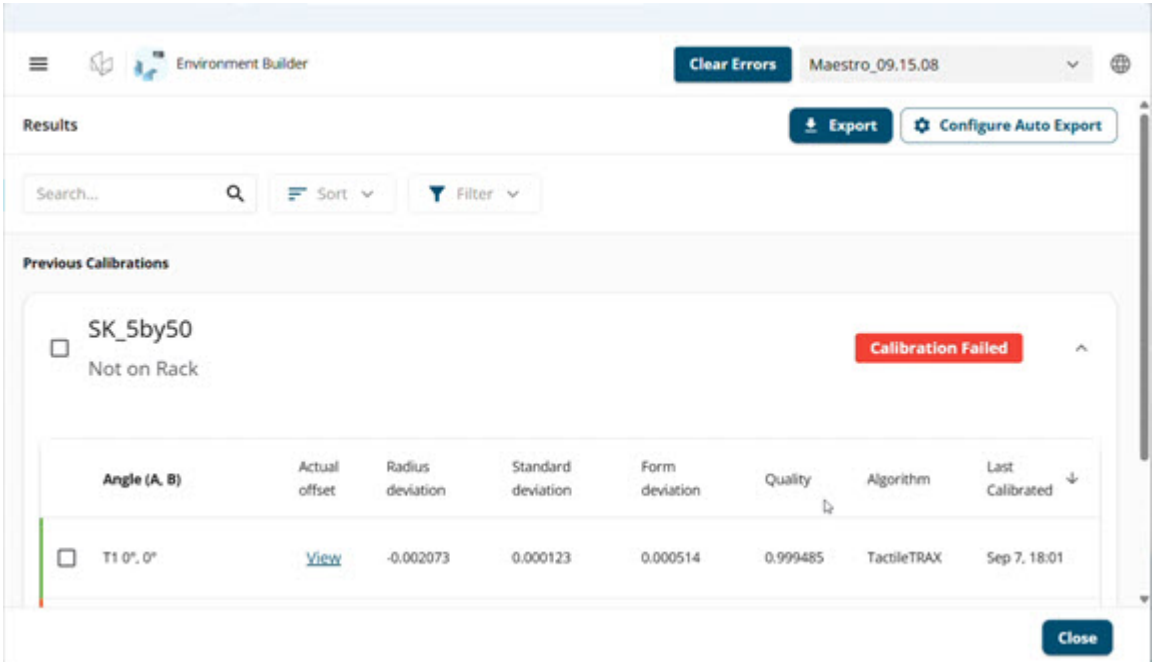
If Environment Builder cannot identify the currently loaded tip, it displays the message "Cannot identify the currently loaded tip! Ensure that you have a tip defined and loaded." To continue, do one of the following:

- Select a reference probe from the list.
- Assign the artifact to a tip. Open the **Edit Angles** dialog box for the loaded probe assembly, assign the artifact to a tip, and then locate the artifact.

You must recalibrate artifacts whose orientation can be changed by hand while they are in use. You cannot calibrate these artifacts in advance. If you change an artifact's **X Position** or **Y Position** after calibrating it, the previous alignment no longer applies. You must measure the artifact's position again.

Reviewing Calibration Results

To review your calibration results, select the menu button ( ) in the upper-left corner of the window, and then select **Results** to open the **Results** page.



The **Results** page contains the following sections:

- **In Progress Calibration** – Shows calibrations that are in progress or waiting to run.
- **Previous Calibrations** – Shows calibrations that have finished.

For each calibrated angle, Environment Builder reports the **Angle**, **X Meas. / Theo.**, **Y Meas. / Theo.**, **Z Meas. / Theo.**, **D Meas. / Theo.**, **Roundness**, and **Std. Dev.** values, along with a status of **Calibration Successful**, **Partially Calibrated**, **Calibration Failed**, or **Calibration Overdue**. Select **Repeat Calibration** to run the entire calibration again, or select **Calibrate Angle** to calibrate a single angle.

Expand a result to view the advanced values that Environment Builder recorded:

- **Tip Details** – Shows the **Size**, **Radial Dev**, **Form**, **Quality**, and **Sigma** for each tip.
- **Probe Offsets** – Shows the **X Offset**, **Y Offset**, and **Z Offset** for the probe.
- **Deflection Matrix** and **Transformation Matrix** – Show the matrices computed during calibration.
- **Maximum Error** – Shows the largest error found for each axis.

If a calibration exceeds the **Sphere Standard Deviation Limit** or **Tip Diameter Error Limit** set in the Environment Builder settings, Environment Builder notifies you and reports each type of deviation in a separate message.

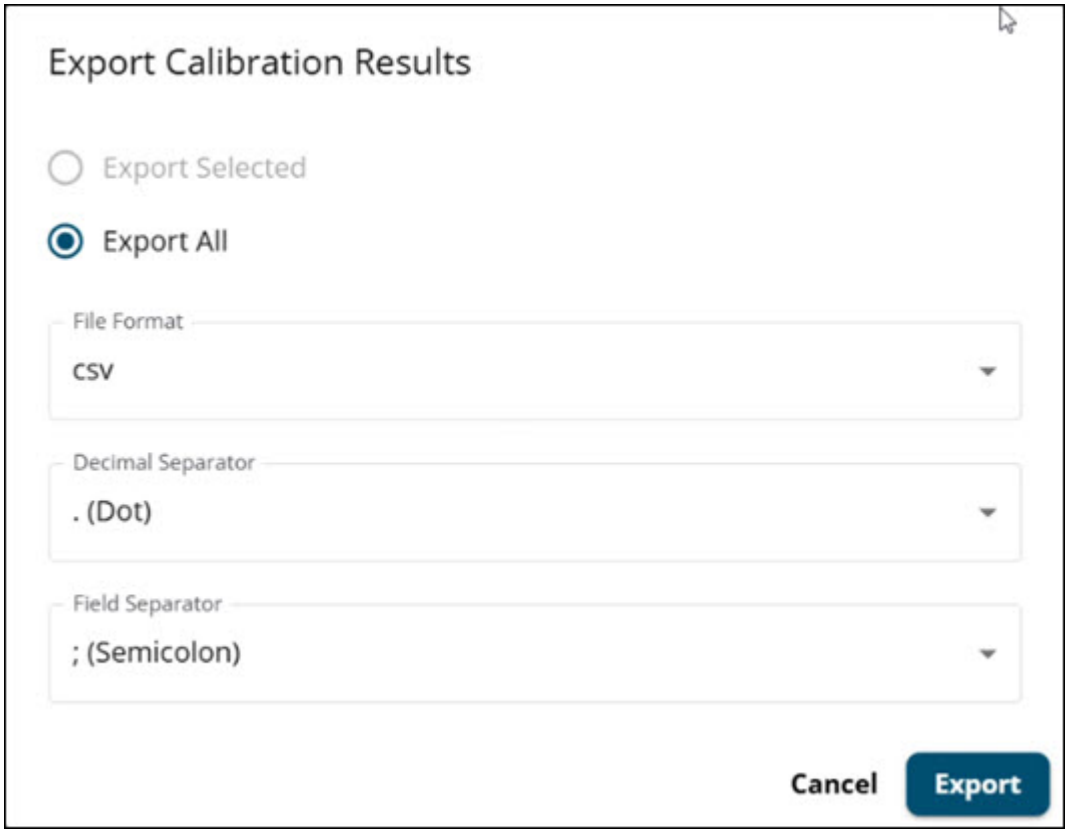
You can also reset a probe assembly's tips to their nominal values. To reset the probe tips, select **Reset Tips** in the **Edit Angles** dialog box. This resets the probe tip offsets to their nominal values and marks the probe assembly as uncalibrated. Use this option only when you intend to recalibrate the probe assembly.

Exporting Calibration Results

You can export calibration results to a file, either on demand or automatically after each calibration.

To export results on demand:

1. Select the menu button ( ) in the upper-left corner of the window, and then select **Results** to open the **Results** page.
2. Select the **Export** button to open the **Export Calibration Results** dialog box.



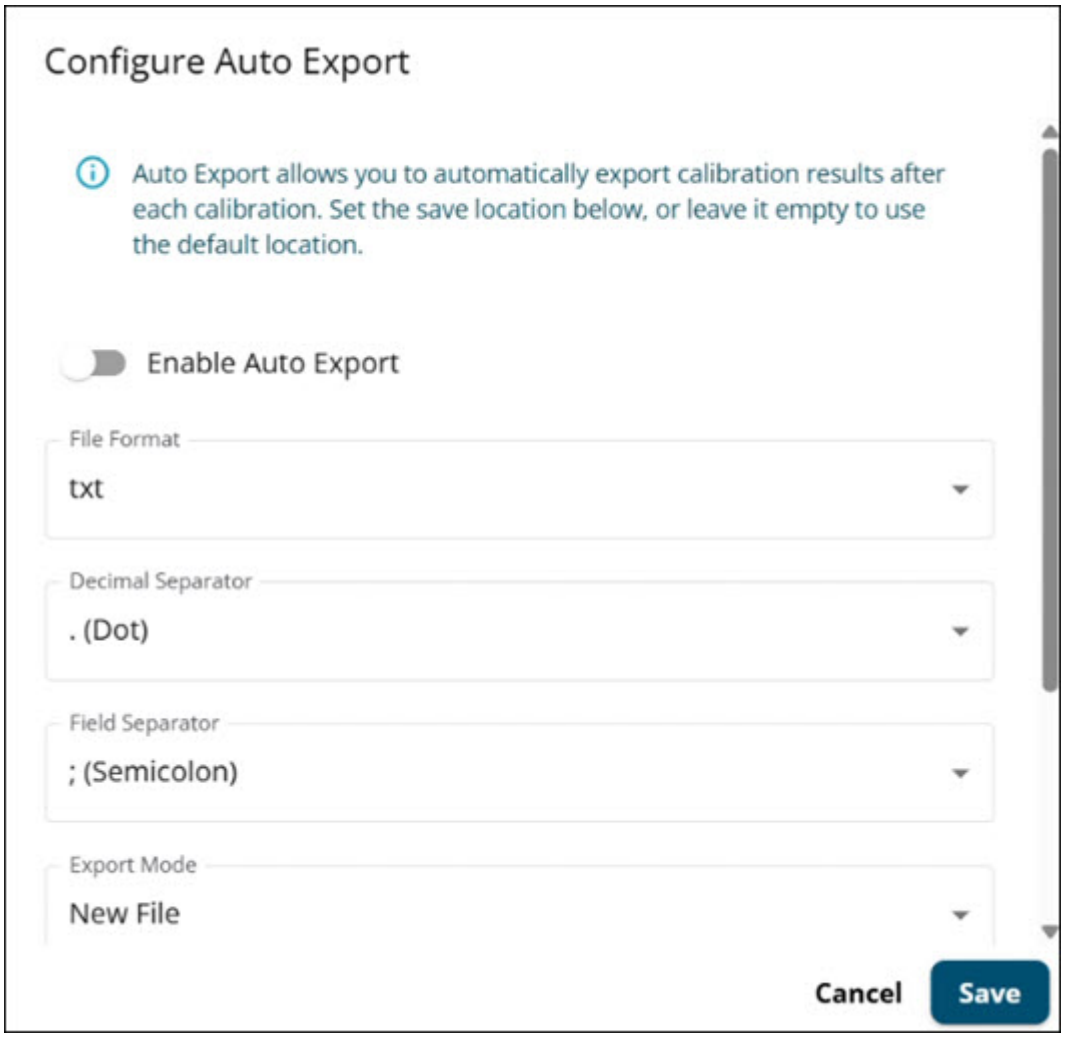
3. Select **Export All** to export every result, or **Export Selected** to export only the results that you selected.

> **Export Selected** is available only when you have selected one or more results in the **Results** page.

4. From the **File Format** list, select the format for the exported file (for example, **CSV**).
5. From the **Decimal Separator** list, select **Dot** or **Comma**.
6. From the **Field Separator** list, select **Comma**, **Semicolon**, or **Tab**.
7. Select the **Export** button to export the results.

To have Environment Builder export results automatically after each calibration:

1. Select the menu button ( ) in the upper-left corner of the window, select **Results**, and then select **Configure Auto Export** to open the **Configure Auto Export** dialog box.



2. Turn on the **Enable Auto Export** toggle.
3. From the **File Format** list, select the format for the exported file (for example, **CSV**).
4. From the **Decimal Separator** list, select **Dot** or **Comma**.
5. From the **Field Separator** list, select **Comma**, **Semicolon**, or **Tab**.
6. From the **Export Mode** list, select **New File** to write a new file each time, **Overwrite** to replace the existing file, or **Append** to add to it.
7. In the **File Name** box, type a name for the exported file. You can use letters, numbers, hyphens, and underscores only.
8. Select the **Save** button. PC-DMIS saves the files to your Downloads folder.

Probe Changers

This section provides you with the workflow that you can use to add, align, and calibrate probe changers in Environment Builder.

Adding Probe Changers in Environment Builder

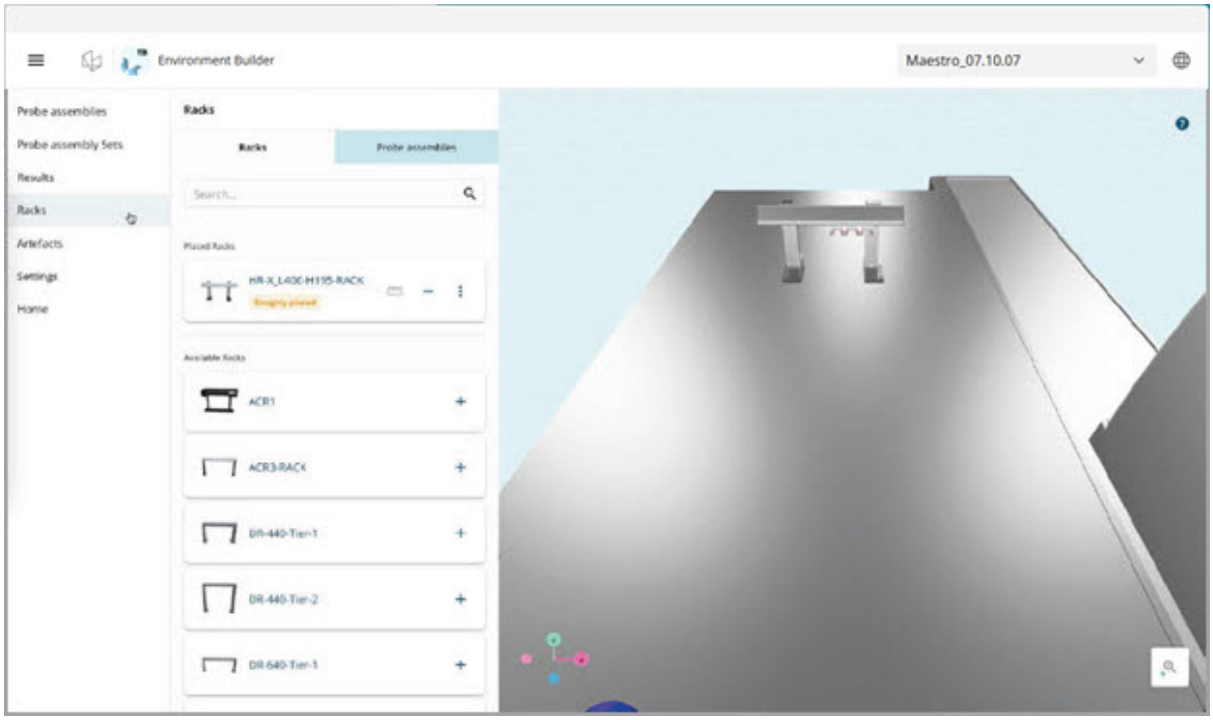
Follow this procedure to add and configure probe changers in Environment Builder.

Familiarize yourself with the list of available racks and their corresponding modules before starting.

Pay attention to part compatibility and probe changer placement.

- To avoid compatibility issues, make sure that the selected probe changer supports the modules that you want to add.
- To prevent operational errors, double-check the probe changer's placement on the CMM table.

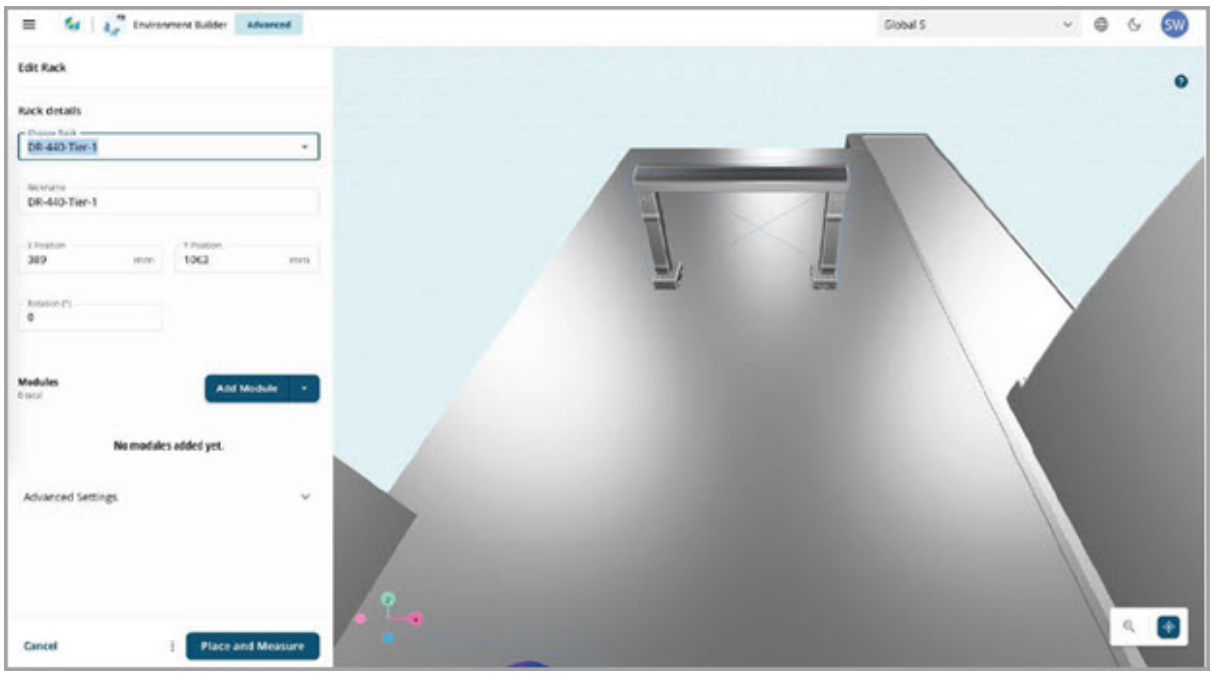
1. Open Environment Builder and click the menu button ( ) located in the upper-left of the window to show the slide-out menu panel and select the **Racks** menu option to open the **Racks** dialog box.



To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

Select the **Close** button ( ) to hide the slide-out menu panel.

2. From the **Racks** page, review the list of available probe changers and select the **Add** (**+**) button to place a probe changer on the CMM table. Drag and drop the probe changer onto a green circle close to where it is mounted on the actual CMM table. For this example, we select the Digital Rack 440 Tier 1 probe changer (DR-440-Tier-1).



3. Refine its position and angle. Select the probe changer in the 3D scene. Type the X, Y, and rotation values into the appropriate boxes (**X Position**, **Y Position**, and **Rotation**) to refine the placement of the probe changer to match the location on the actual CMM table.

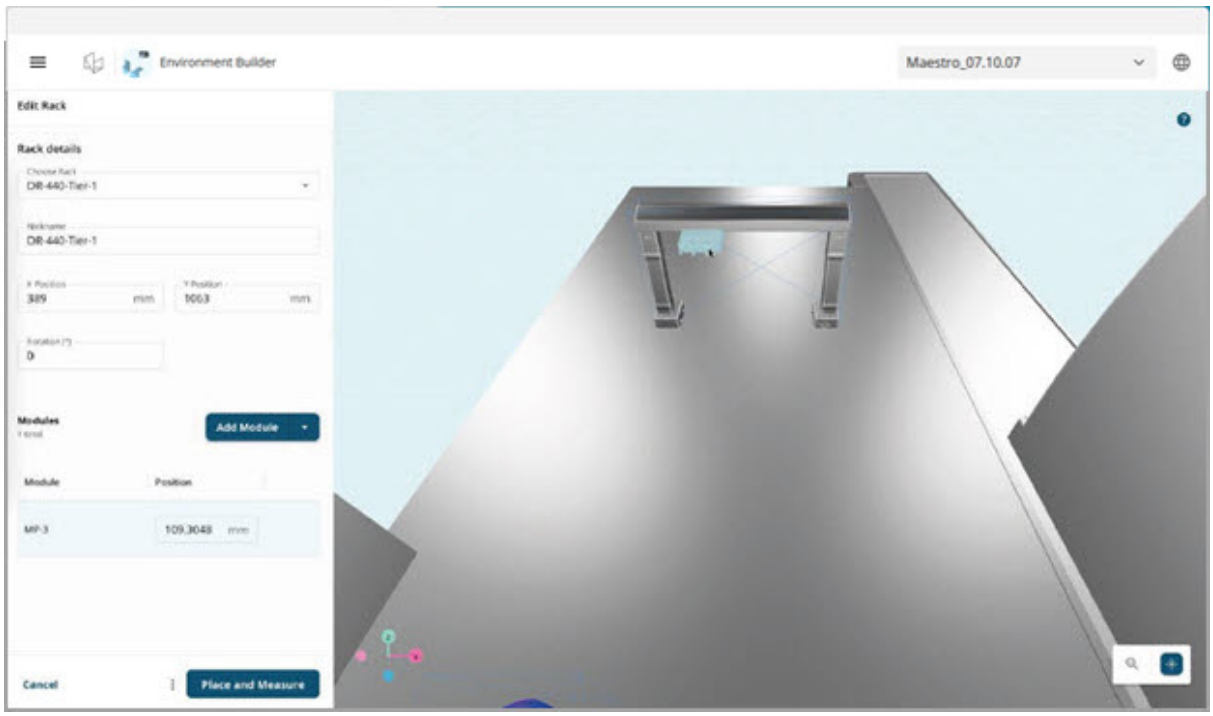
> Use the drag-and-drop functionality for quicker adjustments and positioning.

4. From the **Add Modules** list, select the first module you want to add to your probe changer. Environment Builder populates the list only with compatible modules based on the selected probe changer.

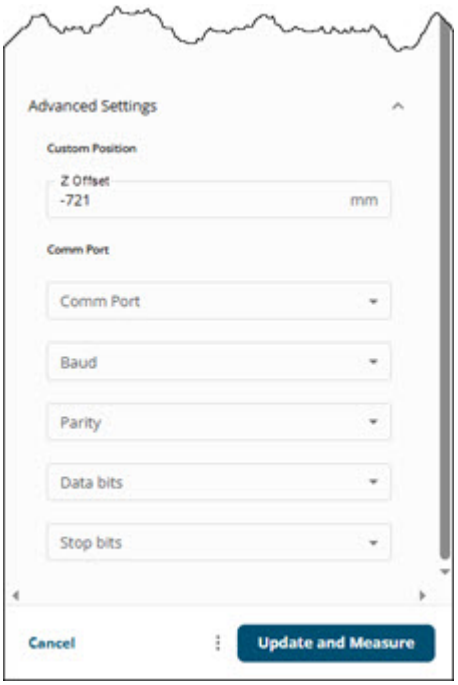
5. Add the remaining modules onto your probe changer. If you need to move a module to a different port on your probe changer, select it, then drag and drop the module onto the new port.

   You must make sure that the placement of each module on the probe changer in Environment Builder reflects the actual configuration of the modules on your CMM's physical probe changer.

   As you add new modules onto the probe changer, you can see the list of added modules with their name and position on the probe changer in the **Modules** section of the **Edit Rack** dialog box.



You have successfully added a probe changer and modules to the probe changer. If your probe changer is mounted on a block, you can account for the additional height by setting the **Z Offset** in the **Advanced Settings** section of the **Edit Rack** dialog box. Also, from the **Advanced Settings** area, you can define the communication settings if needed. These options are shown in the image below.



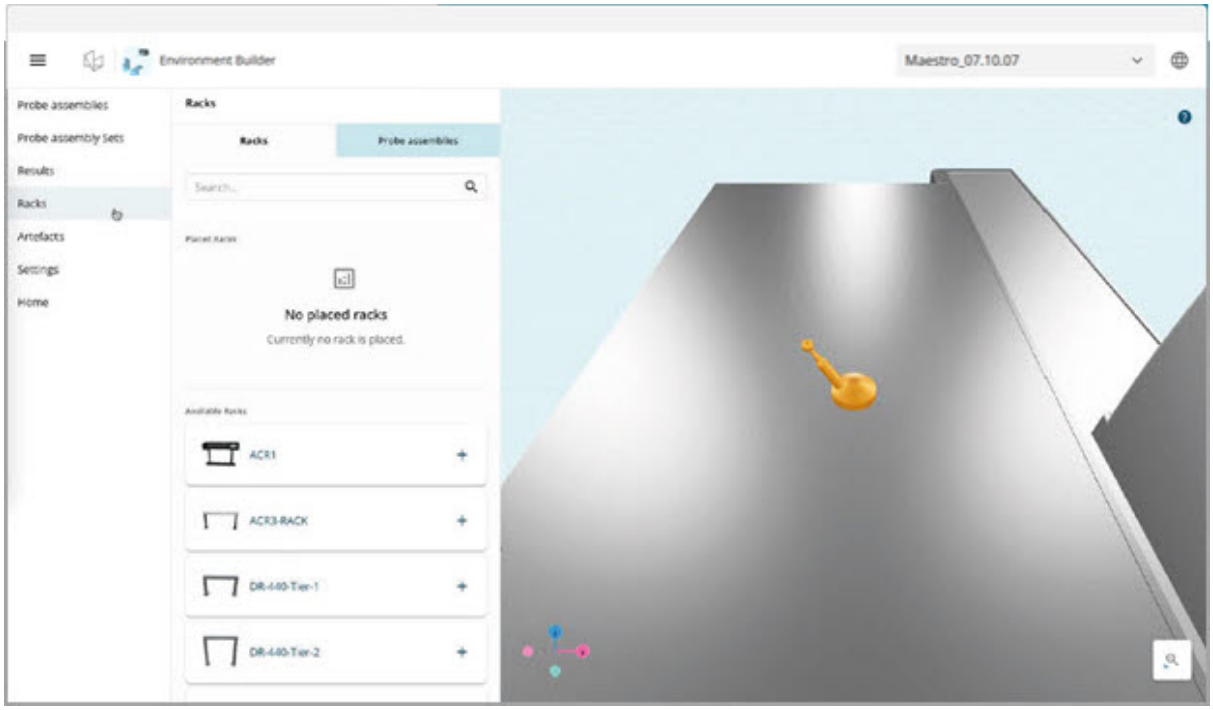
You now need to align and calibrate your probe changer.

Aligning and Calibrating Probe Changers in Environment Builder

Precise alignment and calibration of your probe changer is critical to the operational integrity of your measurement system. Follow this procedure to align and calibrate a probe changer in Environment Builder.

Before you start, familiarize yourself with the Environment Builder interface to move efficiently through the alignment and calibration process.

1. Open Environment Builder and select the menu button ( ) in the upper-left corner of the window to open the slide-out menu panel. Select **Racks** to open the **Racks** page.

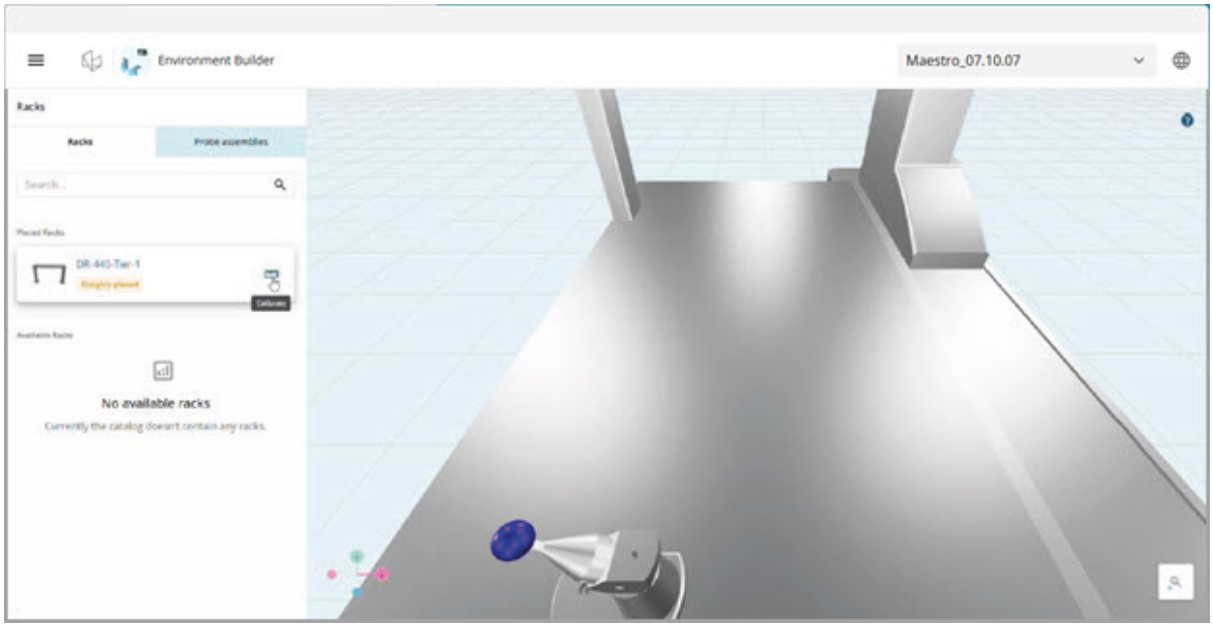


To open Environment Builder, you can use any of the options detailed in the "Environment Builder - Getting Started" topic in the PC-DMIS documentation.

Select the **Close** button ( ) to close the slide-out menu panel.

2. Verify that the probe changer is listed and add it to the Environment Builder CMM table. For information on adding a probe changer, see "Adding Probe Changers in Environment Builder".

3. Select **Calibrate** to open the **Calibration Setup** page. From the **Reference Probe** and **Angle** lists, select the probe assembly and angle to use to align the probe changer.



4. On the **Calibration Setup** page, select the check box for each alignment step that you want to run:

    - **Mount Point Definition** – Establishes the position and orientation of the probe changer.
    - **Tier Alignment** – Aligns the tiers of the probe changer. This step is not available in DCC mode.
    - **Port Alignment** – Aligns the individual ports.
    - **Automatic Port Calibration** – Calibrates the ports automatically. This step requires a valid license.
    - **Extension Alignment** – Calibrates the extensions. This step requires a valid license, and you must assign a tool to an active TKJ port.
    - **Full** – Selects all available steps.

    Before alignment starts, Environment Builder verifies that the required probe tips and angles are available:

    - Use only qualified probe tips referenced to A0, B0. If no qualified tips are available, Environment Builder displays the following message: "Calibration is not possible, as you do not have any qualified tips. Qualify tips to calibrate the rack."
    - For a rack installed along the short CMM axis (X), you need the A90, B180 angle. For a rack installed along the long CMM axis (Y), you need the A90, B90 angle. Environment Builder identifies the missing angle and prompts you to verify the axis along which the rack is installed.

- Select a probe assembly with a straight stylus between D3 mm × L20 mm and D5 mm × L50 mm, such as a DTT or DST sensor.

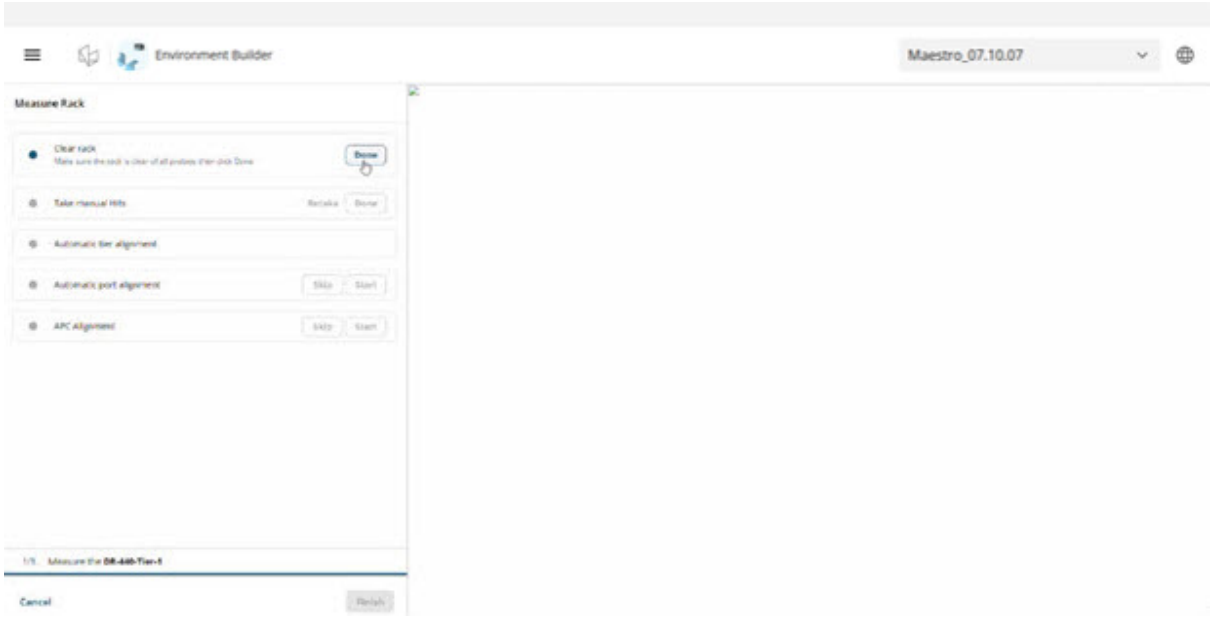
5. Select **Continue** to begin the automatic alignment adjustments and open the **Measure Rack** dialog box. The dialog box displays the probe changer calibration tasks that you must complete.

   All DCC measurements are taken 400 mm from the rack legs. Verify that this area is clear before you continue. In DCC mode, Environment Builder warns you that the machine moves immediately after you select **Continue**.

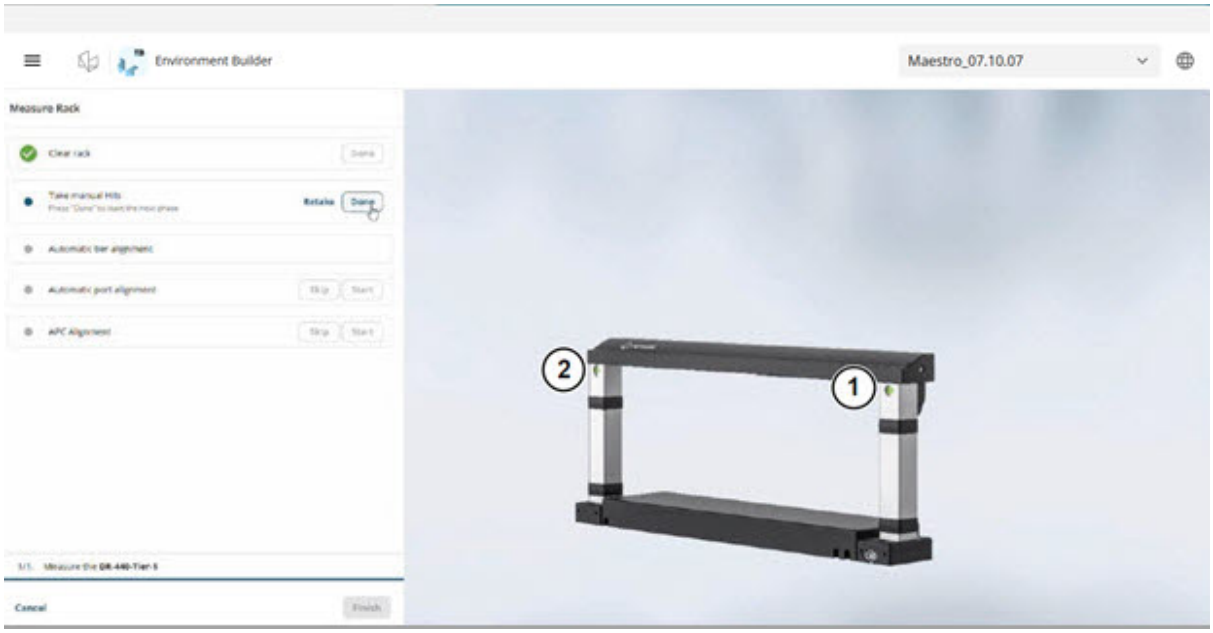
   To stop an alignment that is in progress, select **Stop**. Environment Builder displays the following message: "Are you sure you want to stop the rack calibration? The progress made will not be saved."

6. Verify that the rack is clear. Check that the path from the current probe assembly position to the probe changer is free of obstructions. Also verify that the rack is clear of tools and other objects and is properly adjusted and mounted.

   When the area is clear, select **Done** to complete the task. Environment Builder displays a green check mark for the completed task.



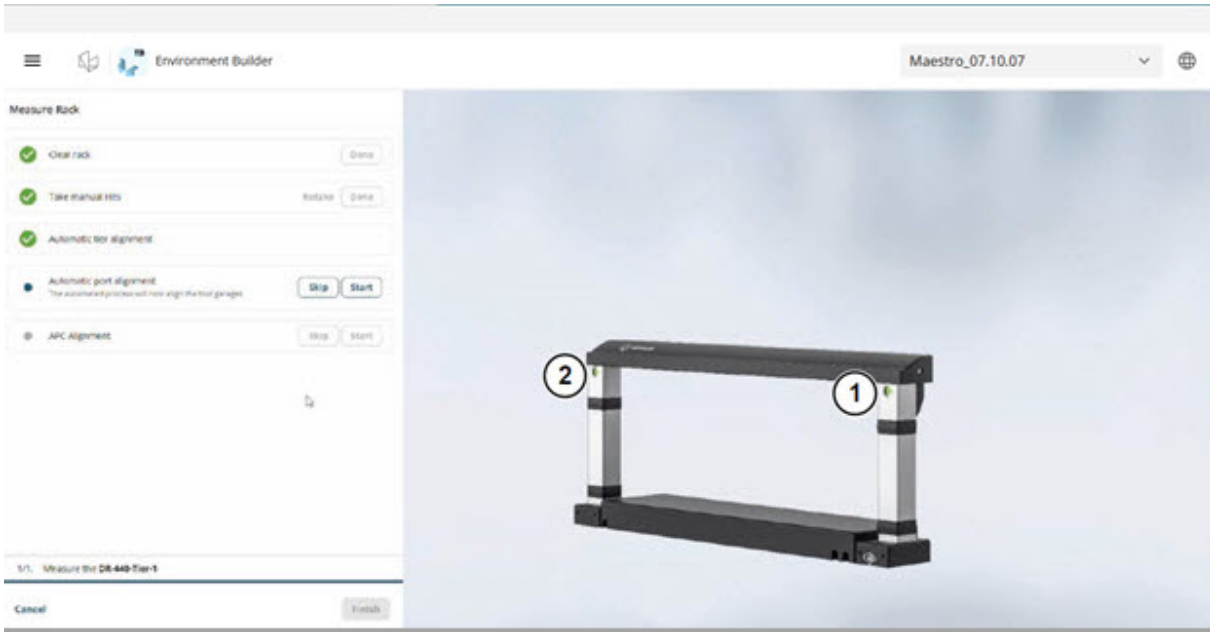
7. On the **Take manual hits** task, follow the Environment Builder prompts to take the first and second manual probe hits at the locations shown below.



You can take the hits anywhere on the flat area at the front of each leg near the top, below the rail connection. Environment Builder uses these hits only to set the orientation of the probe changer for the automatic hits in the next task.

After you take both hits, select **Done** on the **Take manual hits** task. Environment Builder displays a green check mark for the completed task.

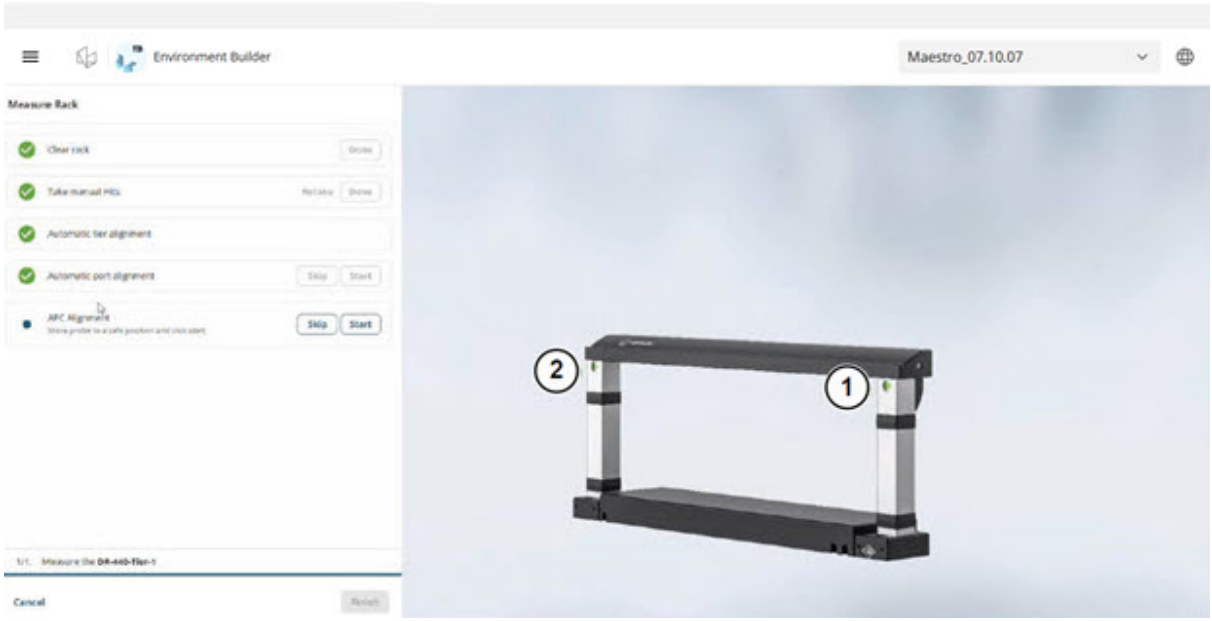
8. The **Mount Point Definition** task runs automatically. No user interaction is required. Environment Builder takes four automatic hits on each hole at the top-front of each probe changer leg, as shown below.



When the task is complete, Environment Builder displays a green check mark for the task.

9. The **Tier Alignment** and **Port Alignment** tasks align the tiers and individual ports of the probe changer. You can start or skip each task.

   Follow the Environment Builder prompts to complete a task. When a task is complete, Environment Builder displays a green check mark and advances to the next task.

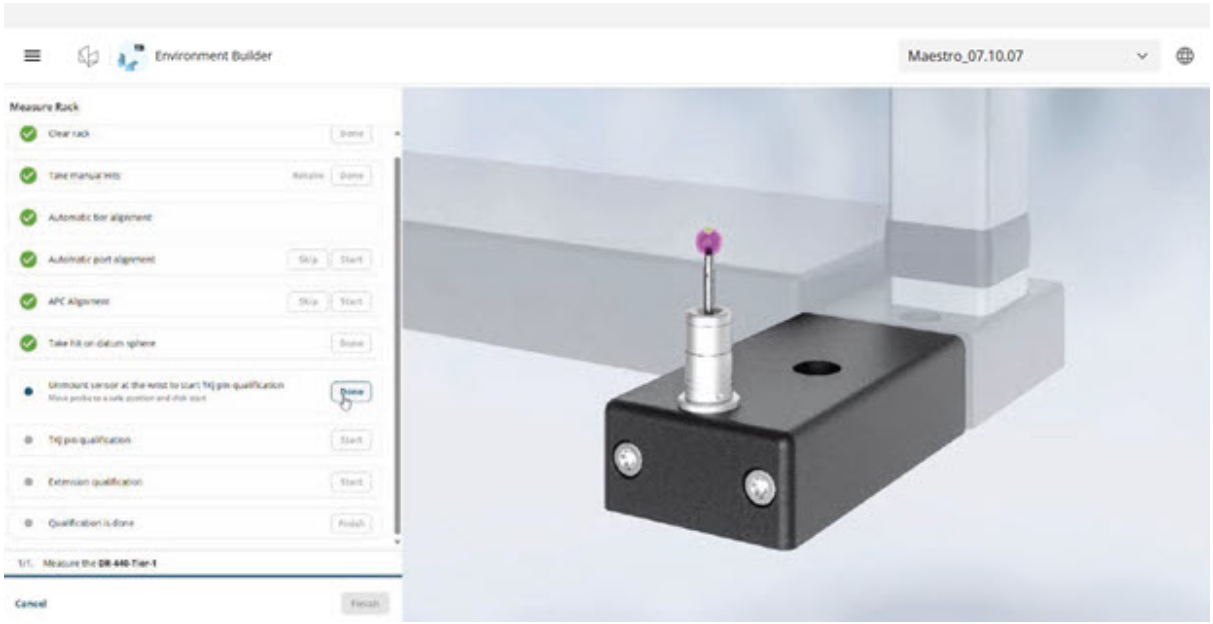


If you select **Skip** for an optional task, Environment Builder displays a green check mark for the task and advances to the next task.

10. The **Automatic Port Calibration** task calibrates the ports automatically. You can start or skip this task. A valid license is required. Without a valid license, Environment Builder displays the following message: "Automatic Port Calibration functionality requires a valid license."

    > For example, you can skip this task if you have an X1 sensor and want to change only the stylus configurations (modules), not the sensor.

    To start the task, select **Start** and follow the prompts to take manual hits on the datum sphere. Environment Builder then displays five additional tasks, as shown below.

    After you take the required hits, select **Done** on the **Take hit on datum sphere** task. Environment Builder displays a green check mark for the completed task.

11. Prepare for TKJ pin calibration. Unmount the TKJ assembly, and then move the probe assembly to a safe location. Verify that the path between the probe assembly and the probe changer is clear.

    On the **Unmount sensor at the wrist to start TKJ pin qualification** task, select **Done**. Environment Builder displays a green check mark for the completed task.

12. Calibrate the TKJ pin. On the **TKJ pin qualification** task, select **Start**. Environment Builder prompts you to take manual hits on the datum sphere.

    After you complete the required hits, select **Done**. Environment Builder displays a green check mark for the completed task.

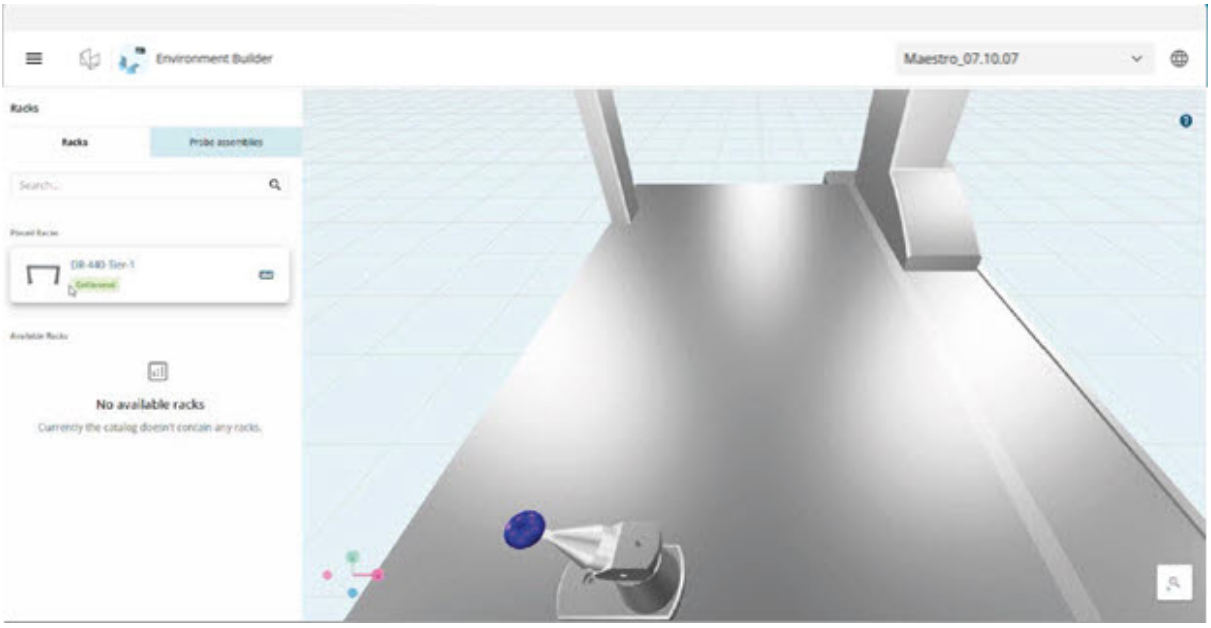
13. The **Extension Alignment** task calibrates each extension in the probe changer. Attach each extension one at a time, and then select **Start** to perform the

qualification check. Repeat this process until you have attached and calibrated all extensions. Then select **Done**.

Environment Builder displays a green check mark for the completed task.

This task requires a valid license, and you must assign a tool to an active TKJ port. Until you meet these requirements, Environment Builder displays the following message: "Extension Alignment is unavailable until a tool is assigned to an active TKJ port."

14. When you complete all the tasks, select **Finish** on the **Qualification is done** task to confirm that the probe changer was aligned and calibrated successfully.



Environment Builder returns to the **Racks** page. The probe changer displays a green "Calibrated" message. You can now use the calibrated probe changer in your measurement routines.

Placing Probe Assemblies on a Probe Changer

After you add and align a probe changer, you can assign probe assemblies to its ports so that the machine knows where to find and return them. You can place an entire probe assembly or only the part that physically sits in the port, called a probe sub-assembly.

To place a probe assembly on a probe changer:

1. From the **Probe Assemblies** page, open the menu on the probe assembly's tile and select **Place on Rack**. You can also right-click a probe changer in the 3D scene and select **Place Probe Assembly**.

2. On the **Parts Selection** step, select one or more adjacent parts to place as a probe sub-assembly. If a part is already placed, Environment Builder tells you where it is: "Part already placed on <rack>, <port>".

3. Select **Continue** to move to the **Rack and Port Selection** step.
4. From the **Choose Rack**, **Choose Module**, and **Choose Port** lists, select where to place the probe sub-assembly.
5. Select **Place**.

The **Racks** page has a **Probe Assemblies** tab that shows **Placed Probe Assemblies** and **Available Probe Assemblies**. Each placed probe assembly shows its location: the rack and port name, **Partially Placed** if only some of its parts are on a rack, or **Placed in Multiple Ports** if its parts are spread across several ports.

Be aware of the following:

- Make sure that the placement of each probe assembly in Environment Builder matches the actual configuration of the physical probe changer.
- If you try to assign the same parts to more than one port, Environment Builder asks you to confirm that you have multiple physical copies of those parts. Otherwise, you cannot assign the same set of parts to another port.
- Use **Probe Utilities** to mark a probe assembly as one that you use with the rack for unloading only.

Clearing Ports and Racks

To remove probe assemblies and modules from a probe changer, right-click in the 3D scene and select one of the following options:

- **Clear Port** – Removes all probe sub-assemblies from the port you right-clicked. Environment Builder asks, "Are you sure you want to remove all probe sub-assemblies?"
- **Clear Rack** – Removes all probe sub-assemblies from the probe changer. Environment Builder tells you how many probe assemblies are affected before you confirm the action.
- **Remove Module** – Removes a module from the probe changer.
- **Remove Rack** – Removes the probe changer from the CMM table.

You can also remove a single probe sub-assembly or all parts of a probe assembly from the racks that contain them. Environment Builder asks you to confirm each action before making the change.

Enabling and Disabling a Digital Rack

You can disable a Digital Rack without removing it from your environment. This is useful when the rack is physically disconnected or when you do not want the machine to use it temporarily.

To disable a Digital Rack, right-click it in the 3D scene and select **Switch Off**. To enable it again, select **Switch On**.

A disabled Digital Rack remains on the CMM table and retains its position, alignment, modules, and probe assembly assignments. You do not have to set it up again when you enable it.

Digital Rack calibration requires a valid license. If you do not have a valid license, Environment Builder displays the message "Digital Rack calibration requires a valid license."

Aligning a Travel Rack

To align a travel rack:

1. Select the travel rack and start the alignment.
2. On the **Select your alignment steps** page, select the steps you want to run:
    - **Mount point definition**
    - **Y-Axis alignment**
    - **Port calibration**
3. Follow the prompts for each selected step. Environment Builder asks you to:
    - Confirm that the rack is clear of tools and other objects and is properly adjusted and mounted.
    - Select the angle for the alignment.
    - Confirm whether the port Y positions have already been adjusted.
    - Move the machine to a safe position before defining the mount point.
    - Move the probe down in Z and adjust Y if needed.
    - Position the sensor down at the wrist when required by the step.

    When the machine is about to move in DCC mode, make sure that the surrounding area is clear and that no personnel are near the machine.
4. If a step fails, clear the machine error and either continue the alignment or abort it.
5. When all selected steps finish, Environment Builder displays "Alignment Completed" and returns you to the **Racks** page.

Creating Auto Features

Creating Auto Features: Introduction

PC-DMIS provides a library of functions and routines to facilitate the automatic measurement of parts. These functions and routines allow PC-DMIS to easily program for measurement a variety of part features and then add them into the measurement routine as "Auto Features". In many cases, this automatic feature recognition is as simple as single-clicking with your mouse on the appropriate feature in the Graphic Display window. While Auto features have their history in sheet metal measurement or in the measurement of other thin-walled materials using PC-DMIS's Direct Computer Control (DCC), today you can use them in both DCC Mode and Manual Mode to measure parts constructed from a variety of materials.

To work with Auto features, select the appropriate feature type from the **Insert | Feature | Auto** submenu. PC-DMIS opens the **Auto Feature** dialog box for the selected feature type. You can then interact with this dialog box to create the necessary Auto features.

The main topics in this chapter include:

- Graphical Representation of Auto feature and QuickFeature Vectors
- Using the ArcReductionAngle Setting
- Quick Ways to Create Auto Features
- Creating QuickFeatures
- Using the Measurement Strategy Widget
- The Auto Feature Dialog Box
- Inserting Auto Features
- Auto Feature Field Definitions
- Setting Up Relative Measure

Depending on your specific version of PC-DMIS, the Auto Feature functionality may only be accessible as an added option to your basic PC-DMIS geometric software package. Consult your PC-DMIS vendor to determine if your version supports this functionality.

Graphical Representation of Auto Feature and QuickFeature Vectors

Vector colors and the dialog box labels associated with the vectors are color-coded for easy identification.

PC-DMIS uses the following colors for both the vector graphical display as well as the respective dialog box label.

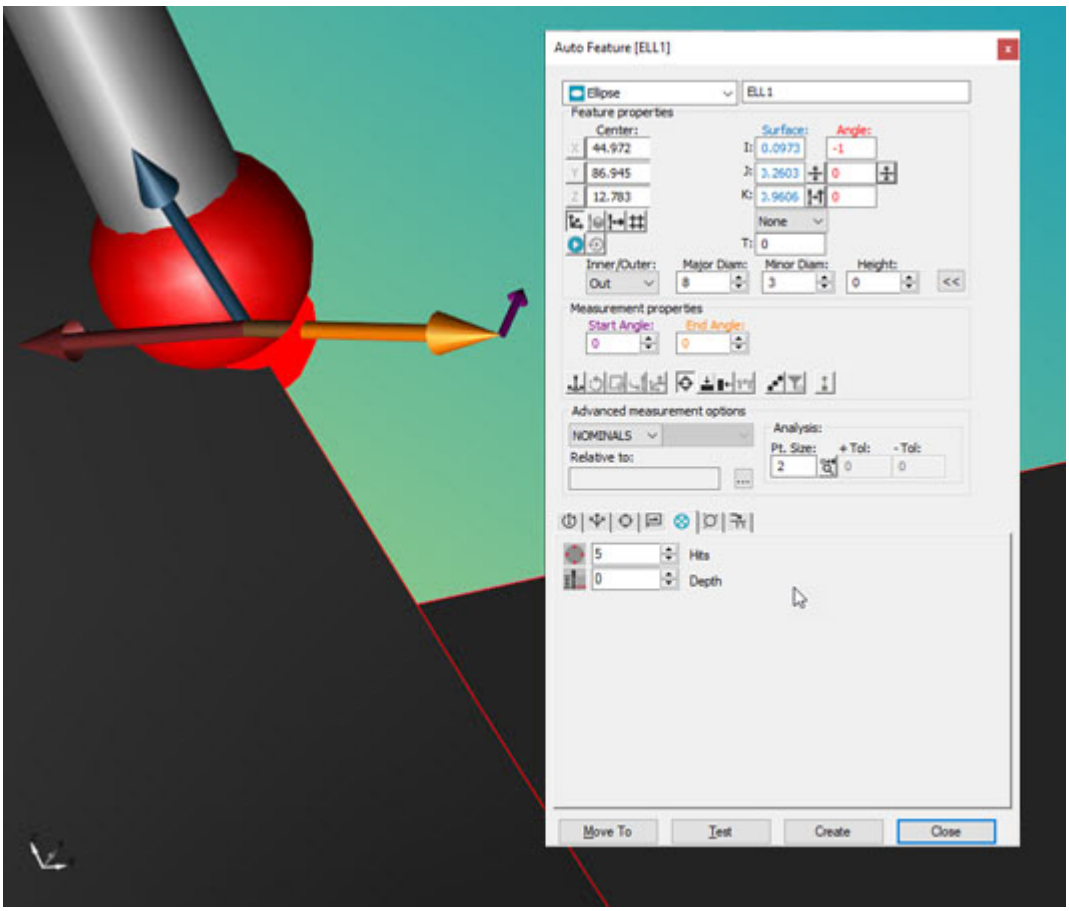
Auto Features

- **Blue** - Auto feature Surface vectors
- **Green** - Auto feature Edge vectors
- **Orange** - Auto feature End Angle vectors
- **Purple** - Auto feature Start Angle vectors
- **Red** - Auto feature Angle and Line vectors

QuickFeatures

- **Light Blue** - QuickFeature Surface vectors
- **Light Green** - QuickFeature Edge vectors
- **Light Red** - QuickFeature Angle/Line vectors

An example of the Ellipse Auto feature showing the Surface vector, Angle vector, Start Angle vector, and End Angle vector is shown below.



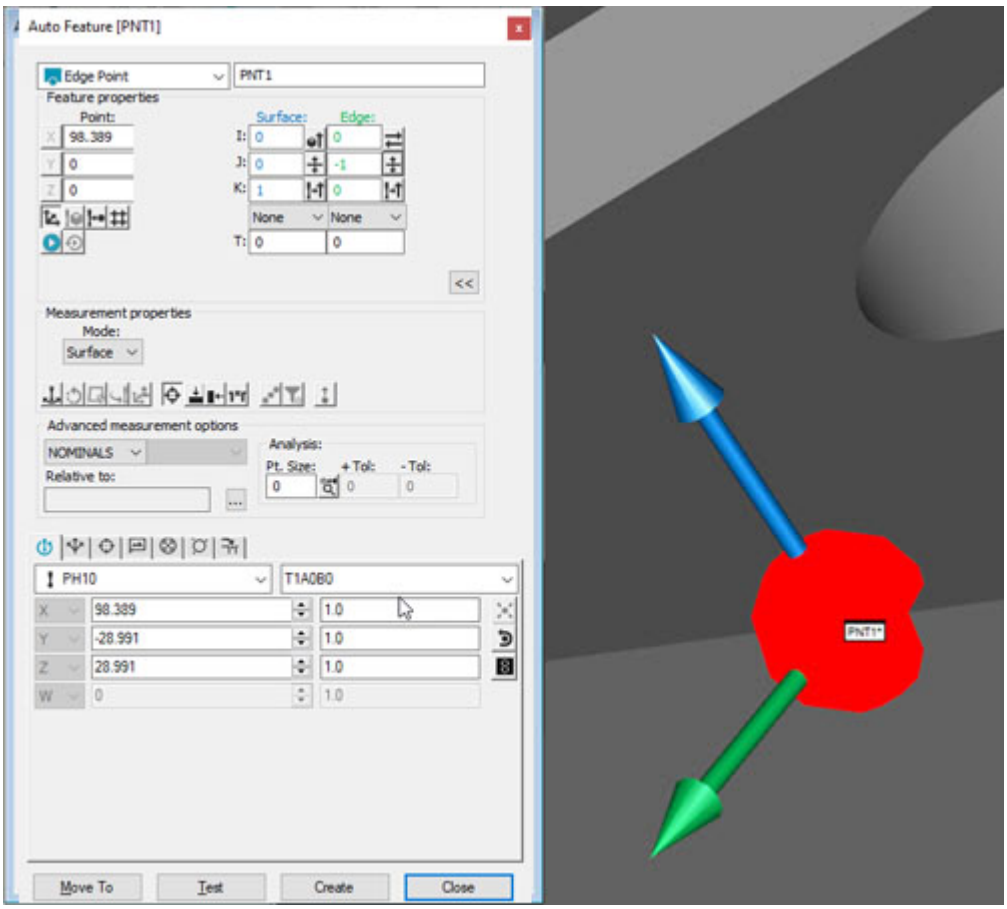
Example showing an ellipse auto feature and its associated vectors.

You can easily determine each vector by associating its color in the Graphic Display window with the labels in the dialog box.

The **Auto Feature** dialog box for the Edge Point example below shows the Surface vector color as blue and the Edge vector color as green. The Surface and Edge labels in the respective dialog boxes are colored accordingly.

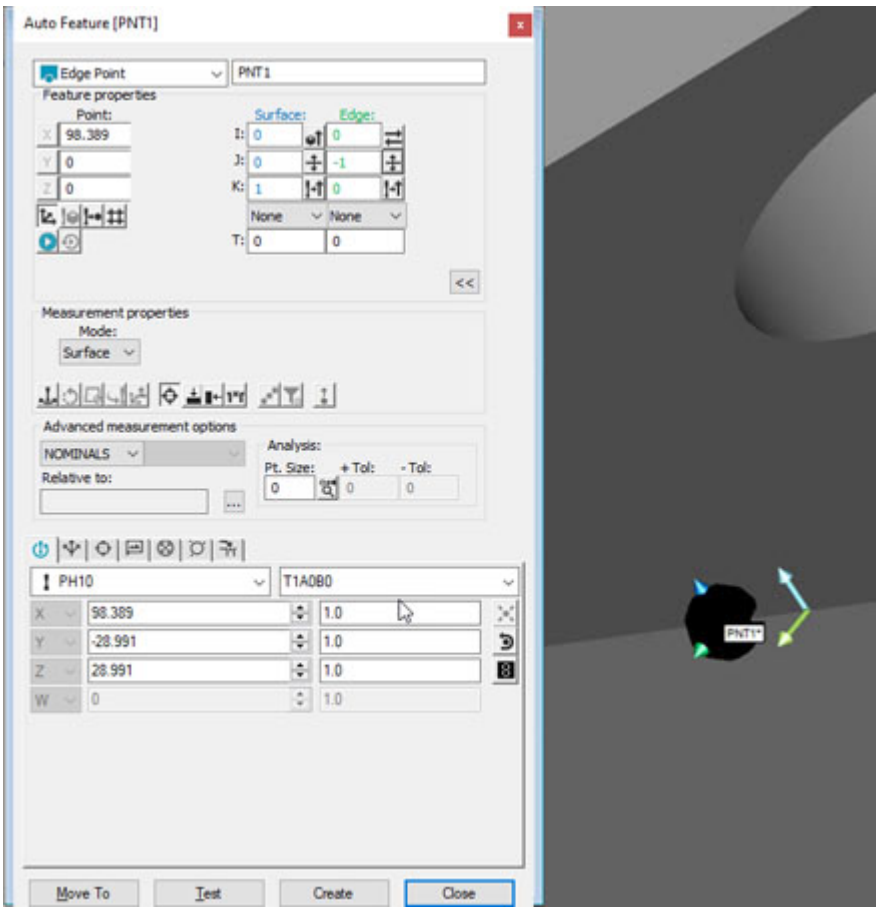
Standardized Vector Sizing

The size of the vectors is set by the **Pt. Size** value entered in the **Analysis** section of the respective dialog. For example, the image below shows the vector size for an Edge Point Auto feature when the **Pt. Size** value is set to 5.



Example of an edge point auto feature with the Pt. Size set to 5.

If the **Pt. Size** value is set to 0 (zero), the vectors are a fixed value and do not auto-scale. This is true for both the Auto features and QuickFeature modes as shown below.



Example of an edge point auto feature with the Pt. Size set to 0 (zero).

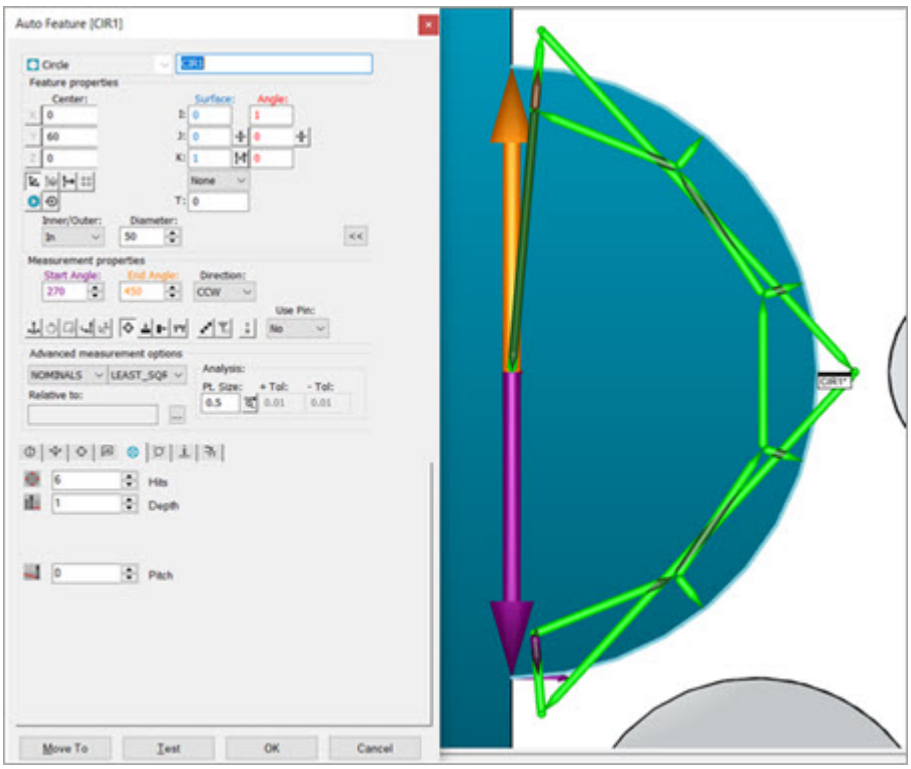
Using the ArcReductionAngle Setting

In the machining process, there is a possibility (based on the tip diameter and/or the part's shape or even in the machining process itself) that the edge of the feature being machined comes out rounded in some way or just roughly machined in the start and end areas. If this is a reoccurring issue, you can define a default arc reduction angle on those features.

The `ArcReductionAngle` setting in the PC-DMIS Settings Editor application allows you to move the first and last probed points away from the edges of a partial hole (or boss). The default value for this setting is 2 degrees but you can set it to a value that works for you. This works for Auto Circle, Auto Cone, and Auto Cylinder feature types.

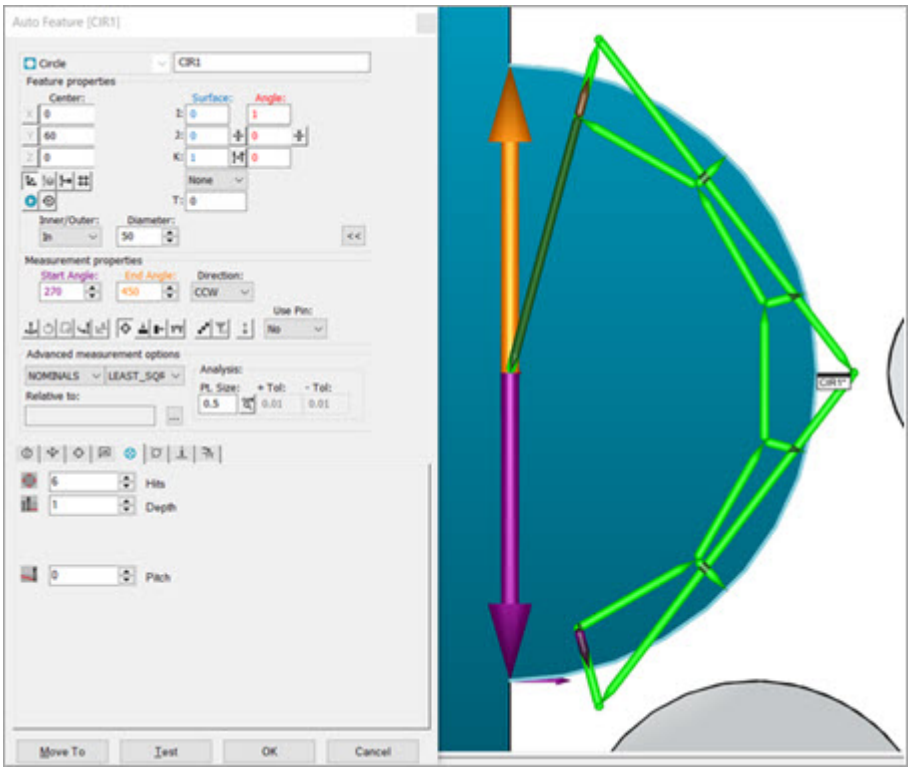
It is important to note that, even on existing measurement routines, if you make a change to this setting, you must restart the measurement routine to allow the change to take effect.

For example, in the images below, you can see the difference when you set the `ArcReductionAngle` value to 5 compared with a value of 15.



Example of a semi-circle feature with the ArcReductionAngle value set to 5.

When you run your measurement routine with this setting and you defined the measurement of this feature to measure from 0 to 180 degrees, PC-DMIS would actually measure this feature between 5 and 175 degrees.



Example of a semi-circle feature with the ArcReductionAngle value set to 15.

When you run your measurement routine with this setting and you defined the measurement of this feature to measure from 0 to 180 degrees, PC-DMIS would actually measure this feature between 15 and 165 degrees.

Quick Ways to Create Auto Features

In addition to entering values to create your auto features, the following modes are also available:

- Box selection - Click and then drag to box-select several CAD elements. Once you click **Create**, PC-DMIS creates, at once, multiple Auto features from the selected set of features.
- Single-Click selection - Click once on a supported CAD element to populate the **Auto Feature** dialog box with the appropriate nominal values.
- QuickFeature selection - Depending on the CAD element, press Shift or Ctrl + Shift, and then position your mouse pointer over a CAD element. Once PC-DMIS highlights the feature, click on the CAD element to create the associated Auto feature. For details on how to create QuickFeatures, see "Creating QuickFeatures".

Box-Selecting to Create Multiple Auto Features

You can draw a box on top of a CAD image to automatically create multiple Auto features for these supported feature types:

- Auto Vector Point
- Auto Surface Point
- Auto Edge Point
- Auto High Point
- Auto Circle
- Auto Cylinder

To Box-Select and Create Features

To use the box-select method to create multiple auto features of either the circle or cylinder Auto feature types, follow this procedure:

1. Import the CAD model that has the Auto features you want to box-select.
2. Rotate the part and select either wireframe or solid to select a view that best shows the features you want to include.

3. Access the **Auto Feature** dialog box (**Insert | Feature | Auto**) for either circle or cylinder Auto features.

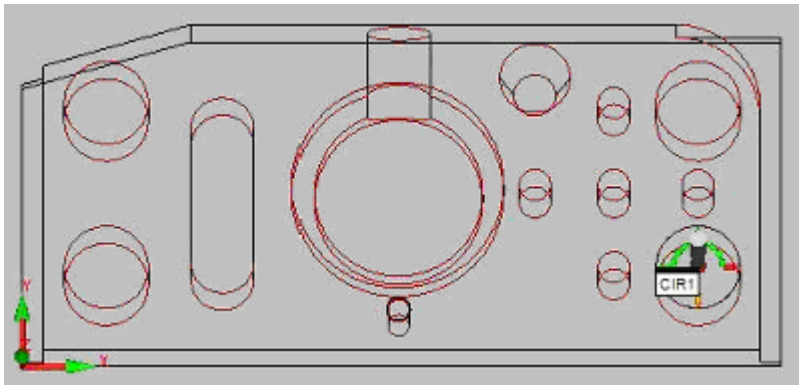
4. With the dialog box open, click the mouse, and drag a box around those feature types for which you want to create Auto features. Release the mouse button. PC-DMIS displays the **CAD Select** dialog box, showing the number of objects selected.

5. Click **Create**. Based on the objects selected, PC-DMIS generates multiple Auto features of the selected type.

Box-Selecting Details

- Box-selecting only works on objects visible in the Graphic Display window. This prevents unseen objects from being used in the creation of features.

- Models with little or no wireframe geometry should be slightly tilted (or rotated) in the Graphic Display window so that the desired surfaces and features are visible as shown below.

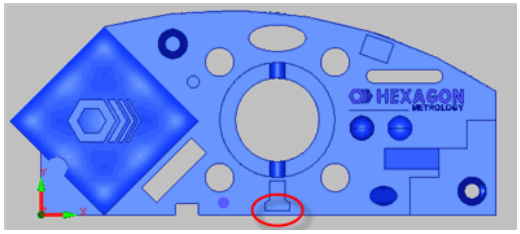


Example of a slightly rotated part.

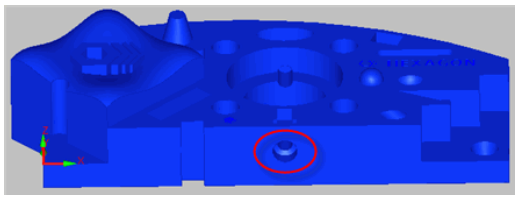
- Because of tessellation and display accuracy, the bleeding of underlying surfaces and curves may result in extra unexpected objects used to create the features. While PC-DMIS attempts to minimize this bleeding by comparing the selected objects with each other to determine the minimum number of pixels necessary to be considered a valid selection, this is not a foolproof method, and some mostly hidden objects may get selected in order to minimize the likelihood of eliminating a valid object.

- When creating the features from the selected objects, objects with vectors perpendicular to the current view are mostly ignored. For example, using the Hexagon block displayed in Z+ with the entire model box-selected, PC-DMIS will not generate a feature for the front bore that intersects the center bore.

  Selecting in the Z+ view, this circle's vector is perpendicular to the workplane and therefore PC-DMIS would not generate a circle for it.



If you tilted the part and then box-selected, PC-DMIS would select this feature.

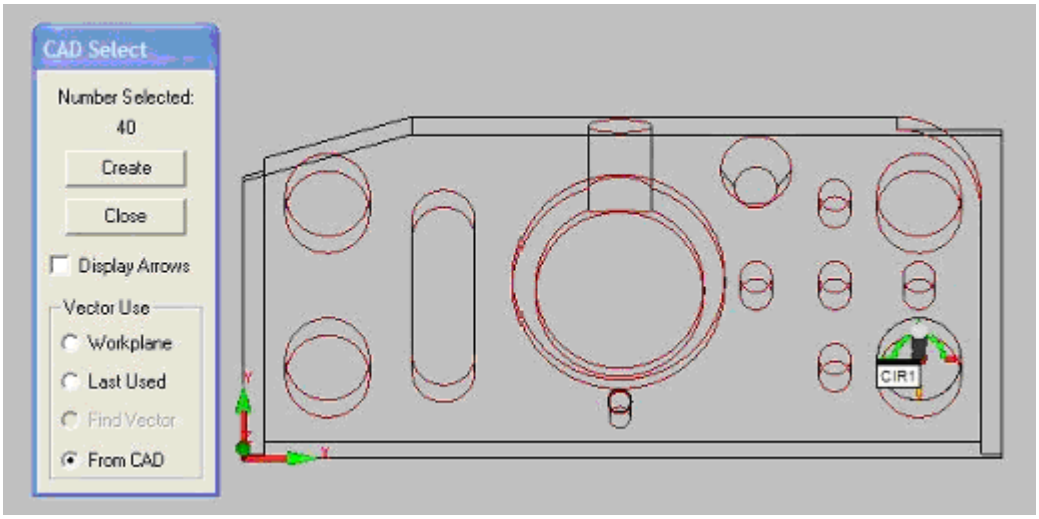


- PC-DMIS performs filtering routines to ensure that it doesn't use the same CAD object to create another feature in the same position.
- As PC-DMIS generates the features after you click **Create**, you can see each item's information displayed in the Status bar.

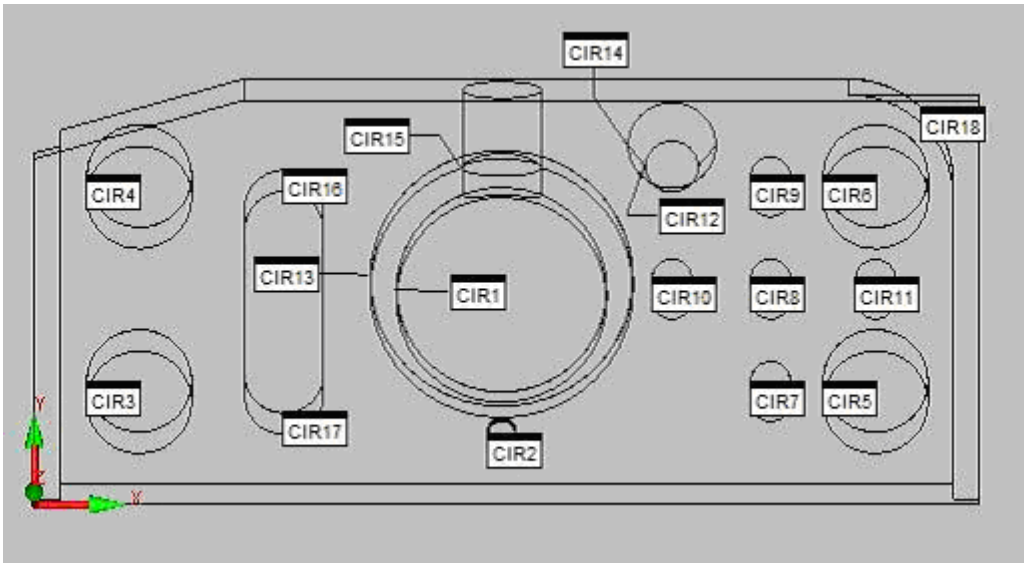
Example 1: Box-Selecting Auto Circles with Wireframe Data

This example uses the Brown and Sharpe wireframe block (Bsbwire.igs) with auto circles:

If you box-select the entire model, which is not in a complete Z+ orientation, you will notice that the top and bottom circles and arcs are selected because they are the visible objects which also satisfy the Auto Circle CAD filter logic.



Click **Create** from the CAD Select dialog box. You will get something like this:



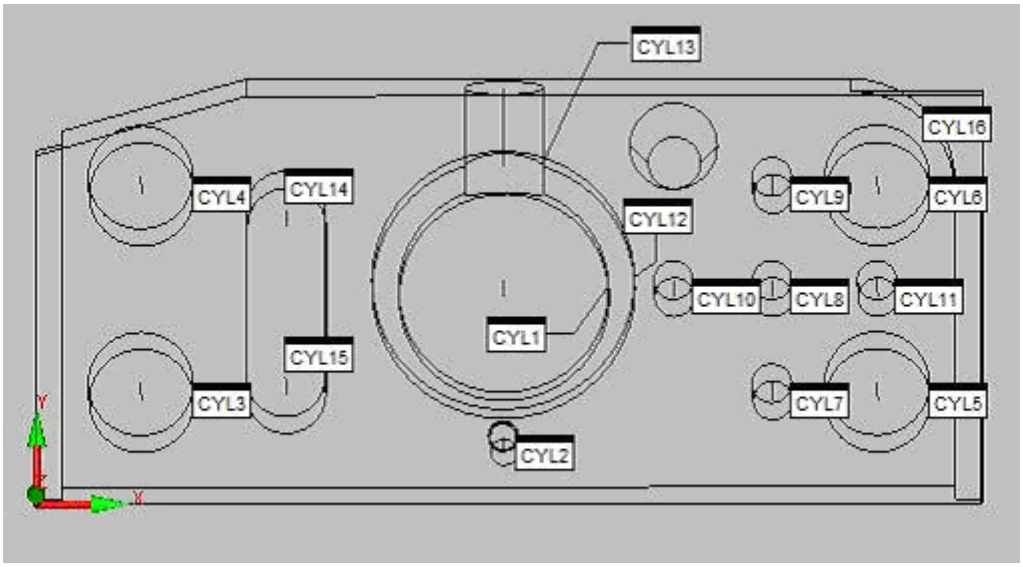
Notice that:

- Circles and arcs of the same diameter and vector are merged into a single feature. So, the top and bottom circles for the two center bores are merged into only two features, CIR1 and CIR13 (as were all the other features computed using the top and bottom circles and arcs). For the cone feature, however, since the diameters are different PC-DMIS creates two features (CIR12 and CIR14).
- A circle gets created for the back bore, CIR15. This is because the model was rotated slightly. If the view had remained entirely in a Z+ orientation, then CIR15 would not have been created.

Example 2: Box-Selecting Auto Cylinders with Wireframe Data

This example uses the Brown and Sharpe wireframe block (Bsbwire.igs) with auto cylinders:

If you do the same as in Example 1, but you use auto cylinders instead, you will get something like this:

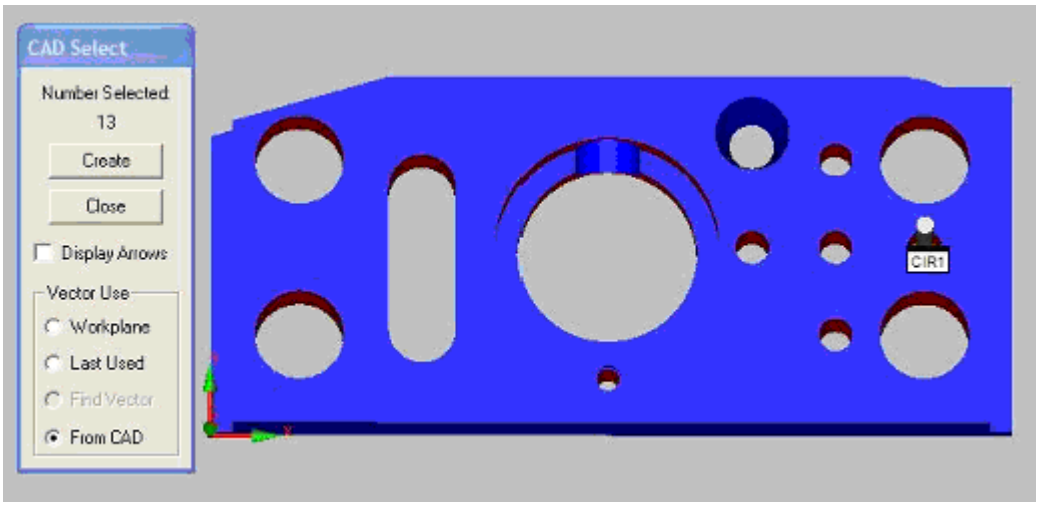


Notice that no cylinder was created at the cone feature because a cylinder must have length and must be the same diameter.

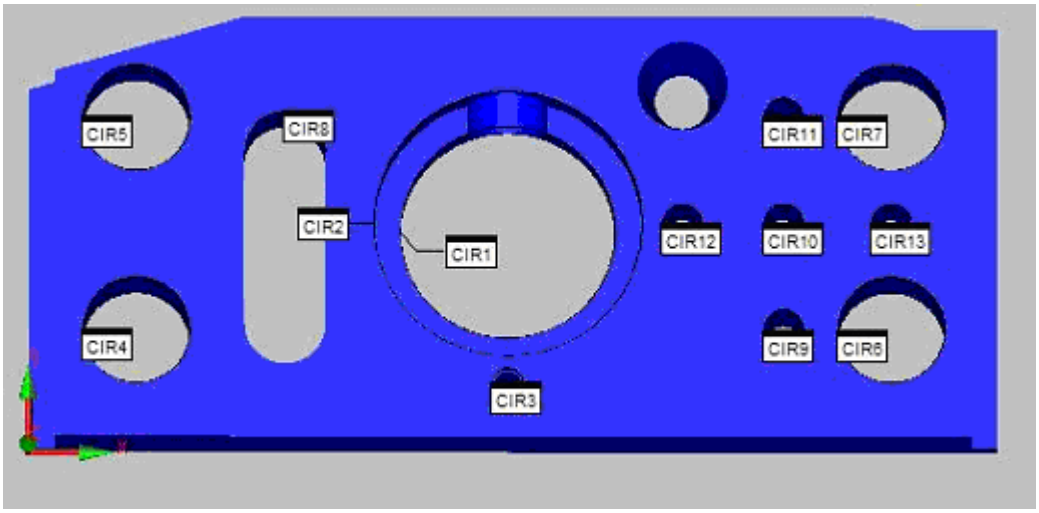
Example 3: Box-Selecting Auto Circles or Cylinders with Surface Data

This example uses the Brown and Sharpe solid block (Bsbsolid.igs) with auto circles and cylinders.

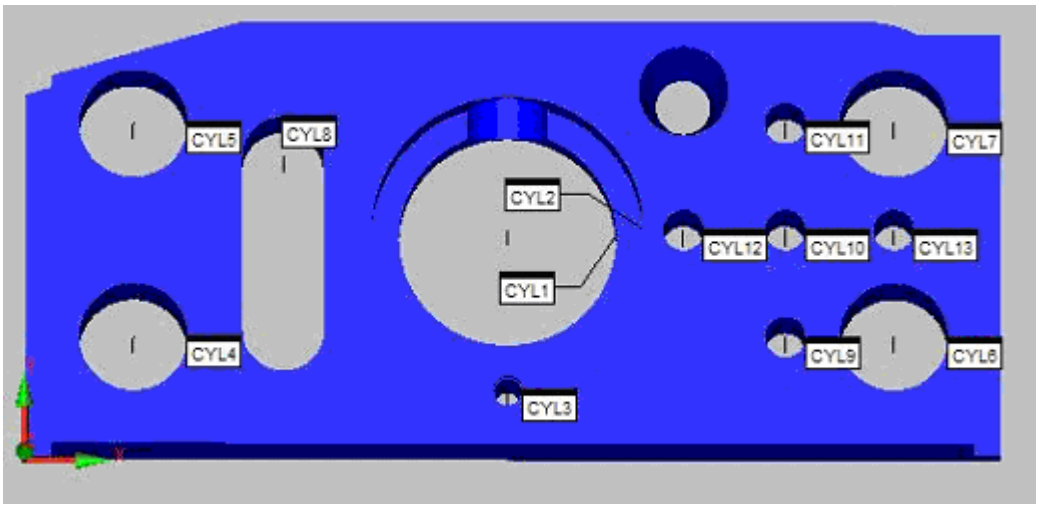
If you box-select to create features with a model that only has surface data (no wireframe data), the model needs to be slightly rotated in the view so that the desired surfaces are visible, like this:



Once you click **Create** and PC-DMIS generates the features from the selected objects, you get something like this:



Example showing circle features.



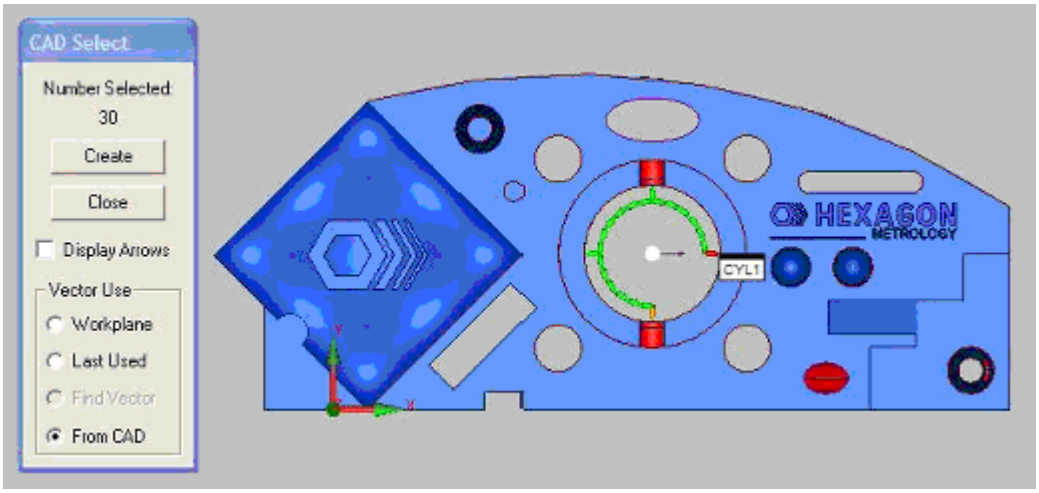
Example showing cylinder features.

Notice that because of the tilting, you do not get a circle or cylinder feature at one end of the slot, nor at the top right outer diameter (OD) cylinder.

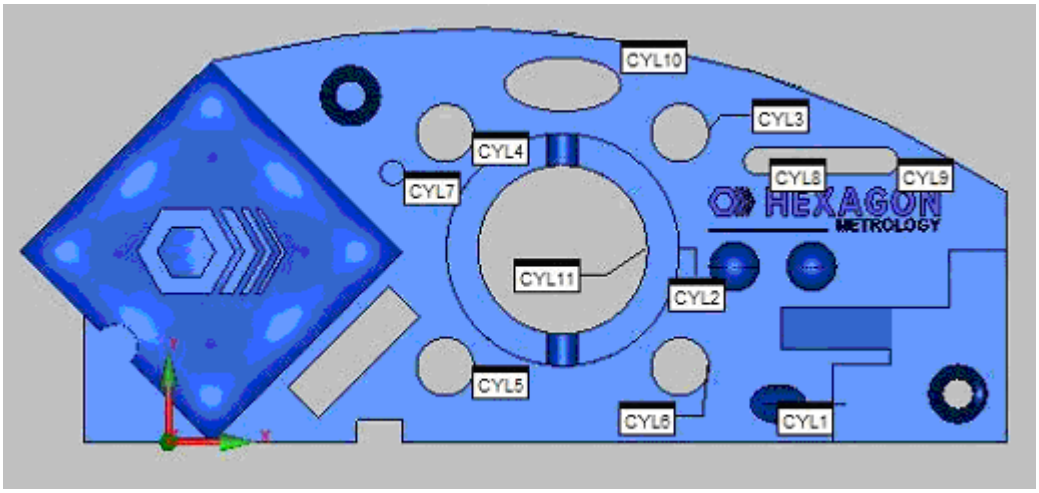
Example 4: Box-Selecting Auto Cylinders with Both Wireframe and Surface Data

This example uses the Hexagon model that ships with PC-DMIS (Hexblock_Wireframe_Surface.igs) with Auto Cylinders.

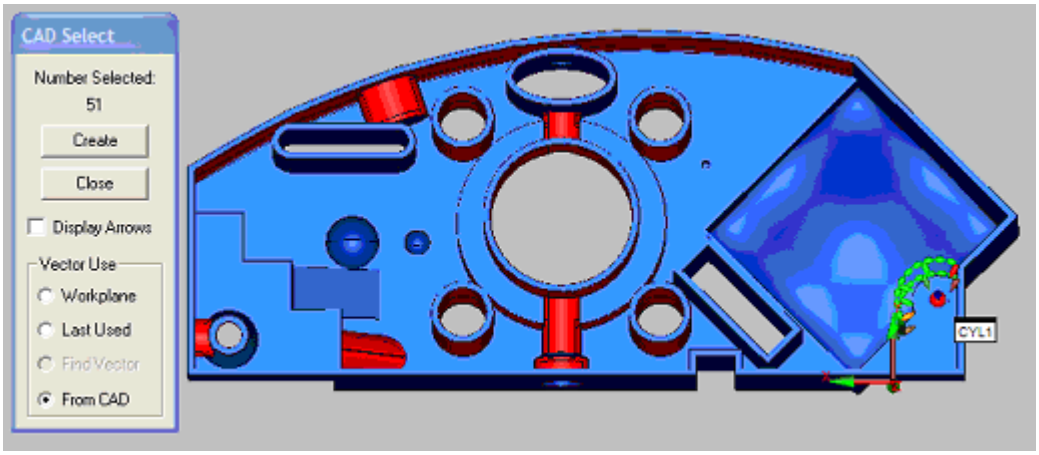
Put the model into a Z+ orientation and box-select the entire model.



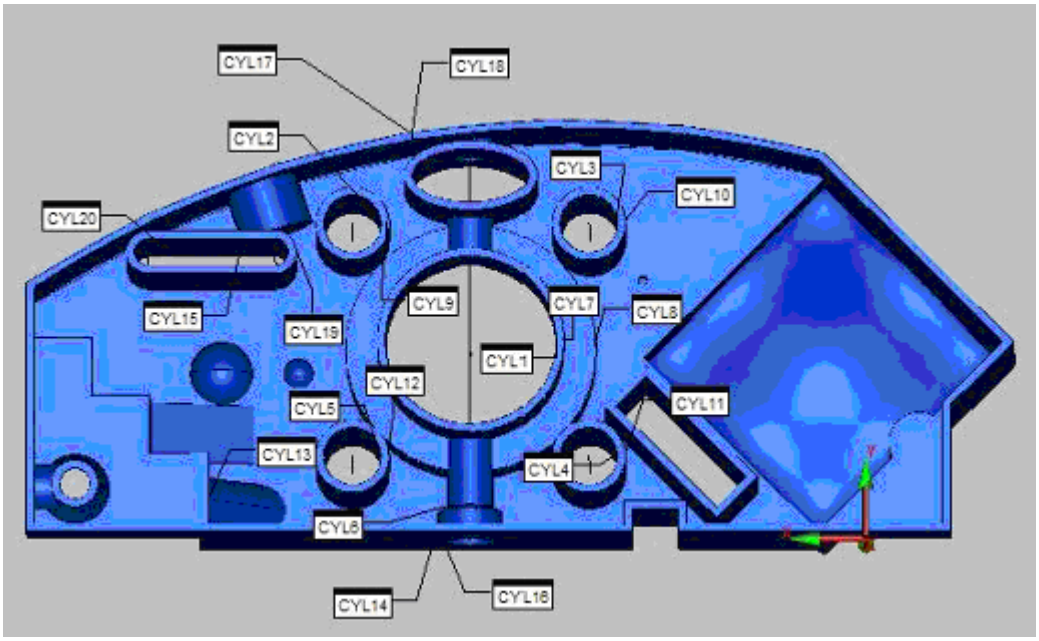
Click **Create** to see the following:



If you flip the part over to a Z- orientation, tilt the part slightly, and then box-select, PC-DMIS shows something like this:



Click **Create** to display something like this:



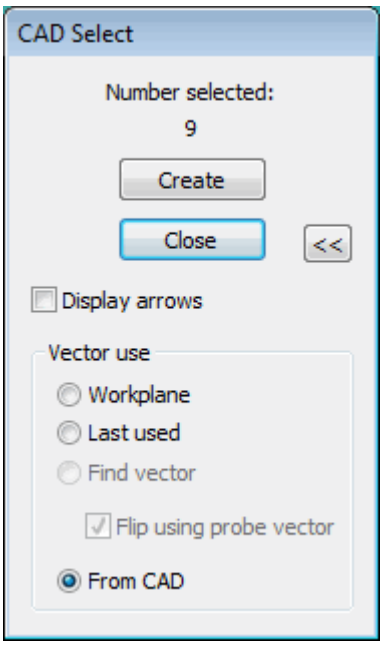
Notice that PC-DMIS generated both an inner diameter (ID) and outer diameter (OD) cylinder feature for most of the cylinders on the part, such as the top right CYL3 and CYL10.

With point features, if the model has defined the points as CAD elements, you can box-select them with QuickFeatures. For more information, see "Creating QuickFeatures".

Understanding the CAD Select Dialog Box

You can use the **CAD select** dialog box to create multiple Auto features at once. The **CAD Select** dialog box appears after you drag a box (box-select) on a CAD model. You must have the **Auto Feature** dialog box open to a supported feature type (for the supported feature types, see the "Box Selecting to Create Multiple Auto Features" topic), and you must box-select CAD elements of the feature type displayed in the dialog box. If you meet these conditions, the **CAD select** dialog box displays the number of selected CAD elements that match the selected Auto feature type.

For points or cylinder CAD elements, once the **CAD Select** dialog box opens, you can deselect and select highlighted elements if you click on them. For circle elements, however, once you deselect a circle, you cannot click on it again to re-select it.



CAD Select Dialog Box

Create - This button creates Auto features of the selected type from the selected CAD elements (currently for points, circles, cylinders, or cones). PC-DMIS closes the **CAD Select** dialog box, and from each box-selected element, PC-DMIS generates the appropriate feature. The **Vector use** area in the advanced portion of the dialog box determines the vector method.

Close - Closes this dialog box and cancels the box selection.

>> or **<<** - Shows or hides the advanced items on the dialog box. These advanced items control the features' vectors and are most often needed for imported DES models:

Display arrows - This check box displays or hides colored arrows showing the direction of vectors used by the methods in the **Vector Use** area.

The **Vector use** area lets you choose the methods PC-DMIS should use to determine the vectors for the nearly-created Auto features.

- **Workplane** - This method uses the vector of the currently active workplane as the vector for each individual feature.
- **Last used** - This method uses the last vector that was placed in the dialog box for the Auto feature. This lets you specify one vector to be used for all the selected features.
- **From CAD** - This method uses the vector that is specified by the CAD feature. This method becomes available if vector data is available for each feature.
- **Find vector** - This method finds the vector using the CAD surface data that is closest to the feature. This method is only available if surface data is available.

- **Flip using probe vector** - During the CAD import process, some types of CAD (usually IGES) may have some surface normal vectors that incorrectly point into the part instead of out of the part. This method flips the selected features' vectors so that they point out of the surface, using the probe vector to help indicate the correct vector direction. That option becomes available when you box-select feature types that may have surfaces with incorrect normal vectors.

'Multi-Clicking' and 'Single-Clicking' Auto Feature Creation

In older versions of PC-DMIS, you often had to provide multiple mouse clicks for PC-DMIS to generate an Auto feature. Over the years, PC-DMIS has continually increased its capability to select auto features on the screen with less and less mouse clicks.

The following table lists the Auto features you can select with a single mouse click in both Curve Mode and Surface Mode. It also shows the number of clicks required for features that do not support single-click selection.

Auto Feature Type	Curve Mode	Surface Mode
Angle Point	No (2 Clicks)	Yes
Circle	Yes	Yes
Cone	Yes	Yes
Corner Point	Yes	Yes
Cylinder	Yes	Yes
Ellipse	Yes	Yes
Edge Point	No (2 Clicks)	Yes
High Point	Yes	Yes
Line	No (2 Clicks)	No (2 Clicks)
Notch	Yes	Yes
Plane	Yes	Yes
Polygon	Yes	Yes
Round Slot	Yes	Yes
Square Slot	Yes	Yes
Sphere	Yes	Yes

Surface Point	Yes	Yes
Vector Point	Yes	Yes

Using Single-Click Mode

- **Angle Point Features**

  With the **Auto Feature** dialog box open, click once near but not on the intersection of the two surfaces. In Surface Mode, PC-DMIS generates the point at the angle where the two surfaces meet.

- **Circle Features**

  With the **Auto Feature** dialog box open, click once on the planar surface that contains the circle for a hole, on the feature's wall, or on a cylindrical end of a stud.

  - The edge closest to the mouse-click determines whether the circle feature is a hole or a stud. (See Note B.) However, due to how the CAD is sometimes defined, PC-DMIS cannot always determine this on its own.
  - If the circle has a length (depth) like a cylinder, then PC-DMIS uses the circle closest to where you clicked the mouse to define the center position.
  - PC-DMIS automatically fills in the dialog box and determines the circle's vector.

  For details on how to create single-point circle features with PC-DMIS Portable, see "Creating Single-Point Circle Features" topic in the PC-DMIS Portable documentation.

- **Cone Features**

  With the **Auto Feature** dialog box open, click once on the planar surface that contains the cone's hole, on the feature's wall, or on a conical end of a cone.

  - The edge closest to the mouse-click determines whether the cone feature is an inner cone or an outer cone. (See Note B). However, due to how the CAD is sometimes defined, PC-DMIS cannot always determine this on its own.
  - The vector of the cone points away from its apex.
  - PC-DMIS automatically fills in the dialog box and determines the center point. Inner cones use the larger radius for the center point. Outer cones

use the smaller radius. PC-DMIS does this to avoid shank collisions when measuring the cone.

- **Cylinder Features**

  With the **Auto Feature** dialog box open, click once on the planar surface that contains the cylinder, on the feature's wall, or on a cylindrical end of an outer cylinder.

  - The edge closest to the mouse-click determines whether the circle feature is a hole or a stud. (See Note B). However, due to how the CAD is sometimes defined, PC-DMIS cannot always determine this on its own.
  - PC-DMIS automatically fills in the dialog box and determines the cylinder's vector. It uses the center location of the cylinder end closest to where you mouse-clicked.

- **Edge Point Features**

  With the **Auto Feature** dialog box open, click once anywhere. PC-DMIS automatically populates the dialog box.

  - If you are using Surface Mode, PC-DMIS generates the edge on the nearest edge where you clicked.
  - If you are in Wireframe Mode, PC-DMIS selects only the edge. A second click is required to create the edge point on the wire.

- **Ellipse Features**

  Same as Round Slot.

- **Notch Features**

  With the **Auto Feature** dialog box open, click once close to the back edge of the notch. When you single-click to create a notch feature, the software always defines it as an inner notch. PC-DMIS automatically fills in the dialog box.

- **Polygon Features**

  With the **Auto Feature** dialog box open, click once on a polygon feature that contains five or more sides. PC-DMIS automatically fills in the dialog box and determines the starting side by the edge closest to the mouse click.

- **Corner Point Features**

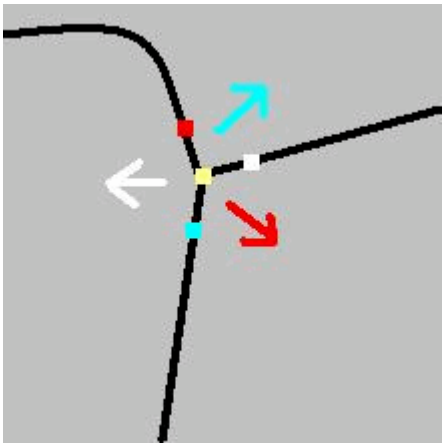
  With the **Auto Feature** dialog box open, click once near an edge that contains the corner point.

  - The closest edge to the mouse click determines the closest end point to the mouse click. This end point becomes the corner point.

- PC-DMIS automatically fills in the dialog box.

How it Works:

- PC-DMIS finds adjacent edges relative to the first edge. It tests each edge to see if it has an end point coincident to the corner point. If so, then it finds a point on that edge very near the corner point. This continues until two edges (and two points) are found that have unique vectors (ideally perpendicular) to each other and the first edge.
- Once PC-DMIS has three points near the corner point, these three points are all on different edges. Using the two different points near the corner point, and the corner point, PC-DMIS computes three planes. For example, in this image, the YELLOW corner point and the RED and WHITE points create a plane whose vector is the BLUE arrow. Similarly, the YELLOW, WHITE, and BLUE points give us the RED arrow and the YELLOW, BLUE, and RED points give us the WHITE arrow.



- **Round Slot Features**

With the **Auto Feature** dialog box open, for an inner round slot, click once on the planar surface that contains the rounded edge or the feature's wall. For an outer round slot, click on the top edge of the rounded end, away from where the curve ends, and the planar sides begin or on the feature's wall. (See Note A).

- The edge closest to the mouse click determines whether the feature is an inner round slot or an outer round slot. (See Note B). However, due to how the CAD is sometimes defined, PC-DMIS cannot always determine this on its own.
- PC-DMIS automatically fills in the dialog box and determines the slot's vector and orientation.

- **Square Slot Features**

With the **Auto Feature** dialog box open, for an inner square slot, click once on the planar surface near any edge of the slot or on the feature's wall. For an outer square slot, click on any top edge or on the feature's wall. (See Note A).

- The edge closest to the mouse click determines whether the feature is an inner square slot or an outer square slot. (See Note B). However, due to how the CAD is sometimes defined, PC-DMIS cannot always determine this on its own.
- PC-DMIS automatically fills in the dialog box and determines the slot's vector and orientation.

- **Sphere Features**

  With the **Auto Feature** dialog box open, click once on the sphere's edge. PC-DMIS automatically fills in the dialog box.

- **Surface Point Features**

  With the **Auto Feature** dialog box open, click once anywhere. PC-DMIS automatically fills in the dialog box.

- **Vector Point Features**

  With the **Auto Feature** dialog box open, click once anywhere. PC-DMIS automatically fills in the dialog box.

  - If you use Curve Mode, PC-DMIS snaps the point to the nearest wire.
  - If you use Surface Mode, PC-DMIS generates the point where you clicked.

By default, PC-DMIS attempts to interpret a single left mouse click on the part model and generate the Auto feature from that input. It momentarily displays the probe on the screen and then inserts the feature information gathered from the mouse click into the open **Auto Feature** dialog box. If, for some reason, this fails, PC-DMIS automatically switches to the older Multi-Click Mode of input (see the Multi-Click Mode information below).

If you want the animated probe to appear permanently on the screen when you click on the part model, you can use the PC-DMIS Settings Editor to locate the DisplayProbeForJustOneMoment entry in the **Option** section, and change its value to FALSE.

Temporarily Switching to Multi-Click Mode

You can temporarily switch to the Multi-Click Mode of input. To do this, with the **Auto Feature** dialog box open, take two clicks within three pixels of each other. When you take the second mouse click, PC-DMIS switches to Multi-Click Mode and draws the cross-hairs at the location of the second mouse click. This click becomes the first mouse click of the Multi-Click Mode. PC-DMIS then waits for the remaining mouse clicks needed to generate

the feature. Once the feature is generated, PC-DMIS switches back to the Single-Click Mode.

Using Multi-click Mode

Once you have switched to Multi-Click Mode, follow the instructions on the Status bar which walks you through the hits you need to take.

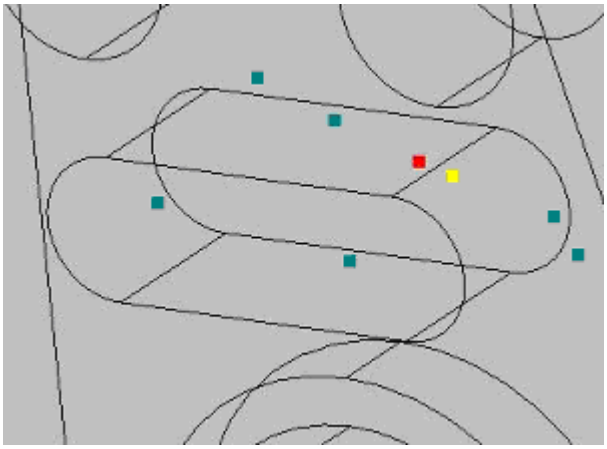
Permanently Switching to Multi-Click Mode

If you want to turn off the Single-Click Mode functionality permanently, access the PC-DMIS Settings Editor, and set the SingleClickCadSelectionDisabled entry (in the **AutoFeatures** section) to TRUE. When you restart the software, PC-DMIS enables Multi-Click Mode.

Note A

In this image:

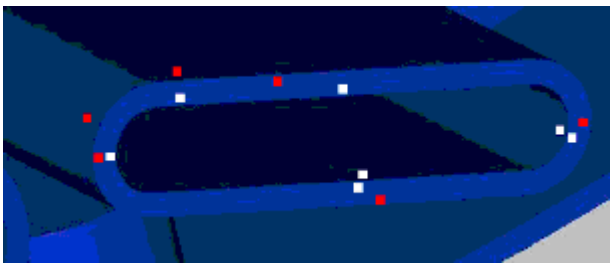
- The GREEN marks should work because the line/curve (or edge if it is a surface) that is closest to the marks is a line/curve that is in the plane that defines the slot.
- The RED mark will not work because the line (or edge if it is a surface) that is closest to the mark is a line/curve that is not in the plane that defines the slot.
- The YELLOW mark will work if the surface is a cylinder.



Note B

In this image, showing a flipped over, Z- orientation of the Hexagon model:

- The RED marks will produce an Outer Diameter (OD) feature.
- The WHITE marks will produce an Inner Diameter (ID) feature.



Creating QuickFeatures

For details on how to implement QuickFeatures in PC-DMIS Laser, see "Implementation of QuickFeatures in PC-DMIS Laser" in the PC-DMIS Laser documentation.

For details on how to implement QuickFeatures in PC-DMIS Vision, see "Implementing QuickFeatures in PC-DMIS Vision CAD View" and "Implementing QuickFeatures in PC-DMIS Vision Live View" in the PC-DMIS Vision documentation.

For details on how to use the Quick Scan functionality to create a Linear Open scan from a polyline or surface, see "Creating Quick Scans" in the "Scanning" chapter in the PC-DMIS CMM documentation.

About QuickFeatures

You can use the QuickFeatures functionality to create Auto features from gestures. These gestures are usually click or click-and-drag operations with key combinations. You perform the gestures with the mouse on the CAD model without any menu options or dialog boxes. Also, if models contain point features, and PC-DMIS is in Curve Mode, you can use QuickFeatures to box-select and create multiple point features at a time. For more information, see "Creating Vector Point Features by Box-Selecting" below.

When you create QuickFeatures, dialog boxes with feature lists (such as for constructions or dimensions) can remain open. As you add the new features into the measurement routine, PC-DMIS adds them into the feature list and automatically selects them for the current operation.

For information about the default settings and on how to use the Measurement Strategy Editor to change default settings, see "Using the Measurement Strategy Editor" topic.
By default, when you create a feature from the QuickFeature functionality and there is no other dialog box displayed, the measurement strategy widget appears. This widget lets you change a feature's main parameters. For more information, see "Using the Measurement Strategy Widget" in the next main topic.

Requirements

Your CAD model can contain either surface data or wireframe data. However, due to limitations in wireframe data, if your model is a wireframe-only model, PC-DMIS cannot create these features using the QuickFeature functionality:

- Vector Point
- Angle Point
- Plane
- Sphere

Also, QuickFeatures only function with contact Auto features.

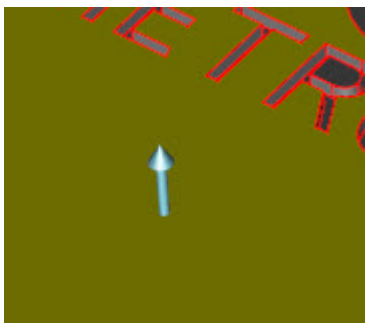
General Process to Create QuickFeatures

You can choose to work with or without the **Auto Feature** dialog box open. However, the procedure below assumes your **Auto Feature** dialog box is not open. Also, the measurement strategy widget does not appear if you use the **Auto Feature** dialog box.

1. In the Edit window, click to define where to insert the new feature.
2. In the Graphic Display window, position your mouse pointer over the CAD element.
3. For point features (Vector, Edge, Angle, Corner), press Ctrl + Shift and click on the CAD element to create the feature.

    - Vector Point

      To create a vector point, press Shift + Ctrl and hover the pointer over the surface. An arrow that represents the surface vector (light blue) appears. Click and release the mouse button to create the feature.

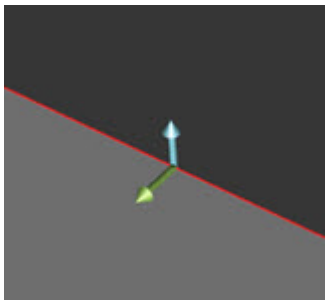
      

      Example showing the vector point selection.

      You can also quickly create vector points by box-selecting them. See "Create Vector Point Features by Box-Selecting" below.

- Edge Point

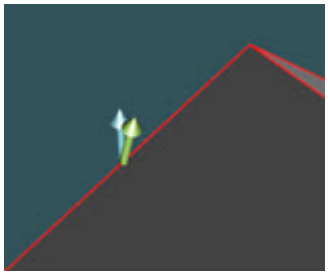
  To create an edge point, press Shift + Ctrl. Hover the pointer near a surface edge where the angle between the two surfaces is 90 degrees. Arrows that represent the surface vector (light blue) and the edge vector (green) appear. Click and release the mouse button to create the feature.

  Example showing the edge point selection.

- Angle Point

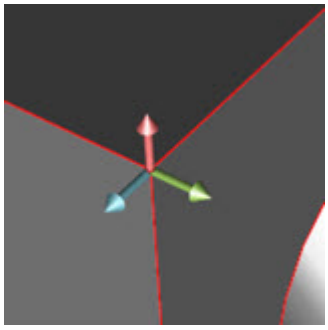
  To create an angle point, press Shift + Ctrl and hover the pointer near a surface edge where the angle between the two surfaces is NOT 90 degrees. PC-DMIS draws arrows to highlight the angle point (the light blue arrow represents the surface 1 normal vector, and the light green arrow represents the surface 2 normal vector). Click and release the mouse button to create the feature.

  Example showing the angle point selection.

- Corner Point

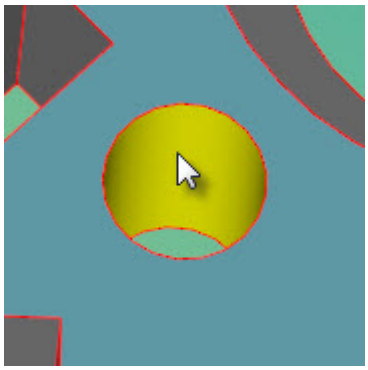
  To create a corner point, press and hold Shift + Ctrl and hover the pointer over a corner. PC-DMIS draws arrows to highlight the corner point. Click and release the mouse button to create the feature.

  Example showing the corner point selection.

If you enable a 3D grid on the Graphic Display window, the software snaps Vector, Edge, and Angle points to the nearest intersection on the grid. For more information, see "Adding a 3D Grid" in the "Editing the CAD Display" chapter.

4. For Plane, Circle, Ellipse, Cylinder, Cone, or Sphere, press Shift and click on the CAD element to create the feature.



Example of a highlighted 3D cylinder.

For Line, Round Slot, Square Slot, Notch Slot, and Polygon feature types, follow the instructions in their respective topics below. These feature types, and other information not covered in this general procedure, are discussed below.

5. By default, the measurement strategy widget appears. You can use this widget to modify common feature properties. For information on the measurement strategy widget, see "Using the Measurement Strategy Widget" in this chapter.

6. Continue to create any other QuickFeatures you need. With each new feature you create, PC-DMIS automatically applies and accepts the previous feature. This also works with multi-QuickFeatures below.

7. Once you're done creating your features, click the green **Apply** button to accept the final feature and close the measurement strategy widget.

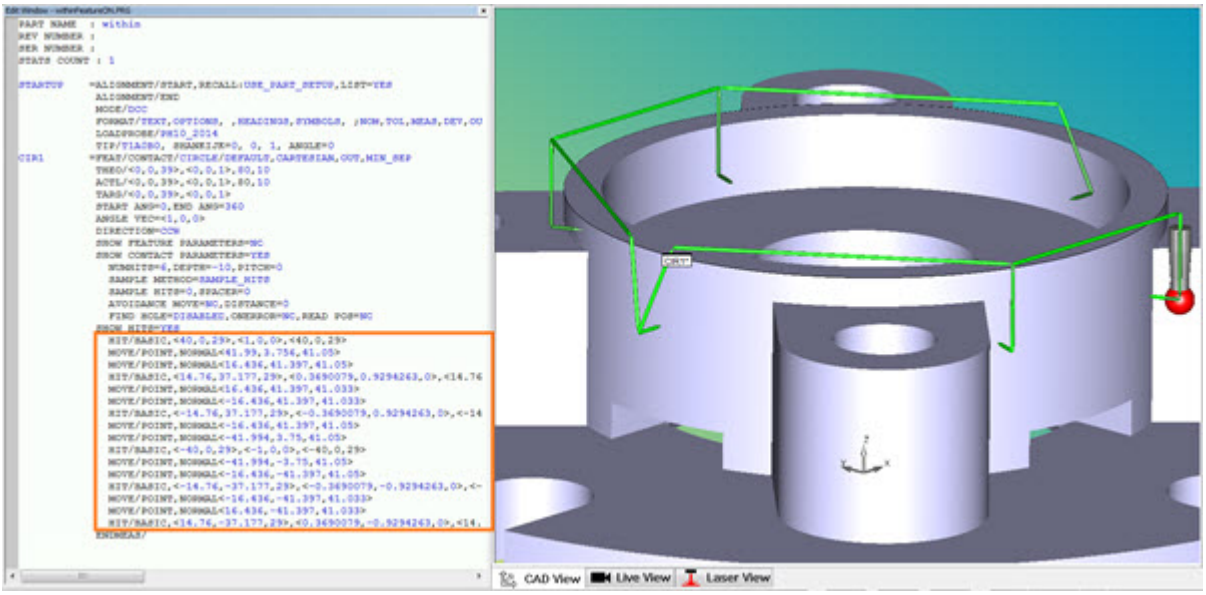
Additional Information on QuickFeatures

- You can select colinear or coplanar elements. To do this, press and hold Shift, hover the pointer over a line or plane, click and hold the mouse button, and move the pointer to a colinear or coplanar element. Once all the elements are highlighted, release the mouse button to create the feature from both elements. For an example, see "Creating Line Features" above.

- You can toggle what features are executed within the Graphic Display window. To do this, press Alt and click on the feature label to toggle its marked state. For information on how to mark features, see "Marking Commands for Execution" in the "Editing a Measurement Routine" chapter.

- You can have PC-DMIS automatically generate safe moves inside the QuickFeature. To do this, select **Operation | Graphic Display Window | Clearance Move | Within feature** prior to creating the features. If this menu option is enabled, moves (between

sample hits and among hits in the same list) are automatically computed and drawn as lines.



Example showing the Within feature option set to active.

For details on how to create clearance moves, see "Inserting Clearance Moves Automatically" in the "Inserting Move Commands" chapter.

- If you select **Operation | Graphic Display Window | Clearance Move | With Feature Creation**, PC-DMIS generates clearance moves between QuickFeatures. The **Auto Feature** dialog box must be closed.
- Automatic safe moves between features that use different probe tip angles are not supported. You must manually define these moves.
- PC-DMIS automatically updates the contents of an open **Auto Feature** dialog box. During feature creation, it retrieves the data from the feature on the CAD model.

  For a created feature, the dialog box updates to the feature you selected.

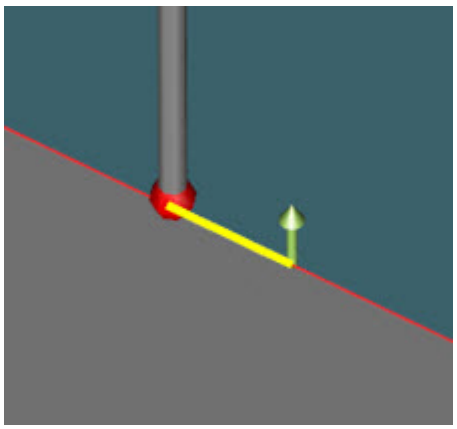
  For an edited feature, it only does this if the selected feature matches the edited one.

  In both cases, the default values for the feature come from entries.

Click the links below to show the descriptions of how to create each of the feature types.

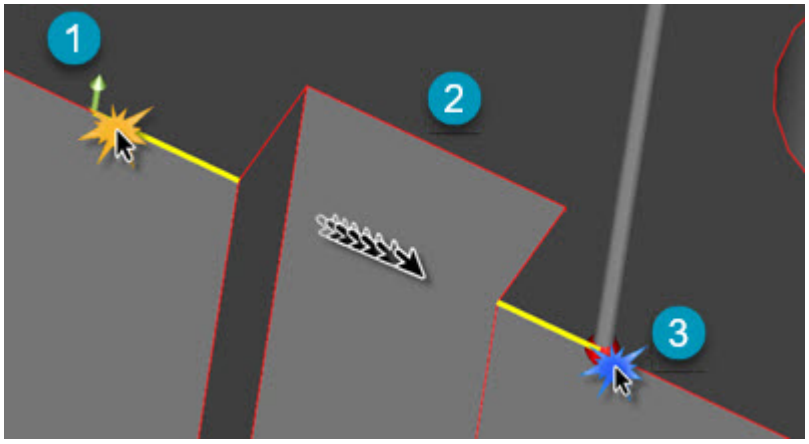
Creating Line Features

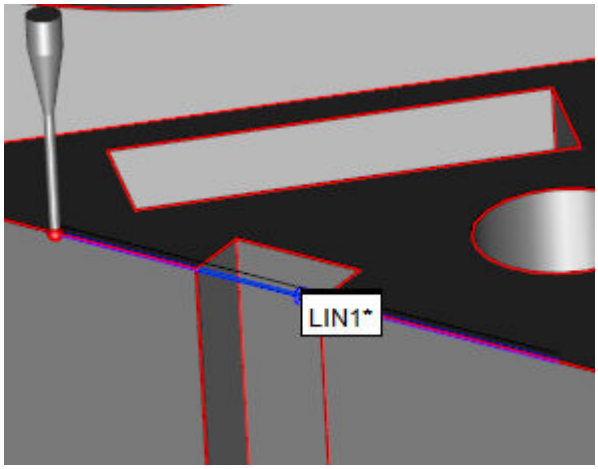
- To create a basic line feature, position the mouse pointer near a surface edge where the angle between the two surfaces is 90 degrees. Press and hold the Shift key on your keyboard. Click and drag the pointer for a short distance along the line. PC-DMIS starts to highlight the line. An arrow representing the edge vector (green) appears.



Drag to define the length of the line and release the mouse button. PC-DMIS places the line's start point where you click the button and the end point where you release the button.

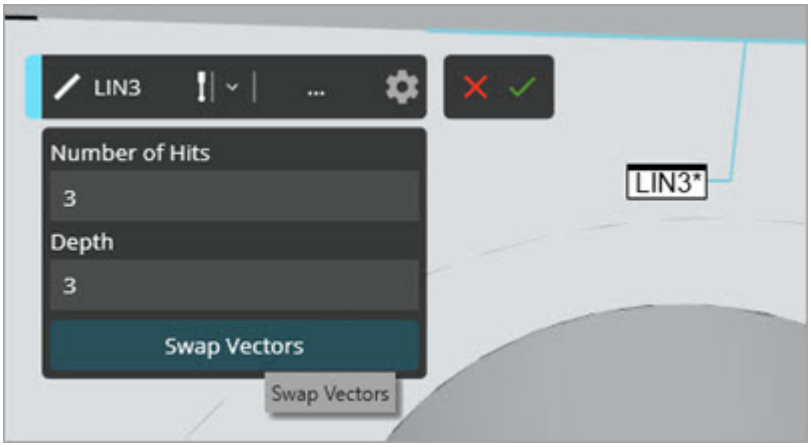
- To create a line feature from multiple, colinear, line elements, press and hold Shift, (1) hover the pointer over a line, click and drag the pointer for a short distance along the line to start to highlight it; then (2) move the pointer to another colinear element and drag to define the length of the line. Once the software highlights the elements and the line is the desired length, (3) release the mouse button to create the line feature from the line elements.





Example of a line feature constructed from two colinear elements.

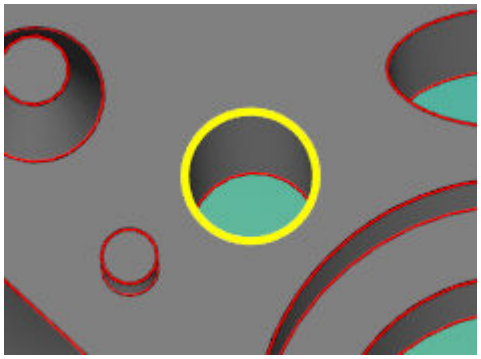
You can use the **Swap Vectors** button on the QuickFeatures Widget to flip the edge vector of the line.



Example of the Swap Vectors button on the QuickFeatures widget.

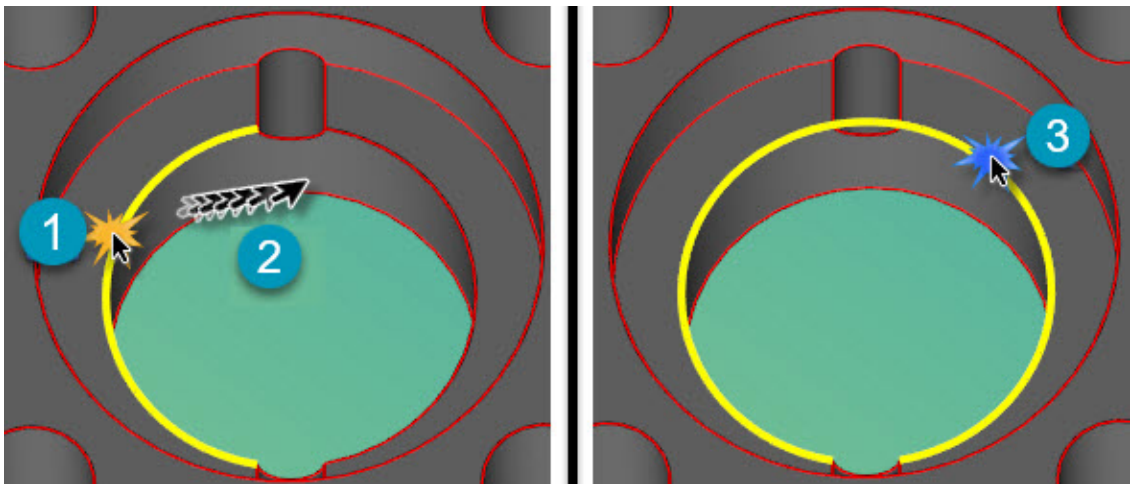
Creating Circle Features

- To create a basic circle feature, press and hold the Shift key, then hover the pointer over the circle's arc. Once it highlights the circle, click the circle to create the feature.



- To create a circle feature from two disjointed arcs:

  1. Press and hold the Shift key, hover the pointer over one arc, and then click to select the arc.
  2. Move the pointer to the other arc to highlight it.
  3. Once the entire circle is highlighted, release the mouse button to create the feature.

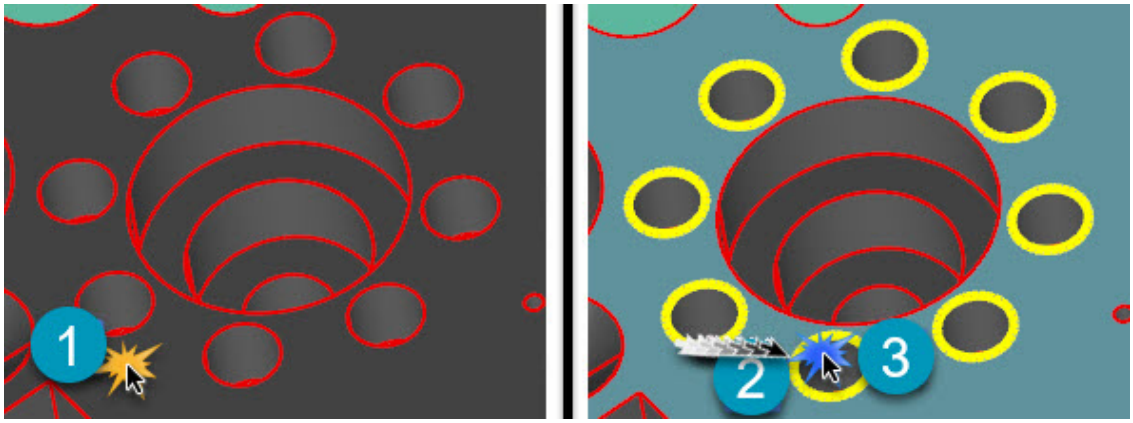


Creating Circular Multi-QuickFeatures

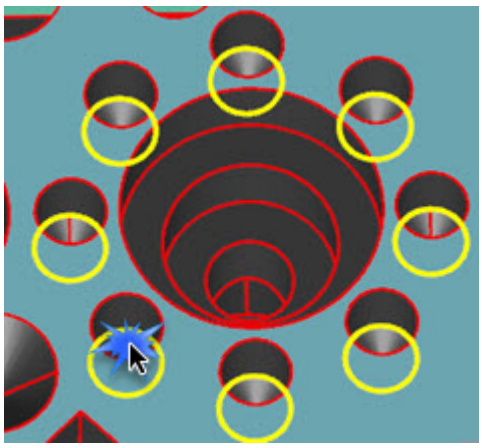
This procedure creates circular features (circles or cylinders) of the same size, such as a bolt hole pattern:

- To create multiple circular features:

  1. Select one or more surfaces, and then press and hold the Shift key.
  2. Hover the pointer over one circular feature to highlight all circular features of that diameter on those surfaces.
  3. Once all the features are highlighted, click the feature to create the highlighted circular features.



- You can also create multiple circular features from countersunk and counterbored holes. To do this, follow the instructions in the preceding paragraph, but select the circular features just below the surface.



The counterbore and countersink algorithm works better with models that natively contain surface topology, such as these model types:

- ACIS
- CATIAv5
- CATIAv6
- Creo
- Inventor
- JT
- NX
- Parasolid
- Solid Edge
- SolidWorks
- STEP

IGES files do not work as well, because they do not contain surface topology.

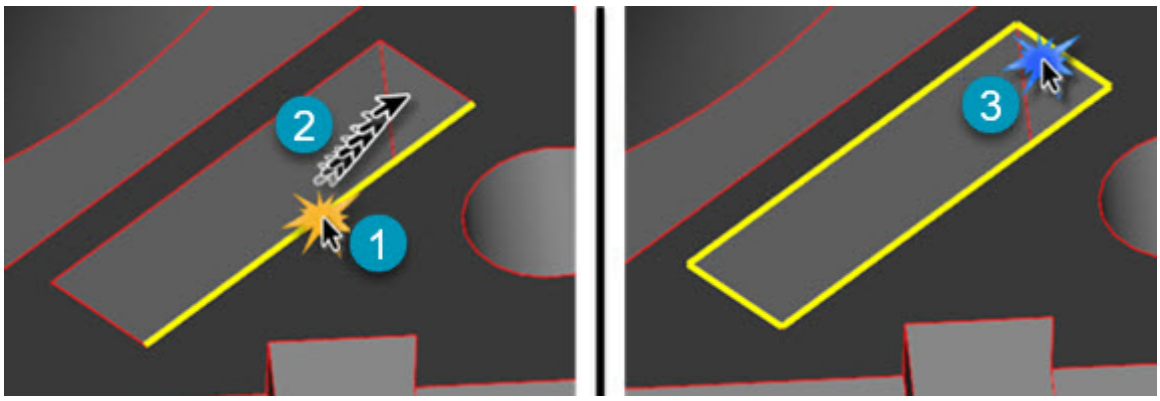
Multi-QuickFeatures work with up to 500 circular features per surface. If you have a surface with more than 500 circular features, Multi_QuickFeatures does nothing. In that case, you need to use the box-select method to create the features. For information, see "Box Selecting to Create Multiple Auto Features".

Creating Square Slot Features

To create a square slot:

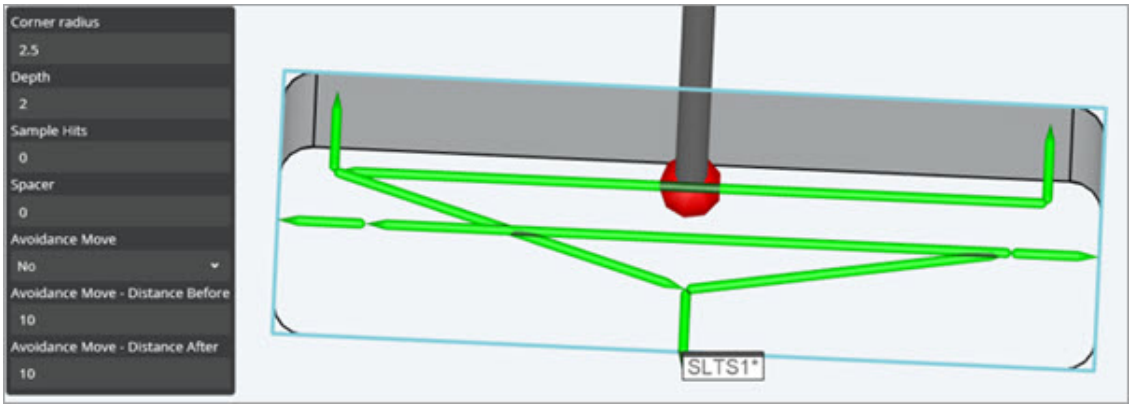
1. Press and hold Shift, and then hover the pointer over one edge of the slot, click and drag the pointer for a short distance along the edge to highlight it.
2. Move the pointer to an adjacent edge.

3. Once the entire slot is highlighted, release the mouse button to create the feature.

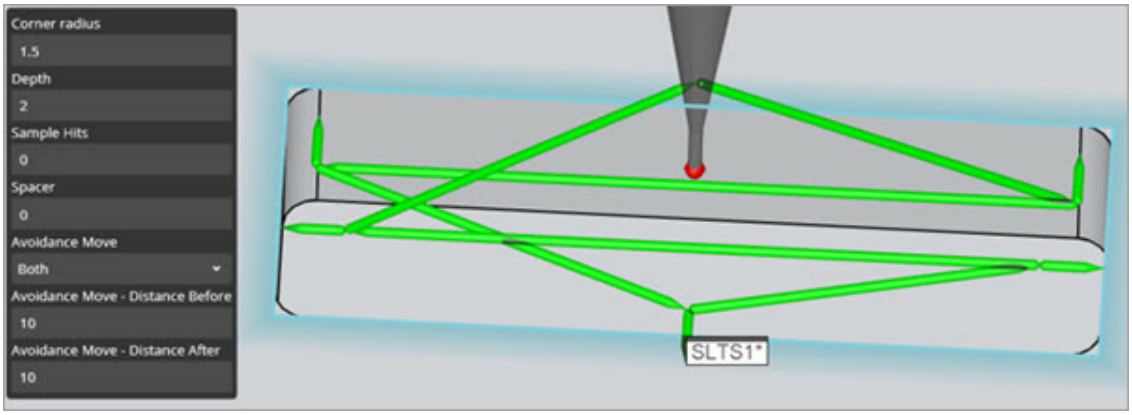


If the slot corner has a radius, PC-DMIS detects it and applies one of these rules:

If the detected blend radius is less than or equal to the currently loaded probe's active tip radius, PC-DMIS sets the auto-slot's corner radius value equal to the active tip radius + 1 mm.



If the detected blend radius is greater than the currently loaded probe's active tip radius, PC-DMIS sets the auto-slot's corner radius equal to the detected blend radius value.

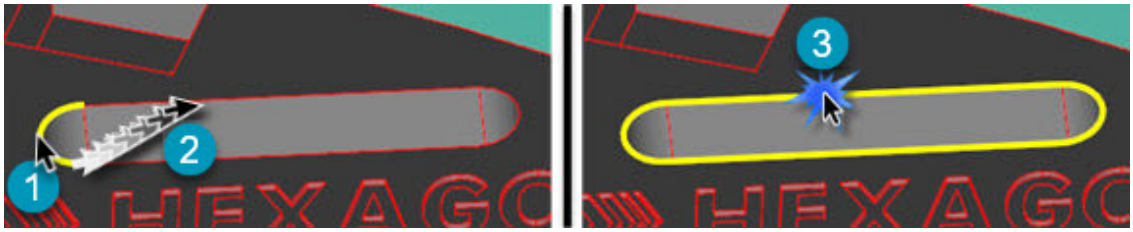


To create square slots that are on a non-planar surface, repeat the above steps but hover the pointer over the planar surface sides of the square slot instead of the edges.

Creating Round Slot Features

- To create a round slot:

  1. Press and hold Shift, and then hover the pointer over one of the circular ends of the slot and click and drag the pointer a short distance along the curve to highlight it.
  2. Move the pointer to a straight side.
  3. Once the entire slot is highlighted, release the mouse button to create the feature.

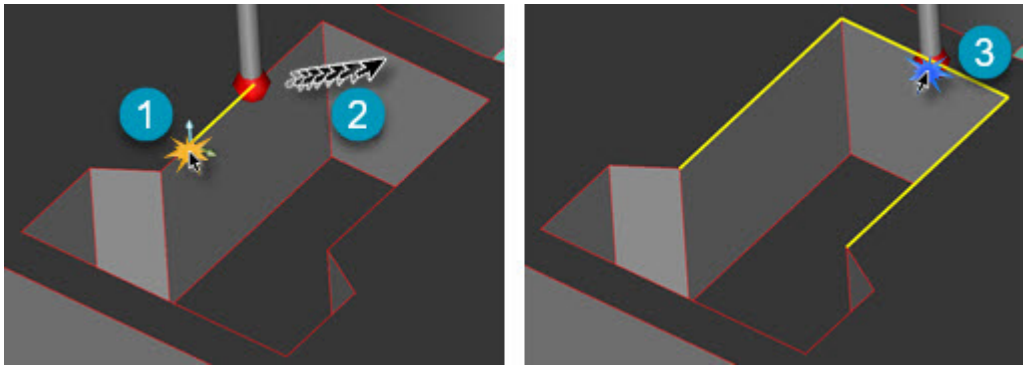
  You can also start by hovering over a straight side, click and drag the pointer a short distance to highlight it. Then move the pointer to a circular end. Once the slot is highlighted, release the mouse button to create the feature.

- To create round slots that are on a non-planar surface, repeat the above steps but hover the pointer over a cylindrical end of the round slot instead of a circular edge.

Creating Notch Slot Features

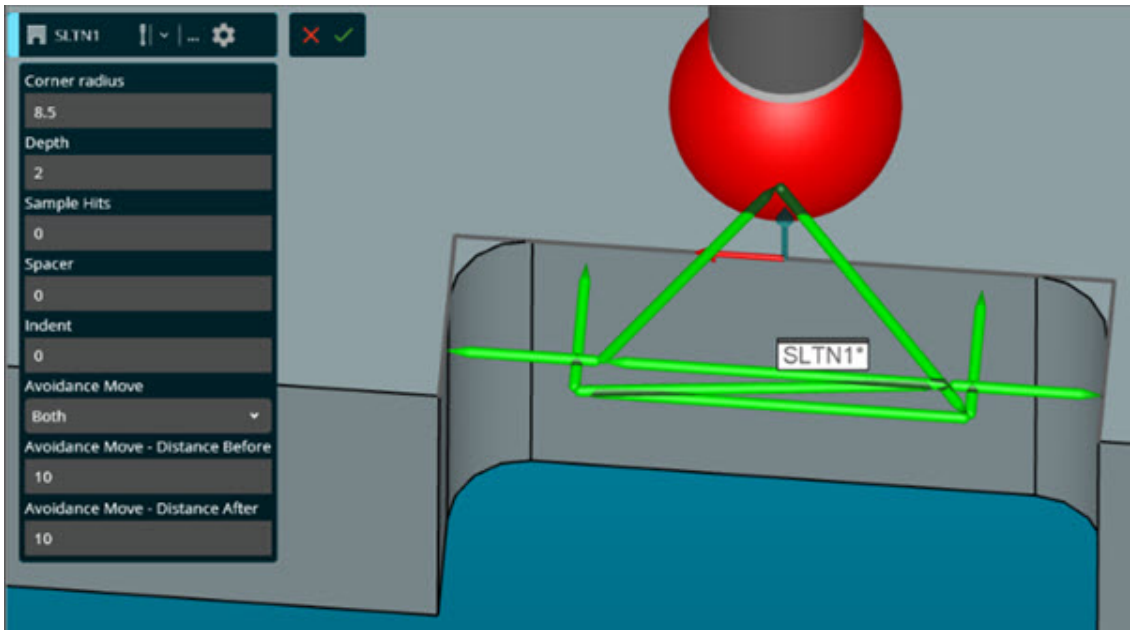
To create a notch slot:

1. Press and hold Shift, and then hover the pointer over one of the legs of the notch.
2. Click and drag the pointer a short distance along the leg to highlight it (1).
3. Move the pointer to an adjacent side (2).
4. Once the entire notch is highlighted, release the mouse button (3).

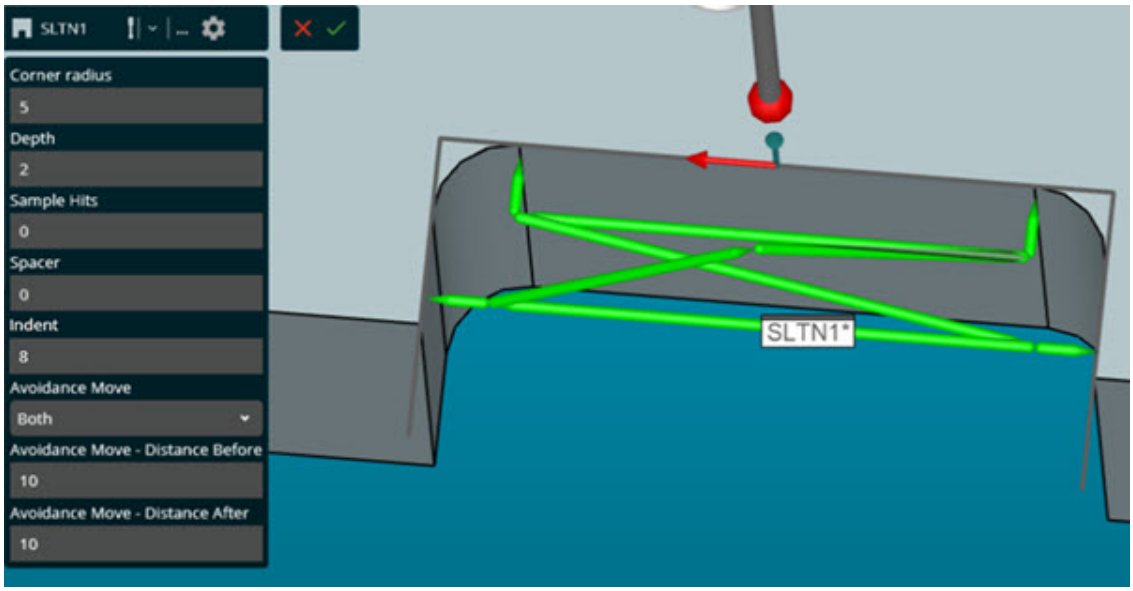


If the notch corner has a radius, PC-DMIS detects it and applies one of these rules:

- If the detected blend radius is less than or equal to the currently loaded probe's active tip radius, PC-DMIS sets the auto-slot's corner radius value equal to the active tip radius + 1 mm.



- If the detected blend radius is greater than the currently loaded probe's active tip radius, PC-DMIS sets the auto-slot's corner radius equal to the detected blend value.



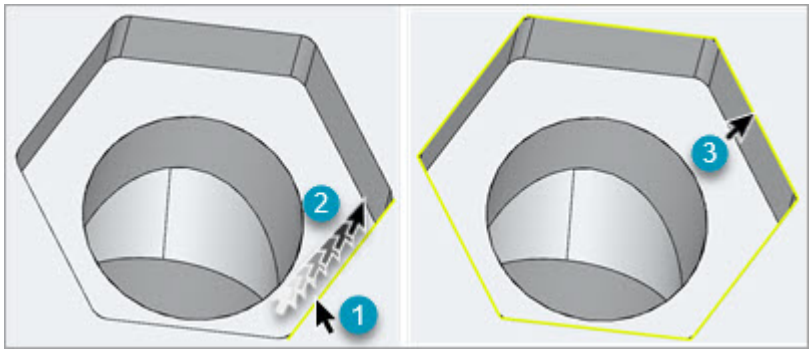
Regardless if the notch has a corner radius or not, the `INDENT` value is always defined to position the hits mid-way along the flat portion of the side walls of the notch.

Creating Polygon Features

To create a polygon feature:

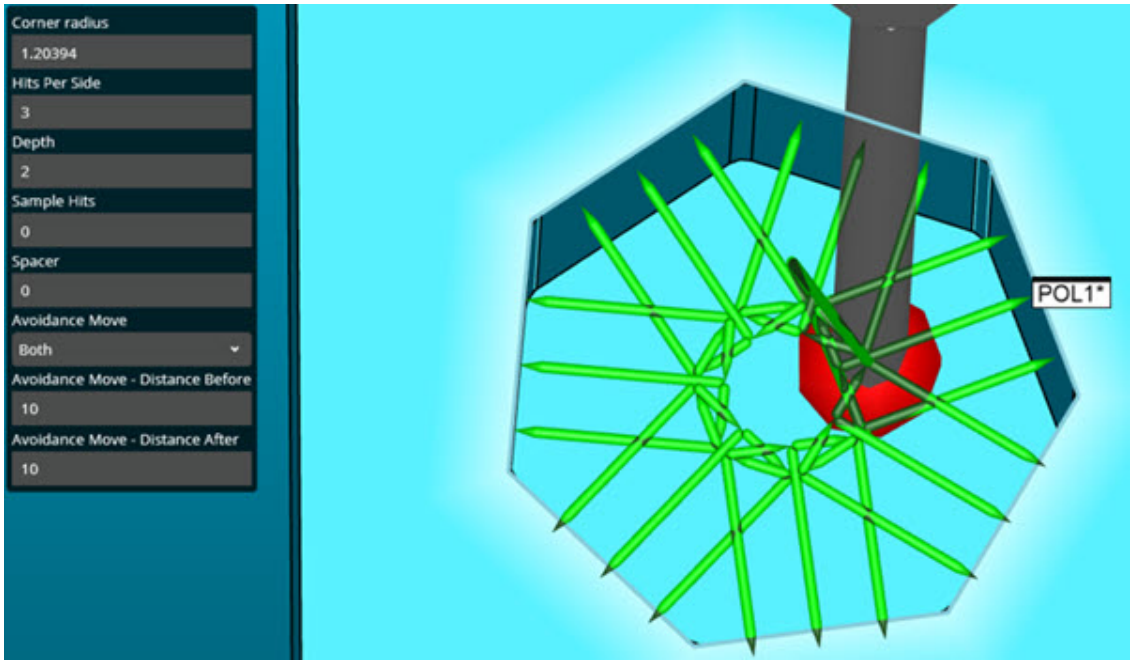
1. Press and hold Shift, and then hover the pointer over one side of the polygon.

2. Click and drag the pointer a short distance along the side to highlight it (1).
3. Move the pointer to an adjacent side (2).
4. Once the entire polygon is highlighted, release the mouse button (3).

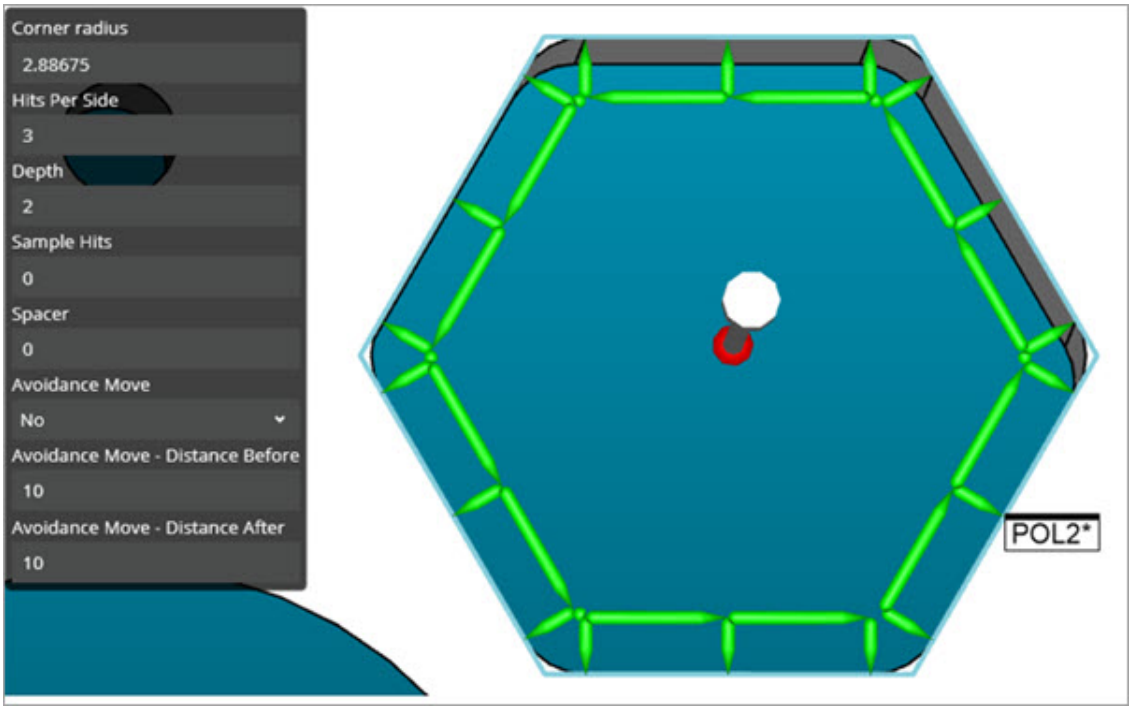


If the polygon corner has a radius, PC-DMIS detects it and applies one of these rules:

- If the detected blend radius is less than or equal to the currently loaded probe's active tip radius, PC-DMIS sets the auto-polygon's corner radius based on a formula which takes the active tip radius, the detected blend radius and the number of sides into account. The probe tip should always clear the corner radius and adjacent wall when taking hits.



- If the detected blend radius is greater than the currently loaded probe's active tip radius, PC-DMIS sets the auto-polygon's corner radius value equal to the tangent point between the detected blend radius and the straight section.



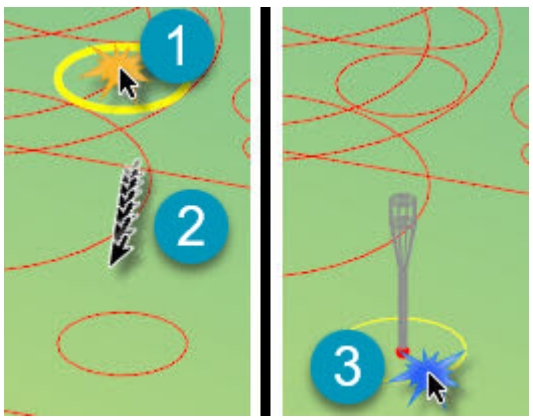
Creating Features from Wireframe Models

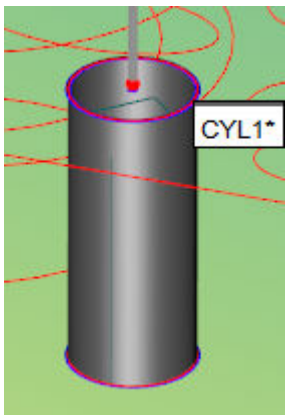
You can create any feature except those mentioned in the "Requirements" heading above.

From the **Graphic Modes** toolbar, select **Curve Mode** and then follow the instructions given in this topic to create the feature as usual.

Cone and cylinder features require two circle elements with coaxial centroids. For cone and cylinder features:

1. Press the Shift key and then hover the pointer over one of the circles making up the feature (1).
2. Click to highlight that circle.
3. Move the pointer to highlight the feature's other circle (2).
4. Once both elements are highlighted, release the mouse button to create the cone or cylinder from the two circles (3).





Example of a cylinder feature created from wireframe elements.

Creating Vector Point Features by Box-Selecting

If your CAD model has a lot of point features, you can quickly create them by box-selecting them. For this to work, the CAD model must already have the points defined as individual CAD elements. Also, the part model must have surface data.

1. From the **Graphic Modes** toolbar, select **Curve Mode** .
2. Press and hold Shift.
3. Click and drag a box around the point features.
4. Release the mouse button to create vector point features from the selected point CAD elements.

Using the Measurement Strategy Widget

The measurement strategy widget lets you change a feature's main parameters for the following items:

- QuickFeatures - For more information on QuickFeatures, see "Creating QuickFeatures" in the previous main topic.
- Features created during GD&T selection of embedded GD&T callouts - For more information on GD&T selection, see "Importing CAD GD&T Callouts" under "Working with CAD GD&T Callouts" in the "Editing the CAD Display: Introduction" chapter.

By default, PC-DMIS shows the widget for CMM configurations. For portable configurations, it hides the widget. For CMM configurations, the measurement strategy widget appears whenever you create one of those features, and there is no other dialog box open. You can change this default behavior from the **Use Measurement Strategy Widget** check box in the **General** tab of the **Setup Option** dialog box. For more information, see "Use Measurement Strategy Widget" in the "Setting Your Preferences" chapter.

When the measurement strategy widget appears, PC-DMIS creates the feature or features in the Edit window. As you make changes to a feature with this widget, PC-DMIS makes the same changes in the Edit window command.

> When the measurement strategy widget is open, you cannot access most other PC-DMIS options, including **Edit | Undo**. If something isn't available in PC-DMIS that you expect to see, check if the measurement strategy widget is open.

The measurement strategy widget has these labeled components:

A. **Grab bar** - The bar on the left is the grab bar. You can reposition the widget with this bar.

B. **[Feature Type]** - This icon and the tooltip text represent the feature type (Circle, Cylinder, and so on). During a GD&T selection import, for some features, you can change the feature type. In that case, a drop-down arrow with additional feature choices appears to the right of the icon.

C. **Feature** - This text shows the feature's name. You can click the text and type a new name.

D. **Sensor** - This is a view-only icon. It represents the type of sensor PC-DMIS needs to measure the feature. A probe icon means PC-DMIS needs to measure the feature with a contact probe sensor.

E. **Strategy** - You can use this list to change the selected measurement strategy.

- If you use a touch-trigger probe, the list contains a strategy named **Touch Trigger User-Defined**. With this strategy, you can click **Properties** and then click on the CAD model to define where PC-DMIS takes the hits.
- If you want to create a custom strategy, you can use the Measurement Strategy Editor. For more information on the Measurement Strategy Editor, see "Using the Measurement Strategy Editor" in the "Setting Your Preferences" chapter.

F. **Options** - This icon shows this menu of options:

- **Save as Default** - This saves your changes as the new default settings for the current feature type. PC-DMIS saves these defaults in the Measurement Strategy Editor.

This is available if you mark the **Use Measurement Strategy** check box from the **General** tab of the **Setup Options** dialog box (F5). For information, see "Using the Measurement Strategy Editor (MSE)" in the "Setting Your Preferences" chapter.

G. **Properties** - Click to display the feature's main properties. You can use the values that appear to change those properties. If you want to change other properties or change what properties appear in the widget, you can use the Measurement Strategy Editor. If you use the **Touch Trigger User-Defined** strategy, the **Properties** shows a **Number of Hits** display. With this display open, you can click on the CAD model to define where PC-DMIS takes the hits.

H. **Cancel** - This closes the widget and removes any features that PC-DMIS created when the widget first opened for the QuickFeature or GD&T callout. It does not remove a feature once you click **Apply** to confirm its creation.

I. **Apply** - This confirms any changes you made to the current feature. If you created only one feature, the widget applies the change and then closes. If you created more than one feature, the widget applies the change to the current feature, and then it shows the next created feature. For QuickFeatures, if you create a new feature and you don't click **Apply** first, the software applies your feature, and the measurement strategy widget shows your next feature. This also works with multi-QuickFeatures.

J. **Apply All** - This confirms any changes you made to all the created features. If you have already used the **Apply** button to modify properties on one or more features, **Apply All** applies the changes to only the *remaining* features.

There's also a grid button that appears on some parameters. The color of the grid button indicates whether the grid is enabled.

- If the button is gray, smart parameters are currently disabled. You can click this button to enable them.
- If the button is green, smart parameters are currently enabled. You can click this button to disable them.

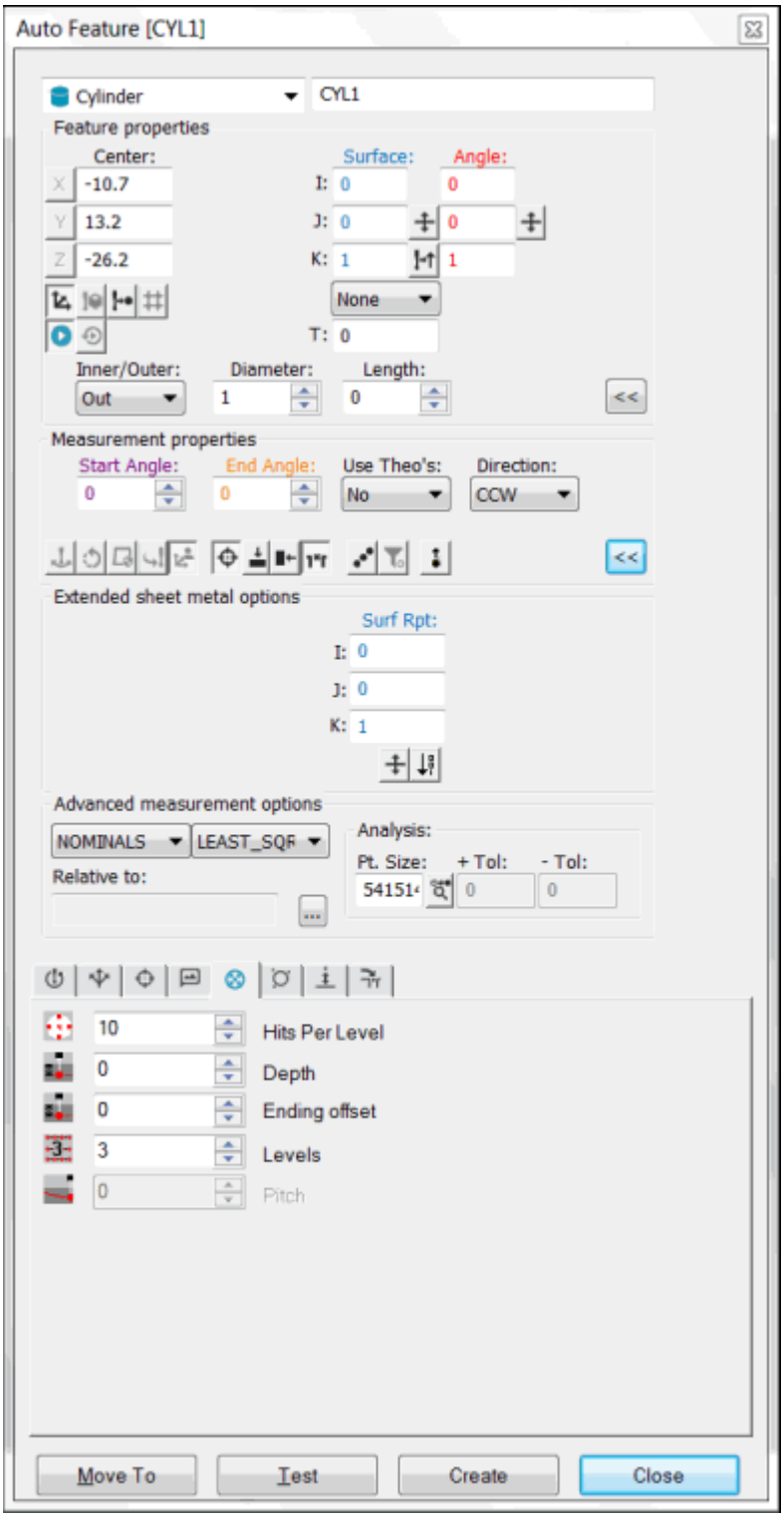
The value of the setting to the left of the grid button shows the value PC-DMIS chooses for the feature. You can click the grid button to switch between the smart parameter value and the default value for the feature you're creating.

If you want to define a custom value, turn off smart parameters and type the value.

For more information on smart parameters, see "Working with Smart Parameters" in the "Setting Your Preferences" chapter.

The Auto Feature Dialog Box

To create an Auto feature, use the **Auto Feature** dialog box. To open the dialog box, select **Insert | Feature | Auto** and then select an item from that menu.



Example of the Auto Feature dialog box for a cylinder feature.

Whenever you open or modify the **Auto Feature** dialog box, PC-DMIS queries the values from and stores them in a JSON file.

For information about the default settings and on how to use the Measurement Strategy Editor to change default settings, see "Using the Measurement Strategy Editor" topic.

Probe Deflection and CAD Click Behavior

If the **Auto Feature** dialog box is open, and PC-DMIS detects a probe hit, it assumes you are trying to learn the currently selected Auto feature type. The software shows a prompt to inform you to measure the remaining hits (if any) to complete the learn process.

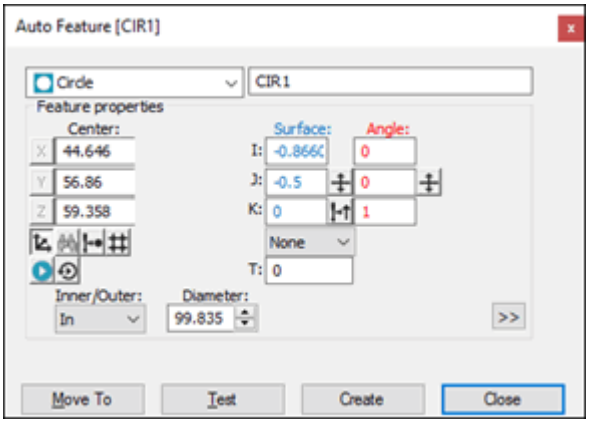
Similarly, if you click on the CAD data while the **Auto Feature** dialog box is open, PC-DMIS assumes that you are trying to learn the currently selected feature type. It populates the dialog box with the information it gathers from the CAD model.

Probe Toolbox

Because PC-DMIS frequently uses the items in the Probe Toolbox when you create Auto features, the Probe Toolbox exists as an embedded portion of the **Auto Feature** dialog box.

Basic or Advanced Configurations

By default, PC-DMIS displays the **Auto Feature** dialog box in a basic configuration where advanced options are hidden from view:



Auto Feature dialog box in its basic configuration.

Click this button in the **Feature properties** area to show this dialog box in an advanced configuration. If the Probe Toolbox is visible before you open the **Auto Feature** dialog box, it is visible, even with the basic configuration. In some configurations of PC-DMIS, such as laser or vision, the Probe Toolbox appears attached to the basic configuration.

Docking or Undocking the Auto Feature Dialog Box

The **Auto Feature** dialog box docks into the left and right sides of the screen by default. If desired, you can change its state to hover over the user interface instead. To do this:

1. Press the Ctrl key, drag the dialog box to a new location, and release the mouse. The dialog box now hovers over the interface.

2. Right-click on the title bar and from the menu, choose **Floating**.

   The next time you open the dialog box, it remains in this floating mode.

3. To return the dialog box to a dockable mode, select **Dockable** from that menu instead.

You can also hold down the Ctrl key while you drag the **Auto Feature** dialog box to prevent it temporarily from docking.

Auto Feature Type List

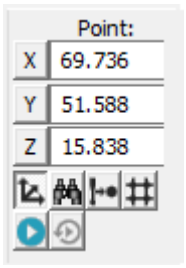
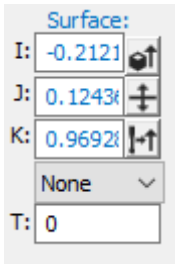
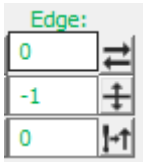
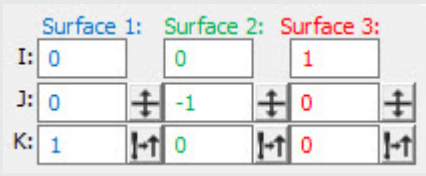
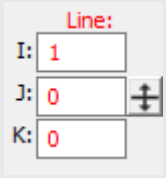
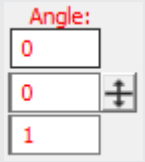
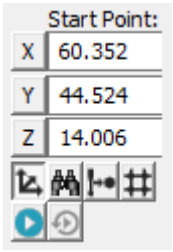
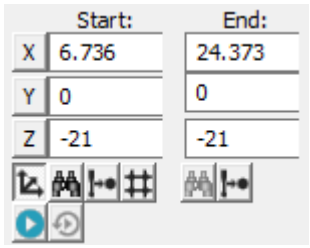
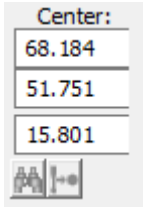
The **Auto Feature Type** list shows the currently selected Auto feature type. You can also use this list to change to a different Auto feature type. All the supported auto features for your configuration are available from this list. If you change to a different Auto feature type, the **Auto Feature** dialog box changes its contents with items used to create the newly selected feature type.

ID Box

The **ID** box shows the current ID for the Auto feature being created. You can change the ID to something different by modifying this value.

Feature Properties Area

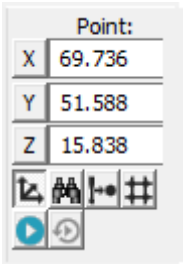
The **Feature Properties** area of the **Auto Feature** dialog box contains some or all of the following items, depending on the Auto feature you are creating.

 XYZ Point boxes	 IJKT Surface boxes Use Thickness	 IJK Edge boxes
 IJK Surface Vector boxes	 IJK Line Vec boxes	 IJK Angle boxes
 XYZ Start Point boxes	 XYZ Start / End boxes	 XYZ Center boxes

An Example:

Auto Line Vectors

XYZ Point Boxes



For a High Point feature, the **XYZ Point** boxes display the X, Y, and Z nominal for the start point.

After the new value is created, PC-DMIS draws the animated probe at the new location. This location indicates the beginning position of the search. After the execution is complete, the XYZ point contains the high point in the current workplane. Subsequent executions however use the original start point for the search.

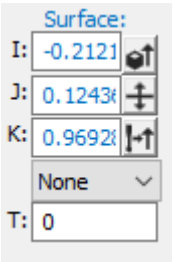
For all other feature types, the **XYZ Point** boxes display the X, Y, and Z nominal for the location of the feature.

After the new value is created, PC-DMIS draws the animated probe at the new location. This location indicates where the probe takes the hit on the part.

For information on the **X**, **Y**, and **Z** axes check boxes, see the "Find Nearest CAD Element" topic.

Icon	Description
	Polar / Cartesian
	Find Nearest CAD Element
	Read Point from Machine
	Snap to Grid
	Measure Now
	Re-Measure

IJKT Surface Boxes



I, J, and K Boxes

These boxes contain the I, J, and K normal vectors that you supply. The I, J, and K should always point away from the surface. After the new value is created, PC-DMIS normalizes the vector, making its length one unit. This vector is used for probe compensation. PC-DMIS displays a corresponding colored arrow showing the surface vector.

If you cannot see the vector arrow or if it appears too small, try changing the value in the **Pt. Size** box in the **Analysis** area. Setting **Pt. Size** to 0 will generally set both the point and the arrow to a desirable size.

For vector point, surface point, and high points: The **IJK Surface** boxes display the approach direction for the hit that was taken to create the Auto feature. For vector points, if you display the feature in **Polar** coordinates, and modify the **A** angle, the surface vector updates automatically. For more information on how to switch between Cartesian and Polar coordinates, see "Polar / Cartesian Toggle".

For high points: After execution, the IJK normal vector will display the approach vector for the high point in the current workplane.

For the circle, cylinder, sphere, and cone features: The **IJK Surface** boxes define the feature centerline. A cone's vector is very important. The normal vector of a cone feature is the direction of the cone from its tip to its base. The height and depth of a cone are always relative to this vector.

For the square slot, round slot, ellipse, and notch features: The **IJK Surface** boxes define the surface normal vector of the plane that the feature lies in (the plane parallel to the feature).

For a plane feature: The **IJK Surface** boxes define the approach direction for the plane's hits.

For a line feature: The **IJK Surface** boxes help define the edge for the auto line hits. Specifically, it defines the surface perpendicular to the surface on which the hits for the line are taken. In other words, it is always perpendicular to the edge vector. See "Example of Auto Line Vectors".

T List and Box

The **T** ("thickness") list and box enable you to select the thickness type (theoretical, actual, or no thickness). For more information, see "Use Thickness".

Icons

Icon	Description
	Find Vectors
	Flip Vector
	Read Vector From Machine

Polar / Cartesian

This icon switches your coordinate system between **Polar** and **Cartesian** modes. This changes the coordinate system PC-DMIS uses to display the point or center values for the Auto feature.

In **Polar** mode, when the icon is raised, the values appear as radius, angle, and height (shown as **R**, **A**, and **H** in the dialog box). The height depends on the workplane currently being used. If the current workplane is ZPLUS, the height is the Z value. In this mode, if you are modifying a Vector Point feature and update the value of the angle in the **A** box, PC-DMIS automatically updates the corresponding surface vector of the feature. This helps you to update the surface vector in measurement routines that don't use CAD. For more information on the surface vector value, see "IJKT Surface Boxes".

Inner/Outer/None list

In **Polar** mode, if you select the Azimuth angle (A) or you select the In or Out option from the list and the proper symmetry is set (Cylindrical or Spherical), the surface vector is correctly defined. If you select None, no change is applied to the surface vector.

The **Inner/Outer/None** is only available when you select the Polar mode option. PC-DMIS applies it when you either:

- Select the Azimuth (A) data value, and validate the data.
- Change the option from In to Out (or Out to In). The software does not apply it when you click on your CAD model in the Graphic Display window. If you select the CAD in the Graphic Display window, you get the surface vector of the exposed CAD.

The **Inner/Outer/None** information for auto vector and auto surface points are not explicitly exposed in the Edit window when it is in Summary Mode, Command Mode, or DMIS Mode.

In **Cartesian** mode, when the icon is depressed, the values appear in XYZ.

Find Nearest CAD Element

When the **Find Nearest CAD Element** button is clicked, PC-DMIS locates the nearest CAD element in the Graphic Display window based on the XYZ location and any selected

axis (or axes). PC-DMIS allows you to either type or surface-select the pertinent information.

Understanding Axes Check Box Selecting

- **For Vector Points or Surface Points** - If you select an **X**, **Y**, or **Z** axis check box, PC-DMIS interprets the selection as the axis that will be modified when you click the **Find Nearest CAD Element** icon.
- **For Edge Points or Angle Points** - If you select an X, Y, or Z axis check box, PC-DMIS interprets the selection of the axis as a selection of the one axis that will NOT be modified when you click the **Find Nearest CAD Element** icon. For example, if you select the **X** check box, PC-DMIS actually select the **Y** and **Z** check boxes signifying that the Y axis and Z axis values will be modified by the Find process.

Understanding the Find Operation without Selecting Axes Check Boxes

- **For Edge Points and Angle Points** - If you click the **Find Nearest CAD Element** icon without an axis selection, PC-DMIS finds the closest CAD edge or angle point.
- **For Vector Points and Surface Points** - If you click the **Find Nearest CAD Element** icon without an axis selection, PC-DMIS finds the closest CAD along the Normal vector specified in the dialog box. PC-DMIS then populates the dialog box with the found vector.

Read Point from Machine

The **Read Point from Machine** icon immediately reads the probe's current position and fills its location into the XYZ of the feature.

Find Vector(s)

The **Find Vector(s)** icon pierces all surfaces along the XYZ point and IJK vector, looking for the closest point. The software then flips this vector based on the current active tip vector.

The surface normal vector is displayed as the `IJK NOM VEC` but the XYZ values do not change.

This icon is available on these Auto features:

- Vector Point
- Surface Point
- Edge Point

- Angle Point
- Corner Point
- High Point
- Auto Line
- Auto Plane

Snap to Grid

The **Snap To Grid** icon snaps a supported auto point feature to the 3D grid display in the Graphic Display window. As long as the **Auto Feature** dialog box is opened to a supported auto point feature, you can use snap to grid even when the 3D grid is not showing.

> To turn on the grid in the Graphic Display window, click the **3D Grid** check box in the **View Setup** dialog box, or click the **Show a 3D Grid in the Graphic Window** icon on the **Graphic View** toolbar.

Supported auto point features include all auto point features except for Corner Point and High Point.

With this icon selected, if you then select a point on a CAD surface, the XYZ value snaps to the grid.

- With the 3D grid turned on, the selected point snaps to the visible grid. For example, if you are looking at the part in a Z+ view, then the X and Y grid lines are used. In general, whichever axis points the most towards you, the other two grid axes are used for the snapping.
- With the 3D grid turned off, the selected point snaps to the invisible grid lines based on the feature normal. For example, if you select a point for a surface point feature, and the surface normal at the selected point is (1,0,0), then the Y and Z grid lines are used for the snapping. In general, whichever axis value of the feature normal is largest, the other two grid axes are used for the snapping.

Measure Now and Re-Measure

Measure Now icon

If you select this icon, the software starts the measurement process for this Auto feature as soon as you click the **Create** button. PC-DMIS measures the part based on the values specified in the Auto Feature dialog box.

Re-Measure icon

This icon is available for these Auto features: Circle, Cylinder, Square Slot, Round Slot, and Notch, both inner and outer.

If you select **Re-Measure**, PC-DMIS automatically re-measures the feature against the measured values obtained the first time the feature is measured.

For all inner and outer features, if you set **Avoidance Move** to a value other than NONE, if avoidance moves are provided, they are used as safe moves in the re-measure path.

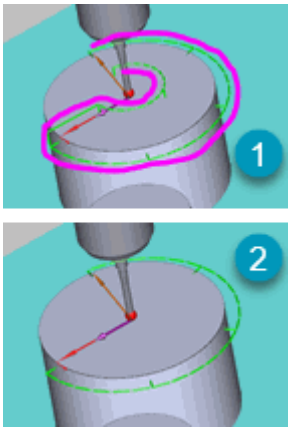
For all inner features, if avoidance moves are not provided (**Avoidance Move** is set to NONE), but **Re-Measure** is set to ON, the safe move is defined to be the center of the feature, with the height equal to the probe diameter.

For all outer features, if avoidance moves are not provided (**Avoidance Move** is set to NONE), but **Re-Measure** is set to ON, the safe moves are defined as the measure path, but REVERSED (see examples below). To use the **Re-Measure** functionality on auto contact outer features, the following conditions must be met:

- The auto contact feature must be an outer feature.
- **Avoidance Move** is set to off.
- The Probe Mode must be set to DCC Mode.
- You must set **Re-Measure** to On.

The following examples describe how the Re-Measure implementation works if the avoidance moves are not provided.

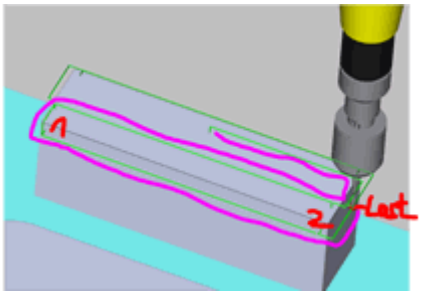
Example 1



When an Outer Circle, Outer Round Slot, or Outer Polygon is being measured, PC-DMIS stores all the collected movements, including the movements to measure the sample hits (1).

When the Re-Measure function is implemented, all movements are sequentially made in reverse to prevent collisions from occurring (2).

Example 2



When an Outer Square Slot auto contact feature is being measured, PC-DMIS stores all the collected movements that include the first and second hits on the first face of the slot, and the last hit on the second face of the slot.

As with example 1, when the Re-Measure function is implemented, all movements are sequentially made in reverse to prevent collisions from occurring.

Flip Vector

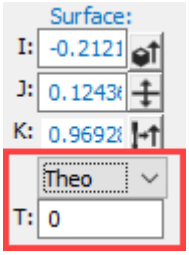
The **Flip Vector** icon allows you to reverse the direction of the surface vectors. Click on **Flip Vector** to reverse the displayed values.

Read Vector From Machine

Clicking this icon tells PC-DMIS to use the currently active tip vector from the machine for the **IJK Surface** boxes used to define the surface vector.

Use Thickness

You can use the **T** ("thickness") list and box in the Feature Properties area of the Auto Feature dialog box to enter the part thickness distance that will be applied to the surface or edge values of a feature depending on the thickness type (theoretical, actual, or no thickness).



When you measure the side of the part that is modeled in PC-DMIS, you should set the part thickness value to zero. You should only use the part thickness option when measuring the side of the part that is not drawn in the CAD data.

Defining a thickness amount is primarily used for thin parts (plastic or sheet metal) where the CAD data only describes one side and you want to measure the other side. Often with thin parts, the CAD engineer will only draw one side of the part, and then specify the material thickness. PC-DMIS will apply this material thickness automatically when using the CAD surface data.

You can use either a positive value or negative value. This thickness will be applied automatically along the surface normal vector each time CAD data is selected. If the feature has more than one normal vector (that is, angle points and corner points), the thickness will be applied along the first normal vector.

From the list, select an option:

Theo - In the **T** box, type the thickness value when the theoretical values need to be adjusted by a thickness, because measurements are being done on opposite side of material. The Edit window command line for this option would read:

```
THEO_THICKNESS = n
```

Where `n` is a numerical value indicating the theoretical thickness of the part.

Actual - In the **T** box, type the thickness value when measurements are being adjusted by the thickness back to the original theoretical XYZ locations. With this option, it will appear that the theoretical and measured values have not been offset

and the target is offset. You still need to modify your target so that PC-DMIS drives to the correct location. The Edit window command line for this option would read:

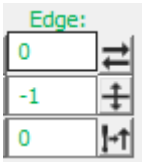
```
ACTL_THICKNESS = n
```

Where `n` is a numerical value indicating the actual thickness of the part.

None - Select this option if you do not need to apply thickness. (You do not need to enter a value in the **T** box.) For most machines, a thickness of 0 or selecting **None** has the same effect. For portable arm machines, however, selecting **None** and specifying a value actually applies the thickness to a shank style measurement. In this type of measurement, you use the cylindrical shank of the probe to measure instead of the probe tip. To do this, you need to first define sample hits. PC-DMIS can then determine the location of the supported feature (circles, ellipses, slots, and notches) using the shank.

When switching between theoretical and actual thickness, the measurement location will not change. When using theoretical thickness, PC-DMIS modifies the theoretical, measured, and target locations to include the (theoretical) thickness. When using actual thickness, PC-DMIS modifies only the target location by adding the (actual) thickness value to the original theoretical location. Later, after feature measurement, PC-DMIS subtracts the (actual) thickness from the measured value. Both methods yield the same measurement location. It is how PC-DMIS reports the feature's theoretical, actual, and target values that changes.

IJK Edge Boxes



These boxes are only available for the Edge Point and Line features.

The **IJK Edge** boxes define the approach direction (vector) for the edge hit or for the points of an auto line. This is a user-supplied I, J, K measured vector. The I, J, K should always point away from the edge and be perpendicular to the edge that is being measured.

After you define a new value, PC-DMIS normalizes the vector, making its length one unit.

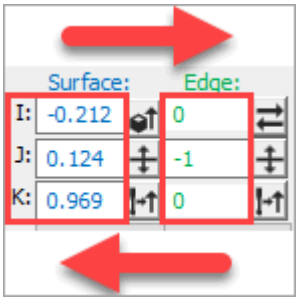
T List and Box

The **T** ("thickness") list and box enable you to select the thickness type (theoretical, actual, or no thickness). For more information, see "Use Thickness".

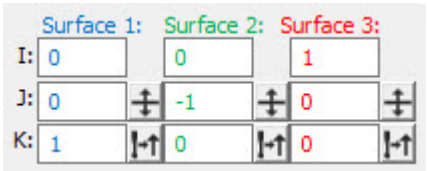
Icon	Description
	Swap Vectors
	Flip Vector
	Read Vector from Machine (for Edge Point)

Swap Vectors

The **Swap Vectors** icon causes the current Edge vector and Surface vector to switch vectors with each other.



IJK Surface 1, 2, and 3 Vector Boxes



Surface 1 and **Surface 2** boxes are only available for Angle Point or Corner Point auto features. **Surface 3** boxes are only available for Corner Point Auto features.

The I, J, K vector should always point away from the measured surface.

- **Surface 1** (blue) - Determines the surface normal vector of the first measured surface.
- **Surface 2** (green) - Determines the surface normal vector of the second measured surface.

- **Surface 3** (red) - Determines the surface normal vector of the third measured surface.

After you define a new value, PC-DMIS normalizes the vector, making its length one unit.

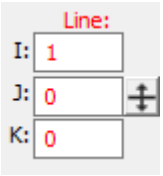
For the three surfaces, PC-DMIS displays corresponding colored arrows showing the surface vectors.

If you can't see the vector arrow or it appears too small, try changing the value in the **Pt. Size** box in the **Analysis** area. If you set **Pt. Size** to 0, it generally sets both the point and the arrow to a desirable size.

Icon	Description
	Flip Vector
	Read Vector from Machine

For more information on surface boxes, see "IJKT Surface boxes".

IJK Line Boxes



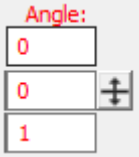
These boxes are only available for the Angle Point and Line features.

The **Line** boxes display the vector of the line on which the angle point or line lies. This is a user-supplied I, J, K normal vector.

After you define a new value, PC-DMIS normalizes the vector, making its length one unit.

Icon	Description
	Flip Vector

IJK Angle Boxes



For Circle, Cylinder, Sphere, and Cone features, the **Angle** boxes define the 0° position about the normal vector. The start and end angles are calculated relative to this vector. If the vectors are not perpendicular, the angle vector is adjusted to the normal vector.

For Square Slot, Round Slot, and Ellipse features, the **Angle** boxes define the feature's secondary vector. This is a user-supplied I, J, K normal vector. The feature centerline and normal vector must be perpendicular to each other.

For a Notch feature, the **Angle** boxes define the notch's secondary vector orientation. This is a user-supplied I, J, K normal vector along the back side of the slot. The notch angle vector and notch normal vector must be perpendicular to each other.

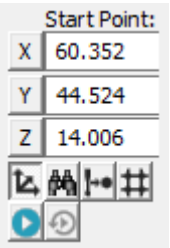
For a Plane feature, the **Angle** boxes define the plane's secondary vector. This helps control the orientation of the plane's path.

For a laser Auto High Point feature, if you select the **Box** method from the **Mode** list in the **Measurement properties** area of the laser **Auto Feature** dialog box, the **Angle** boxes define the vector for the region that PC-DMIS uses to determine the high point.

After a new value is created, PC-DMIS will normalize the vector, making it a length of 1.

Icon	Description
	Flip Vector

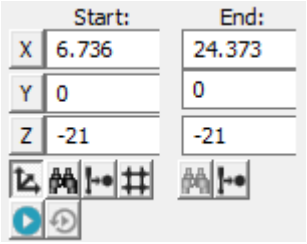
XYZ Start Point Boxes



The **XYZ Start Point** boxes define the XYZ location where the search begins for the highest point in the search region. This is used with the Auto High Point feature.

Icon	Description
	Polar / Cartesian
	Find Nearest CAD Element
	Read Point from Machine
	Measure Now
	Re-Measure

XYZ Start / End Boxes



The **XYZ Start** and **XYZ End** boxes display the starting and ending points for an Auto Line feature. You can click on the **Read Point from Machine** icons to get the current probe's XYZ location. Or you can simply probe points on the part to show the start and end point values.

Note that the **End** boxes only appear if you select **Yes** from the **Bounded** list from the **Measurement Properties** area.

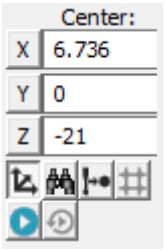
See the "Bounded List" topic.

Icon	Description
	Polar / Cartesian
	Find Nearest CAD Element
	Read Point from Machine
	Measure Now
	Re-Measure

XYZ Center Boxes

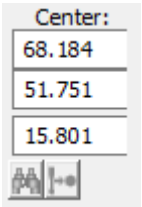
These boxes are only available with these Auto feature types: High Point, Plane, Circle, Ellipse, Round Slot, Square Slot, Notch, Polygon, Cylinder, and Sphere.

For Plane, Circle, Ellipse, Round Slot, Square Slot, Polygon, Cylinder, and Sphere, the **XYZ Center** boxes indicate the nominal center location of the feature.



For a Notch, these boxes indicate the midpoint of the notch along the non-parallel side.

For a High Point, these boxes indicate the center of the search region.

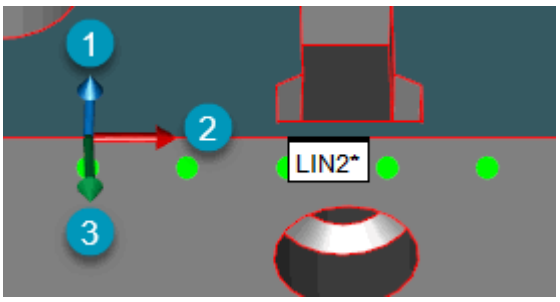


After you define a new value, PC-DMIS normalizes the vector, making its length one unit.

If a cylinder is defined as a hole, the center point must be defined at the top of the cylinder. If the cylinder is defined as a stud, then the center point must be defined at the bottom of the cylinder.

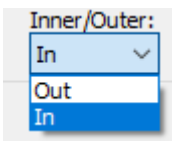
Icon	Description
	Polar / Cartesian
	Find Nearest CAD Element
	Read Point from Machine
	Measure Now
	Re-Measure

Example of Auto Line Vectors



1. *IJK Surface*
2. *IJK Line Vec*
3. *IJK Edge*

Inner / Outer List



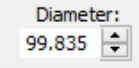
The **Inner / Outer** list tells PC-DMIS to construct the Auto feature as either an internal feature or an external feature.

- If you select **Out**, PC-DMIS constructs the circle as an external feature that protrudes from its surrounding surface.
- If you select **In**, PC-DMIS constructs the feature as an internal feature that sinks into its surrounding surface.

See also:

Exporting Plane Features in IGES

Diameter Box



This box is only available for these Auto features: Circle, Cylinder, Sphere, Cone, Polygon

The **Diameter** box defines the diameter of the feature. In the case of a stud, the diameter box displays the nominal, user supplied value.

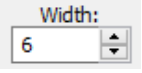
> *For an Auto Cone*, the **Diameter** value in the **Feature properties** area represents the nominal diameter at the location of the cone where you've defined the THEO XYZ location.
>
> *For a polygon*, the diameter is the distance between two opposing sides of even-sided polygons. For other polygons, such as an equilateral triangle, it is twice the radius of the largest circle you can inscribe inside the polygon.

To change the feature's diameter:

1. Select the existing value.
2. Type a new value.

Once the feature is created, PC-DMIS updates the size of the feature in the Graphic Display window.

Width Box

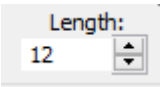


High Point

The **Width** value defines the width of the search region. If you have a value for **Length** but leave the **Width** value 0, then the **Width** value corresponds to the length along the major axis of the current workplane.

Square Slot, Round Slot, Ellipse, or Notch
The **Width** box displays the width of the feature.

Length Box



High Point
The **Length** defines the length of the search region. If you have a value for **Width**, but leave the **Length** value 0, the **Length** value corresponds to the length along the minor axis of the current workplane.

Square Slot, Round Slot, Ellipse, Notch, or Line
The **Length** box displays the length of the feature.

Cone
The **Length** box displays the length of the cone.

> *A positive Length* value indicates that the centroid is toward the cone's tip (the end of the cone feature that has the smaller diameter).
>
> *A negative Length* value indicates that the centroid is toward the cone's base (the end of the cone feature that has the larger diameter).

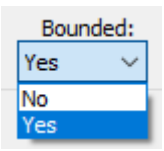
Cylinder
For a hole, the **Length** box defines the nominal length of the feature. If you enter a **Length** value but do not define a depth, PC-DMIS equally divides the **Length** value by the number of rows indicated in the **Levels** box.

The probe then works its way down the cylinder in increments until it reaches the indicated length value.

If a depth value is defined, the actual measured feature becomes the length minus the depth value.

For a stud, if the **Sample Hits** list displays a non-zero value, PC-DMIS will take an extra hit on the top of the stud in the center, as long as the length value is a non-zero positive value. PC-DMIS then will compute the length of the stud.

Bounded List



Bounded list (only for a Line feature)

The **Bounded** list defines whether or not the auto line is bounded by an ending point or an unbounded, open line.

If you select **Yes**, then some **End** boxes will appear in the **Feature Properties** area with the XYZ values for the ending point. PC-DMIS automatically calculates the length of the line based off of the distance between the **Start** and **End** points and displays the line's length in the **Length** box.

If you select **No**, then PC-DMIS expects you to type a value in the **Length** box. It then calculates the line from the starting point, along the line vector for the distance specified in the **Length** box.

See "XYZ Start / End Point Boxes" and "Length box" for additional information.

Num Sides List

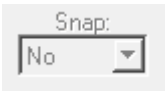
The **Num Sides** list defines the number of sides making up the polygon feature.

Measurement Properties Area

The **Measurement properties** area in the **Auto Feature** dialog box contains some or all of the following items, depending on the Auto feature that you are creating.

Snap List

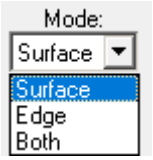
The **Snap** list is automatically enabled in the user interface when working with a Vector Point or a Surface Point feature. For a Circle feature, it is only visible if `EnableCircleDCCSnap` is set to TRUE in the PC-DMIS Settings Editor. In addition, since snap only functions well after a rough alignment, it is disabled until an alignment is established.



The **Snap** list determines whether or not measured values "snap to" the theoretical vector for a Vector Point, a Surface Point, and when enabled, for a Circle feature. This simulates a perfect machine staying exactly on the approach vector, not deviating by as much as a micron when measuring the point. If you set this to **Yes**, then the measured values snap to the theoretical vector with all the deviation along the vector of the point. This is useful for focusing on a deviation along one particular vector.

For example, suppose you want to measure the height (in Z) of a table's top. You don’t really care about your X and Y (secondary and tertiary) axis errors which can occur from machine drift (tunneling error). In this case, with **Snap** set to **Yes**, you will only be reporting the Z value. Any errors in X and Y will be ignored as the measured X and Y values will equal their theoretical counterparts.

Measure Order List



This is only available with the edge point feature.

The **Measure Order** list in the **Measurement Properties** area allows you to choose the order in which the sample points will be taken before the final hit is taken. The choices are: **Surface**, **Edge**, or **Both**.

Surface
Measures the three hits, first on the surface, then on the edge.

Edge
Measures the two hits, first on the edge then the surface.

Both
Measures the surface, then the edge, and then the surface again.

Interior / Exterior List

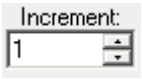
This list is available only on the Angle Point auto feature.

The **Interior / Exterior** list defines the angle as either interior or exterior.

Interior angles have the solid angle of the part less than 180°, while exterior angles are greater than 180°.

It is very important to make sure this option is set correctly because of the differences in measurement sequence for each type.

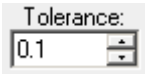
Increment Box



This box is available only on the High Point auto feature.

The **Increment** box allows you to define the increment used for the highest point in the search area. During execution, PC-DMIS begins searching from the start point (or search point) by the amount specified in the **Increment** box.

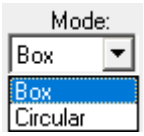
Tolerance Box



This box is available only on the High Point auto feature.

The **Tolerance** box allows you to define a tolerance value that essentially tells PC-DMIS when to stop searching for a high point within a given area. The tolerance value should always be less than the increment value. During the search process, PC-DMIS decreases the increment value until it is less than or equal to the **Tolerance** value supplied, indicating that the highest point in the current workplane has been located.

Box / Circular List



This list is only available for the High Point feature.

The **Box / Circular** list allows you to define the search mode that PC-DMIS will use to return the highest point. You can choose **Box** mode or a **Circular** mode.

Box Mode

Selecting **Box** defines a rectangular search region for the High Point feature. The rectangle is defined by the **Width** and **Length** values. PC-DMIS will report the highest point within that area.

Circular Mode

If you select **Circular**, the **Width** and **Length** boxes change to **Outer Rad** and **Inner Rad** boxes. The search region for the High Point feature becomes a circular search band specified by the **Outer Rad** and **Inner Rad** values.

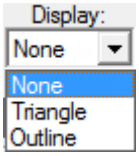
For the Auto Laser High Point feature, PC-DMIS does not display the **Outer Rad** and **Inner Rad** boxes since they are not needed.

- If a complete circular region is desired, set the inner radius to 0.
- If a circular search line is desired, set the inner and outer radii to the same value.

The highest point along the circumference will be reported.

No matter which mode you select, the start point should lie within the defined search region. For the special cases of line searching, the start point will automatically be adjusted to be on the line.

Display List



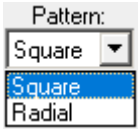
For an Auto Plane feature, the **Display** list determines how the plane will be displayed in the Graphic Display window.

- Selecting **None** will not draw the plane feature even though it's still created in the measurement routine.
- Selecting **Triangle** will cause the plane to appear as a triangle symbol around the region where the plane hits will be taken. The size depends on the hit locations.
- Selecting **Outline** will cause the plane to appear as a square or rectangular outline around the region where the plane hits will be taken. The size depends on the hit locations.

See "Using the Display Area" under the "Constructing a Plane Feature" topic for similar examples of these display types.

When you create an Auto Plane feature, it remembers the last Auto Plane feature and defaults to the last used display state.

Pattern List



For the Auto Plane feature, the **Square** and **Radial** options in the **Pattern** list allow you to determine whether the hits for the plane feature are taken in a square or radial pattern.

If you select **Radial**, PC-DMIS will create the hits in rows coming from the plane center in a circular or radial pattern. PC-DMIS takes 360 degrees and divides it by the value in the **Number of Rings** box to determine the angle between each row. For example, if the

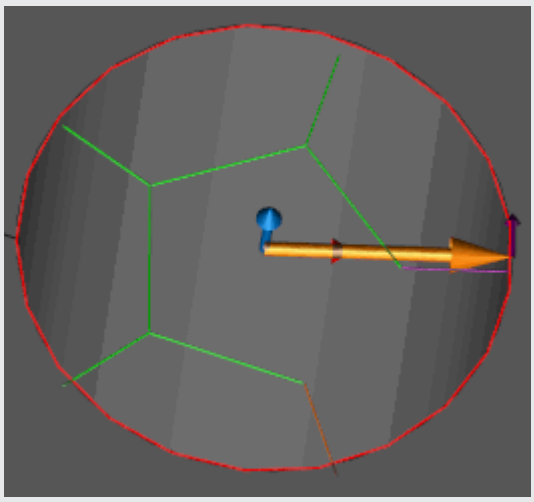
Number of Rings box had 6 and your **Number of Hits** box had 3, PC-DMIS would generate a row of three hits every 60 degrees, for a total of 18 hits.

If you select **Square**, PC-DMIS will create the hits in a grid pattern about the center point of the plane. For example, if the **Number of Rows** box had 3 and the **Number of Hits** box had 4, PC-DMIS would take a total of 12 hits in a grid pattern with the plane center point at the center.

Start and End Angles

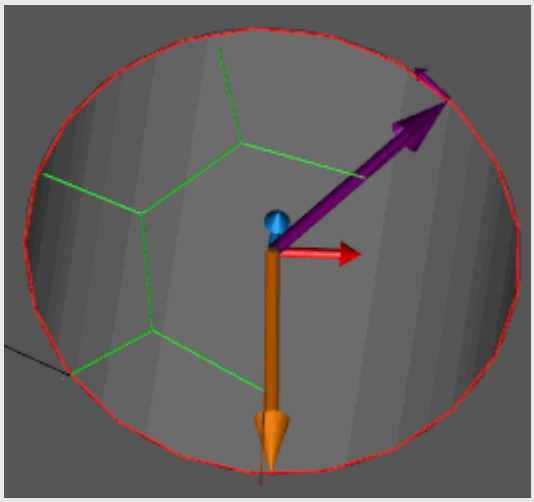
For a circular feature, the different **Start Angle** and **End Angle** boxes define where PC-DMIS probes the feature. For many situations, the default values are sufficient, but there may be times when you need to measure a circular feature that is partially obstructed by another feature or otherwise only partially available for probing. Enter the start and end angles in a counter-clockwise manner.

If you measure a hole with six hits and a **Start Angle** of 0 and an **End Angle** of 360, it looks something like this:



Notice that the **Start Angle** and the **End Angle** are the same: 0 and 360. Also, the six points making up the circle feature are equally distributed between the two angles. In this case, each point is taken at 60 degree intervals with the last point taken at 300 degrees.

However, if you change the **Start Angle** to 45 (purple arrow) and the **End Angle** to 270 (orange arrow), those values limit the hits to a certain portion of the circular feature:

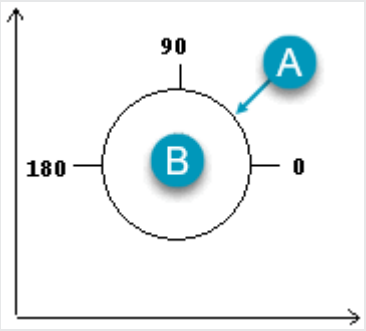


Start and End Angle Boxes

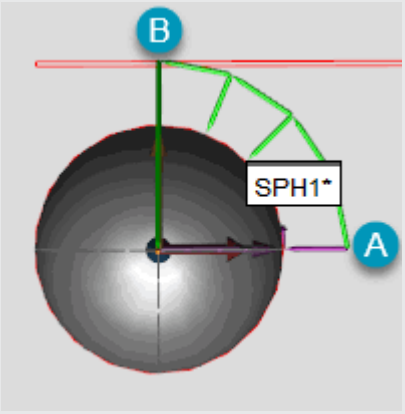
Start Angle and **End Angle** boxes - These boxes let you change the default starting and ending angles on the feature. This is a user-supplied angle in decimal degrees. The starting and ending angles are relative to the values in the **IJK Angle Vector** boxes. If you rotate the view of the feature so that you look down its center, PC-DMIS spaces the desired number of hits beginning at the **Start Angle** in a counterclockwise direction around the centerline until it reaches the **End Angle**.

Consider this example:

Suppose you have a sphere feature with a Start Angle of 0, an end angle of 90, and an IJK Angle Vector of 1,0,0 (along the X+ axis). The starting and ending angles are relative to the angle vector. Measurement proceeds in a counterclockwise manner, equally spaced between 0 and 90 degrees:



(A) - Sphere (B) - Top



Screen capture of an Auto Sphere showing path lines and hit locations from Start Angle of 0 (at A) and End Angle of 90 (at B).

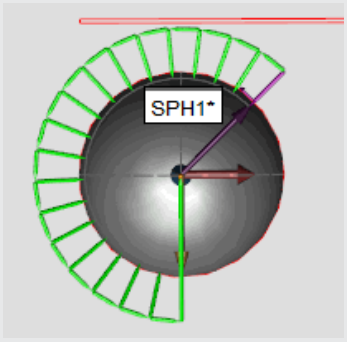
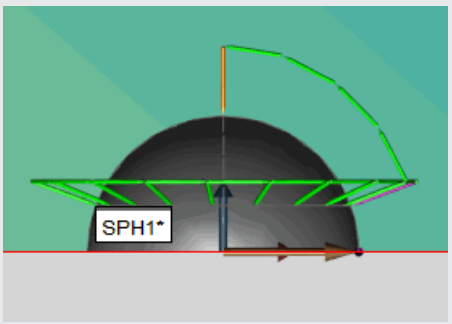
These boxes are only available for Circle, Cylinder, Ellipse, Cone, and Sphere Auto features.

Start Angle 2 and **End Angle 2** boxes - The **Start Angle 2** and **End Angle 2** boxes let you determine the secondary starting and ending angle on a sphere feature. This angle is in decimal degrees. The secondary angle, if looking at a side view of a sphere, starts at the sphere's equator and then moves towards the top pole as the angle increases to 90 degrees and then back down to the equator on the opposite side of the sphere at 180 degrees. These angles allow you to place the hits in an area where the probe can reach them without obstruction.

Consider this example:

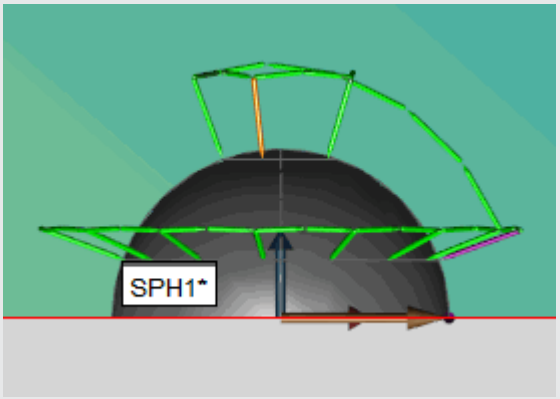
Suppose you have an external sphere that's halfway visible above the surrounding surface. If you use a **Start Angle 2** value of 0 degrees, the probe will collide with the surrounding surface when it attempts to take the hits around the sphere's equator. A slight adjustment in the **Start Angle 2** value resolves this problem.

If you create the sphere with 20 hits, and give it a **Start Angle** of 45, an **End Angle** of 270, a **Start Angle 2** of 20, and an **End Angle 2** to 90, PC-DMIS spaces 19 hits around the sphere 20 degrees up from the equator of the sphere, like this:

	
A Top View of the Sphere	*A Side View of the Sphere*

This provides the probe with plenty of room to measure the sphere. The sphere's second level only consists of one hit on top of the sphere.

If you adjusted **End Angle 2** to something like 110 degrees, the second level of hits has 5 hits adjusted 20 degrees down from the top of the sphere:

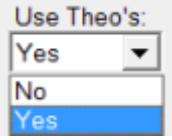


The **Start Angle 2** and **End Angle 2** boxes are only available for Sphere Auto features.

2 Degree Buffer Offset

When you generate an arc using one of the circular Auto features (Circle, Cylinder, Cone, Sphere, or Round Slot), PC-DMIS offsets the start and end angles each by two degrees. This way arcs retrieved from the CAD aren't probed on their starting and ending angles, possibly catching a corner. This shouldn't be an issue in most cases unless you are attempting to generate a small arc of only a few degrees. For example, if you want to generate a four degree arc of an Auto **Circle**, you need to type in an eight degree arc parameter in the **Start Angle** and **End Angle** boxes, knowing that PC-DMIS shortens the arc by two degrees on each angle.

Use Theo's List



This list is only available for the Cylinder feature.

The **Use Theo's** list specifies whether to use the theoretical information displayed in the **Auto Feature** dialog box when calculating the actual feature information from the measured data collected.

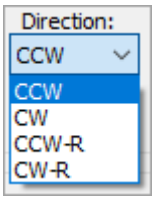
When you select **Yes**, the Best Fit algorithm will use the theoretical values (position, vector, and diameter) as an initial guess for the calculation to ensure the correct convergence of the algorithm to the desired solution. You can select **Yes** when the valid theoretical values and alignment are defined.

Select **No** when no valid alignment or theoretical values are available.

Use Pin List

Setting this list to **Yes**, shows the **IJK Punch** and **IJK Pin** boxes in the **Extended sheet metal options** area for Circle, Square Slot, and Round Slot features. The default value for this for new Auto features is **No**. See the "Extended Sheet Metal Options area" topic for information on enabling that area and using the options it contains.

Direction List



The **Direction** list specifies the direction in which PC-DMIS takes the hits.

The CCW-R and CW-R options are only available for Circle and Cylinder features when you use tactile probes.

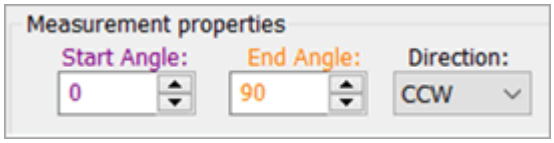
When you select the CCW-R option for example, PC-DMIS:

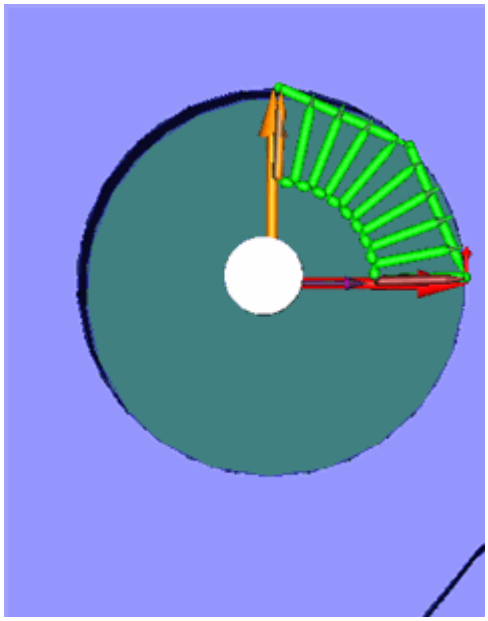
- Automatically reverses the Start and End angles
- Depending on what the original measurement direct was set to, changes the measurement direction
- Updates the feature definition in the Graphic Display window accordingly

The CW-R and CCW-R options are only available from the **Auto Circle** or **Auto Cylinder** feature dialog box. You cannot change the **Direction** option from the Edit window.

CCW Option

CCW - Select this option to change the measurement direction to counterclockwise.

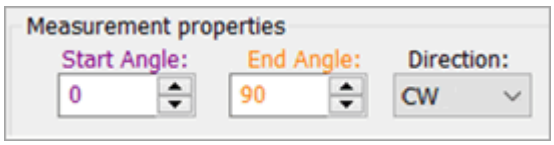


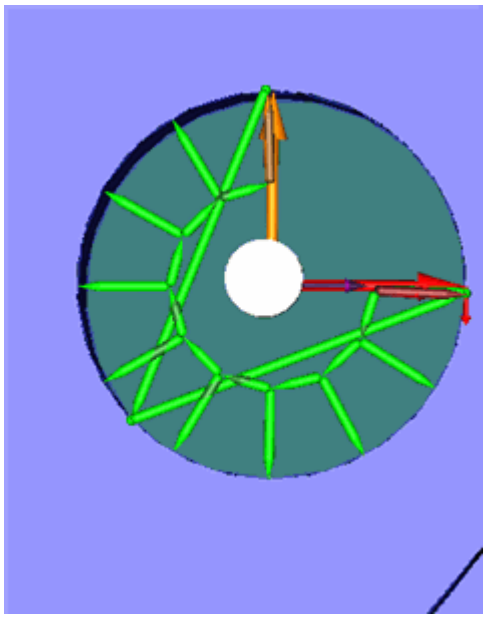


*Example of a circle with **Start Angle** = 0, **End Angle** = 90, and CCW measurement direction.*

CW Option

CW - Select this option to change the measurement direction to clockwise.



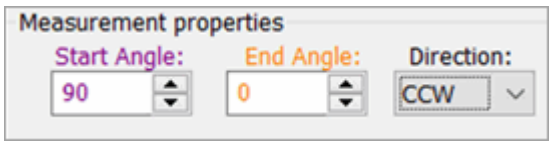


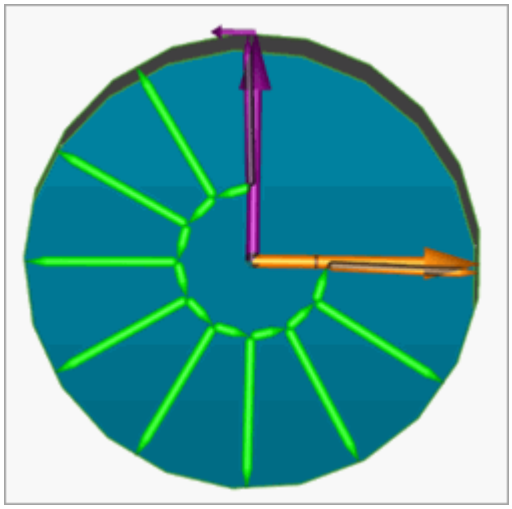
*Example of a circle with **Start Angle** = 0, **End Angle** = 90, and CW measurement direction.*

CCW-R Option

CCW-R - Select this option to reverse the Start and End angle values and to change the measurement direction to counterclockwise. When you select this option, the Graphic Display window immediately refreshes with the new parameters.

For example, if the parameters were originally set as in the **CCW Option** described above and you selected the **CCW-R** option, the result is shown below.





*Example of circle with reversed **Start Angle** = 90, reversed **End Angle** = 0, and CCW measurement direction.*

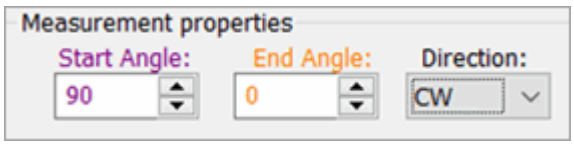
In this case, since the starting measurement direction was CCW and the selected option from the **Direction** list was **CCW-R**, only the **Start angle** and **End angle** values were reversed. The measurement direction remains CCW.

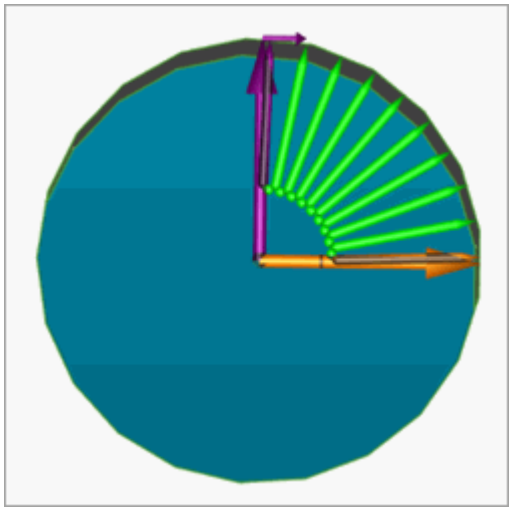
If you instead selected the **CW-R** option from the **Direction** list, PC-DMIS would reverse all three parameters.

CW-R Option

CW-R - Select this option to reverse the Start and End angle values and to change the measurement direction to clockwise. When you select this option, the Graphic Display window immediately refreshes with the new parameters.

For example, if the parameters were originally set as in the **CCW Option** described above and you selected the **CW-R** option, the result is shown below.



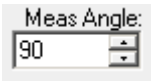


*Example of a circle with reversed **Start Angle** = 90, reversed **End Angle** - 0, and CW measurement direction.*

In this case, since the starting measurement direction was CCW and the selected option from the **Direction** list was **CW-R**, PC-DMIS reversed the **Start angle** and **End angle** values, and the measurement direction changed to CW.

If you instead selected the **CCW-R** option from the **Direction** list, PC-DMIS would only reverse the **Start angle** and **End angle** values.

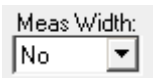
Meas Ang Box



Round slots have two rounded ends, each having up to 180 degrees of measurement radius. The value in the **Meas Ang** box defines how much of the radius to measure. PC-DMIS takes the input value and divides it in two, measuring half on each side of the slot angle vector.

Suppose you type 90 in this box. When creating the Round Slot, PC-DMIS would then measure 45 degrees to the left, and 45 degrees to the right of the angle vector (or vice versa depending on the angle vector).

Meas Width List



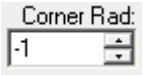
This list is available only on the Square Slot auto feature.

When you select **Yes** from **Meas Width** and then click the **Create** button, PC-DMIS will do the following during measurement:

- Measure two hits on the side of the slot.
- Adjust the angle vector.
- Measure two hits on the opposite side to calculate the width.
- Adjust the position of the last two hits at each end for the width.

Generally, Square Slots only require five hits. With this option, however, taking six hits will yield a better measurement of the width.

Corner Rad (Radius) Box



This box is only available for these feature types: Square Slot Notch, Polygon.

Square Slots, Notches, and Polygons are not always exactly square. They often have radii instead of corners. The **Corner Rad** box holds a value the size of that radius. The radius value controls where the hits take place on these features.

- *For a Square Slot,* it is used to determine the location of each hit during the measurement of the slot to avoid hitting on the radii.
- *For a Notch Slot,* hits along the edge opposite the open side will be adjusted off the radius.
- *For a Polygon*, hits along the polygon edge are adjusted from the corners of the feature adjusted off the radius.

Auto Feature Bar

The **Measurement properties** area in the **Auto Feature** dialog box has a bar with icons. You can use these icons to turn certain functions on or off.

Auto Wrist

Auto Rotary Table Moves

Clearance Plane

Circular Moves

Manual Pre-Position

Show Hit Targets

View Normal

View Perpendicular

Void Detection

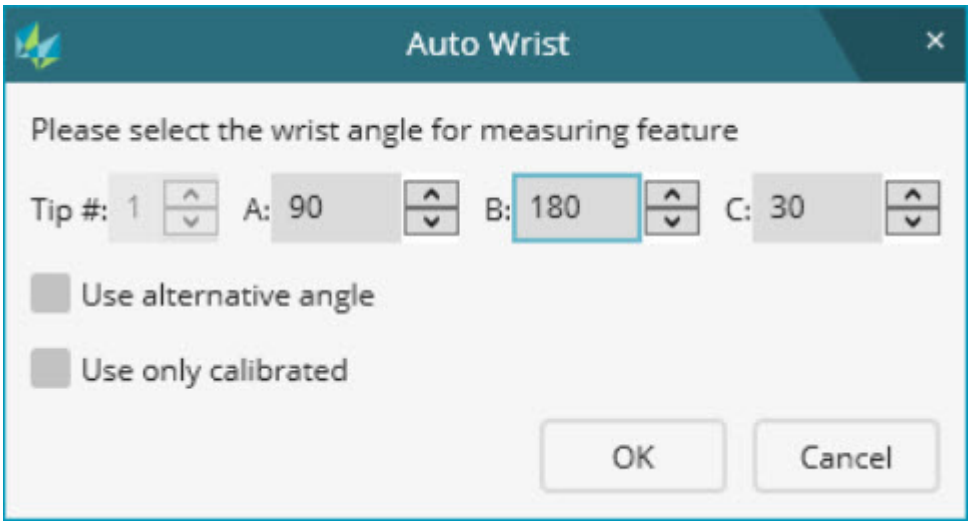
Show Measured Points

Show Filtered Points

To Points

Auto Wrist

When you select the **Auto Wrist** icon on the **Auto Feature** bar, PC-DMIS first checks whether the current active tip can be used to measure the selected feature. If not, it determines the best wrist position (tip) to use when an Auto feature is measured. Upon feature creation, PC-DMIS displays the **Auto Wrist** dialog box in case a new tip is needed:



Auto Wrist Dialog Box

Use the options in the dialog box to specify the wrist angle that best approaches the feature:

- **Tip** # box - This box represents the tip number. Typing or selecting a number in this box inserts the associated TIP command into the measurement routine.

- **A**, **B**, and **C** boxes - These boxes define the A, B, and C angles for the wrist. Click the arrow to the right of each box to increment or decrement the respective angle to the next valid value.
- **Use alternative angle** check box - If you want to avoid a possible collision during the probe rotation, select this check box.
- **Use only calibrated** check box - If you want to only use existing calibrated tips, select this check box.
- **OK** button - To insert the TIP command with the selected angles before the Auto feature, click this button.
- **Cancel** button - To use the current wrist position to measure the feature, click this button.

When you directly measure a Laser Cone or Laser Cylinder with a Tesastar wrist, the best scanning direction may be along the feature's vector or normal to it. This however depends on the ability of the head to rotate to an angle that allows you to scan in a direction normal to the stripe's orientation. For details, see "Auto Cone Paths" or "Auto Cylinder Paths" in the PC-DMIS Laser documentation.

If you select the **Auto Wrist** icon, PC-DMIS chooses a position that most closely approximates the best approach direction:

- For angle point features, the best approach direction is the average of the two surface vectors.
- For corner point features, the best approach direction is the average of the three surface vectors.
- For all other auto types, the best approach direction is the feature surface vector.
- The Vector Point, Surface Point, Angle Point, Corner Point, Line, Edge, and Plane Auto Contact Features use the cone angle of 46 degrees. This is done to prevent a tip change in case the orientation of the current tip lies inside the acceptable cone.
- The tip for the Line and Edge Auto Contact Features has to be inside of a half cone (46 degrees) specified by the edge vector.

If the icon is *not* selected, PC-DMIS uses the current wrist position to measure the feature.

Auto Rotary Table Moves

This is currently non-functional.

Clearance Plane

This icon lets you determine whether or not PC-DMIS inserts an automatic `MOVE/CLEARPLANE` command before measuring the first automatic hit of any Auto features created after selecting the toggle icon.

In order to enable the **Clearance Plane** function in the **Auto Feature** dialog box, a clearance plane command needs to be in the measurement routine (press the F10 key on your keyboard and then click the **Clearance Plane** tab): `CLEARP/ZPLUS,0,ZPLUS,0,OFF` or `CLEARP/ZPLUS,0,ZPLUS,0,ON`)

The OFF/ON at the end of this command acts as the default for the **Clearance Plane** function, each time you open a new **Auto Feature** dialog box to create a new feature. This means that each time a new **Auto Feature** dialog box is opened, the default is the OFF/ON you have on the last `CLEARP/ZPLUS,0,ZPLUS` command. If you want to create an Auto feature WITHOUT the **Auto Feature** dialog box open (using the **QuickFeature** mode for example, with hovering – Shift + hover), the value of the next QuickFeature is the last value you set it to in the dialog box. This is why with QuickFeatures you don't have control over this, while you can control it if you open the **Auto Feature** dialog box.

- If selected, PC-DMIS automatically inserts a `MOVE/CLEARPLANE` command (relative to the current coordinate system and part origin) into the Edit window before the first automatic hit of the feature. This causes the probe to move to the defined clearance plane before measuring the feature. After the last hit on the feature is measured, the probe will stay at probe depth until called to the next feature.
- If not selected, PC-DMIS creates Auto features normally without inserting any `MOVE/CLEARPLANE` commands.

Using clearance planes reduces programming time because the need to define intermediate moves is reduced. It can also help protect your hardware from an inadvertent collision with the part. For additional information on Clearance Planes, see "Parameter Settings: Clear Plane tab" in the "Setting your Preferences" chapter.

When measuring studs, be sure to set the spacer value to a distance that allows the probe to move around the stud.

Move Clearance Plane Active

The default for the **Clearance Plane** function for all Auto features is defined by the Clearance Plane Active option (if defined) and is recalled whenever the **Auto Feature** dialog box opens to create new features.

Circular Moves

This icon determines whether or not the probe will move along an arc as opposed to usually moving in a straight line when moving from one hit to the next. This option is particularly helpful when working with ring grooves.

> This icon is available for these Auto features: Circle, Cylinder, Cone, Sphere, and Plane. For a Plane feature, it becomes available if you are using a radial pattern for the feature's hits.

For information on how to insert circular moves into the Edit window, see "Inserting a Move Circular Command" in the "Inserting Moves" chapter.

Manual Pre-Position

> This icon only functions if you have PC-DMIS Vision enabled on your PC-DMIS license.

If selected, this icon prompts the user to move the camera to the position that is over the target before continuing. For additional information, see the PC-DMIS Vision documentation.

Show Hit Targets

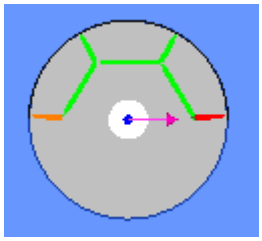
Click this icon to show the path lines and hit locations for the current feature. If the Probe Toolbox is visible, it also displays the toolbox's **Hit Targets** tab. Deselecting it hides this information.

The path lines appear as green lines on the CAD model. The red line indicates the starting hit, and the orange line indicates the ending hit. You can also modify hit locations by simply selecting and dragging the lines with your mouse.

You can also right-click on any path line or hit and use a shortcut menu to perform a variety of functions. See the "Auto Feature Path Lines Shortcut Menu" topic in the "Using Shortcut Keys and Shortcut Menus" chapter for more information.

The following example uses an auto circle feature to explain this functionality.

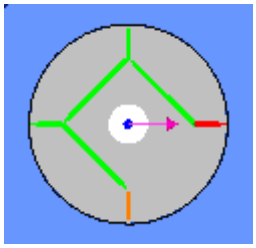
In this example, the starting and ending angles are set to measure only 180 degrees of the circle with four hits.



If you edit the starting or ending angle boxes, the hits display changes as well.

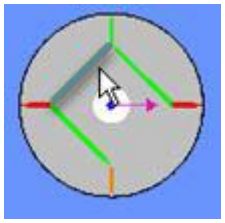
For example, if you change the ending angle from 180 to 360, PC-DMIS would then show the hits around the entire circle.

Alternately, with supported Auto features, *you can click on a hit target and drag it to a new location*. The software updates the start or end angle in the dialog box accordingly.



You can click on a path line and drag any hit to a new location.

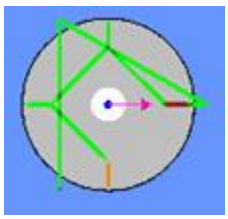
To modify a path line, move the mouse over the path line until PC-DMIS highlights the line, then click and drag the hit to a new location.



Suppose your auto circle uses three sample hits to locate the surface around the circle. PC-DMIS shows these lines as well.

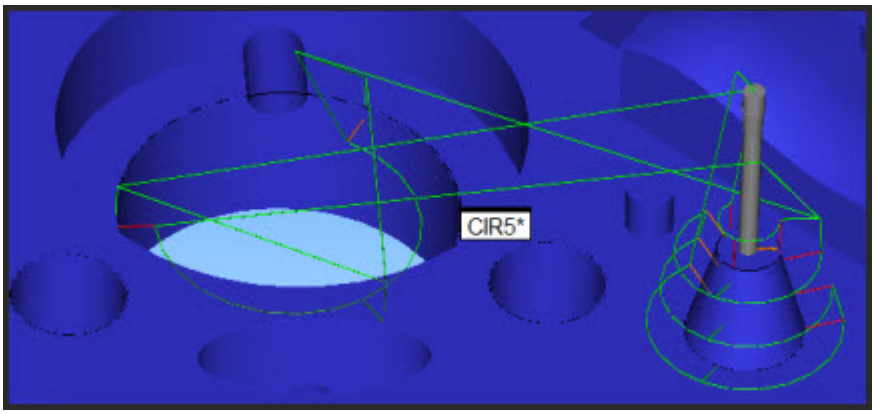
To modify sample hit path lines, click on and drag these lines to a new location.

- If you don't have user-defined hits, PC-DMIS dynamically updates the **Spacer** value as well as the feature hits.
- If you already have user-defined hits, PC-DMIS will modify only that one sample hit's location.



Show Path Lines from Previous Auto feature

With the **Show Hit Targets** icon selected, you can also show temporary path lines from the previous Auto feature to the one you're currently creating. To do this, select the **Operation | Graphic Display Window | Clearance Moves | With Feature Creation** menu item before you access the **Auto Feature** dialog box.



Example showing temporary path lines between auto features.

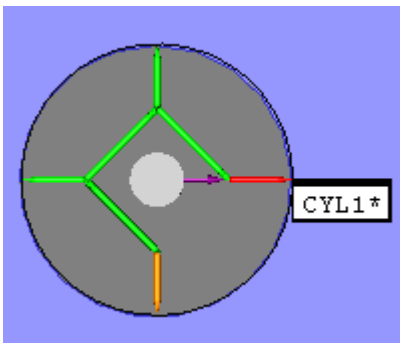
Clicking **Create** from the **Auto Feature** dialog box when doing this action creates the Auto feature as usual, and it also inserts a preceding `MOVE/POINT` command into the measurement routine.

To expand this functionality to test for collision detection between the two features, select the **Operation | Graphic Display Window | Clearance Moves | With Collision Detection** menu item. See "Inserting Clearance Moves with Collision Detection" in the "Inserting Move Commands" chapter.

View Normal

> This icon is unavailable for selection if your measurement routine is in Manual Mode.

Clicking this icon orients the CAD so that you look down on the feature. Deselecting it returns the CAD to the previous view. You can also select this by right-clicking on the path and selecting **View Normal** from the resulting menu.

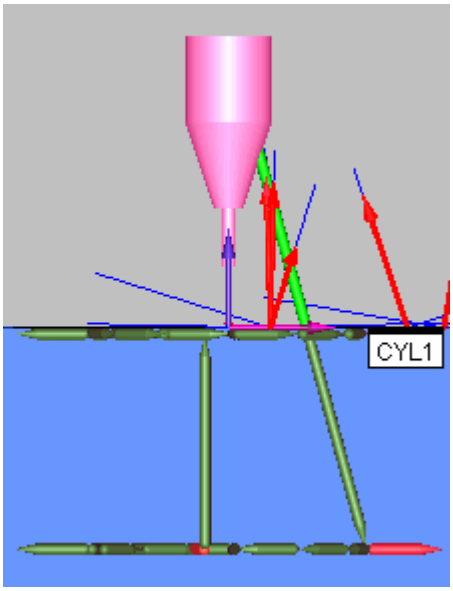


Example of a cylinder feature with the View Normal option selected.

View Perpendicular

> This icon is grayed out and unavailable for selection if your measurement routine is in Manual Mode.

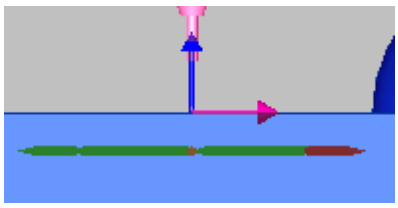
Clicking this icon orients the CAD so that you look at the side of the feature. This is ideal for defining a feature's depth or adding additional rows of hits for features that support additional levels, such as cones or cylinders. To set additional rows, right-click and select **Add Row** from the resulting menu.



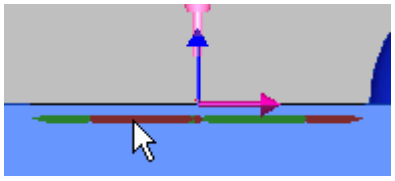
Example of a cylinder feature with the View Perpendicular option selected.

Deselecting **View Perpendicular** returns the CAD to the previous view. You can also set the view to perpendicular, by right-clicking on the path and selecting **View Perpendicular** from the resulting menu.

> With **Show Path** and **View Perpendicular** selected, you can view the feature in its perpendicular view and see at what depth PC-DMIS will take the hits.



To modify the depth, highlight the green line of hits in this view, and then click and drag either up or down to set the new depth.

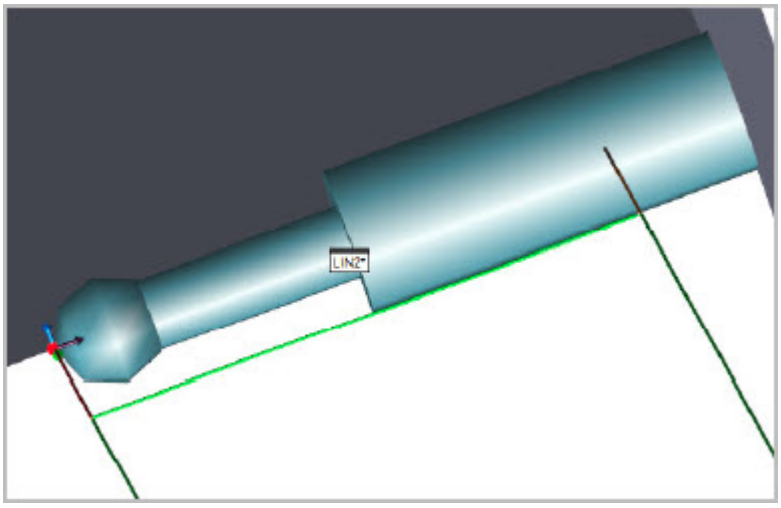


Void Detection

This icon only functions if you are working with one of these supported features: Surface Point, Edge Point, Line, Plane, Circle, Cylinder, and Round Slot.

If selected, PC-DMIS detects hit targets that would normally occur in voids (empty spaces) on the CAD model and repositions them to a safe location, usually near the edge of the void.

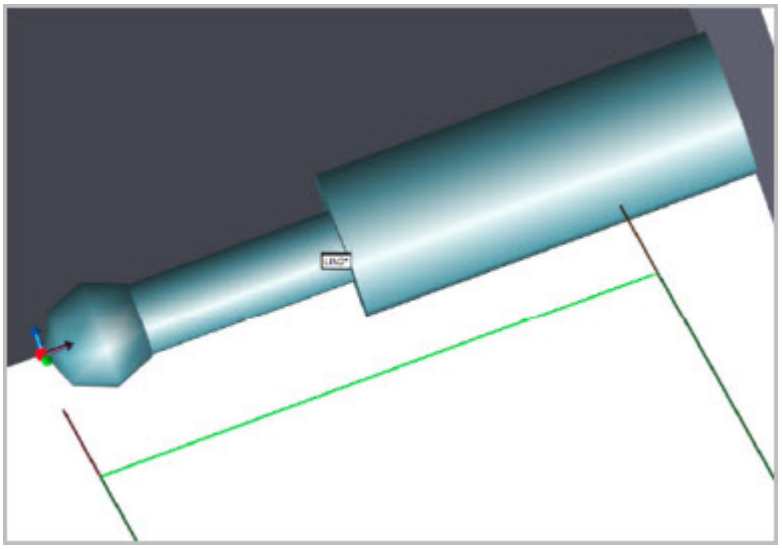
As an example, the first image shows the behavior of a line created right on the edge of two different surfaces with no Depth or Void Detection enabled.



Example of a line feature generated with Void Detection option disabled.

In this case, PC-DMIS places the hits for the line just on the edge. If you enable Depth Detection without Void Detection enabled, the hits are as far from the edge as the depth parameter.

The next image shows the behavior when Void Detection is enabled on the same line and Depth Detection is set to 0 (zero).



Example of a line feature generated with Void Detection option enabled.

The Void Detection algorithm is designed to add some intelligence to the hits distribution. Due to the instability of calculating a line on or very near to an edge, a "security distance" is determined. PC-DMIS then uses this to place the hits for the feature. The "security distance" is based on a multiple of the probe's radius.

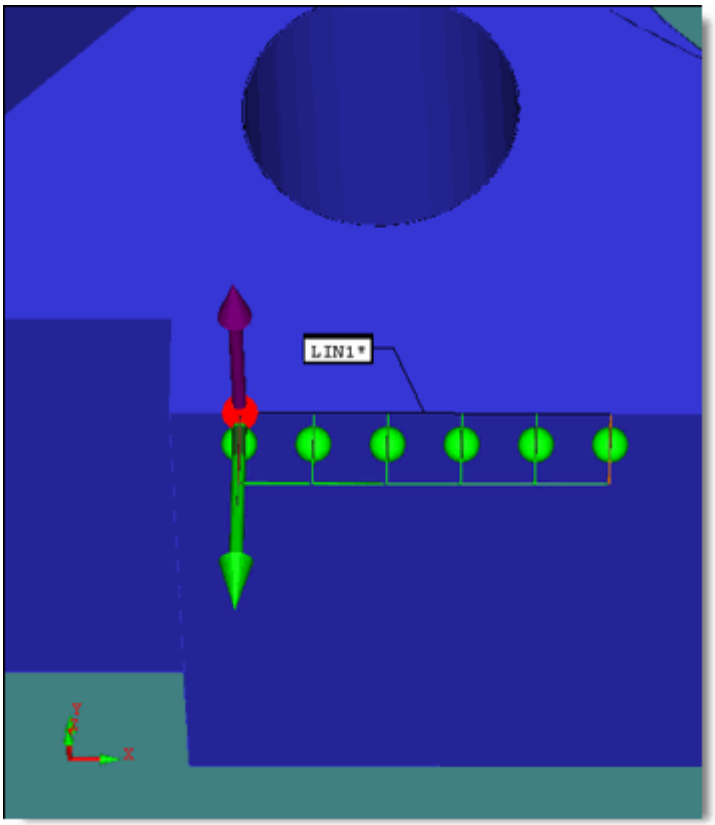
In QuickFeature mode, you can select multiple colinear or coplanar entities. Void Detection creates a hit pattern that considers all selected entities. For details on how to turn this function On and Off, see the "VoidDetectionNewAlgorithm" topic in the Settings Editor documentation.

You can also drag-select hits to determine an edge. Note that when you drag-select hits with **Void Detection** set to **On**, the Void Detection algorithm is still used to calculate the starting point of the drag operation. Once dragging starts, however, the **Void Detection** option is automatically set to **Off**, and the remaining hits are detected manually. The Void Detection algorithm is used to intelligently define the path while drag-selecting manually determines the path.

Show Measured Points

This icon only functions with features that are already measured. Until a feature is measured, either with the **Test** button on the **Auto Feature** dialog box or with the actual measurement routine execution, the icon remains unavailable for selection.

Select this icon to show, in the Graphic Display window, a visual depiction of the data points used to measure the feature.



Example showing the measured points of an auto line feature.

Show Filtered Points

> This icon only functions if you have PC-DMIS Vision enabled on your PC-DMIS license. You also need to have a feature with outlier filtering enabled and some filtered points.

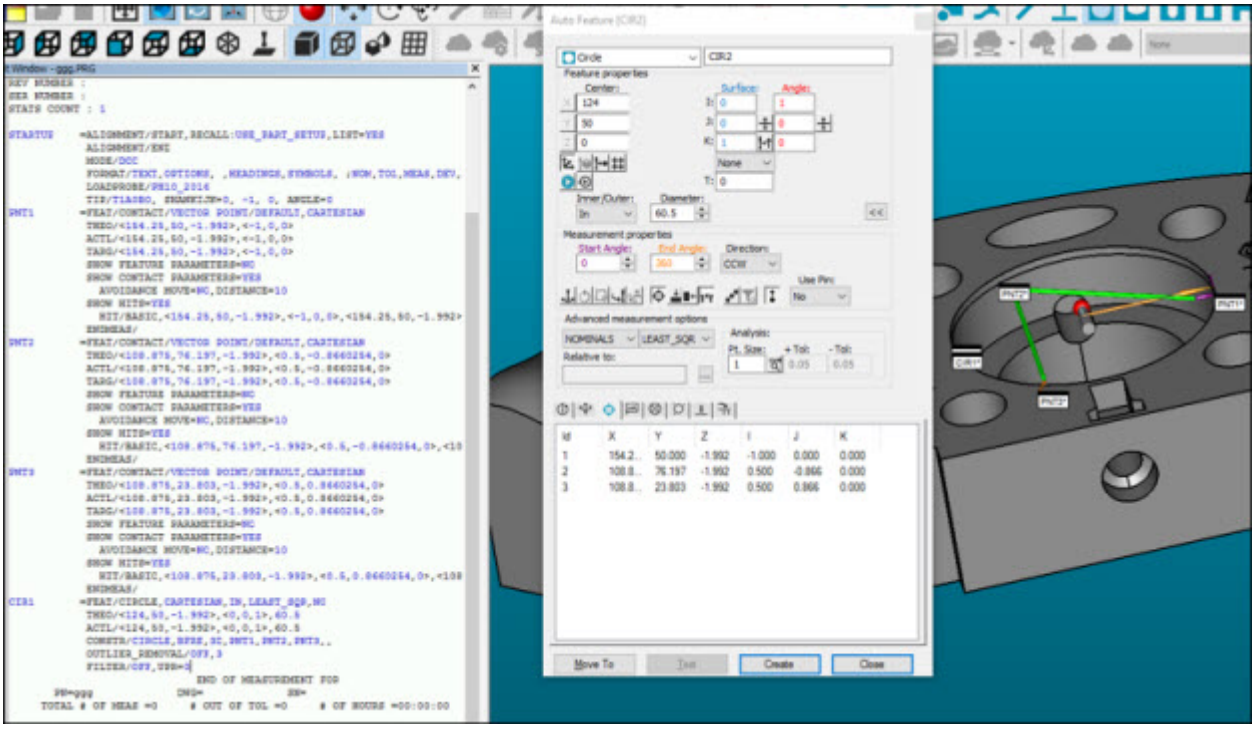
Select this icon to show on the Live and CAD Views image processing data points that were acquired and then discarded by the current filter settings. For additional information, see the PC-DMIS Vision documentation.

To Points

> The **To Points** function is available for auto contact features of type Plane, Circle, and Cylinder.

If you select the **To Points** icon on the Auto feature bar, PC-DMIS creates the feature as individual Vector Points with the associated shape constructions. PC-DMIS uses the hits of the Auto feature to obtain the Vector Points.

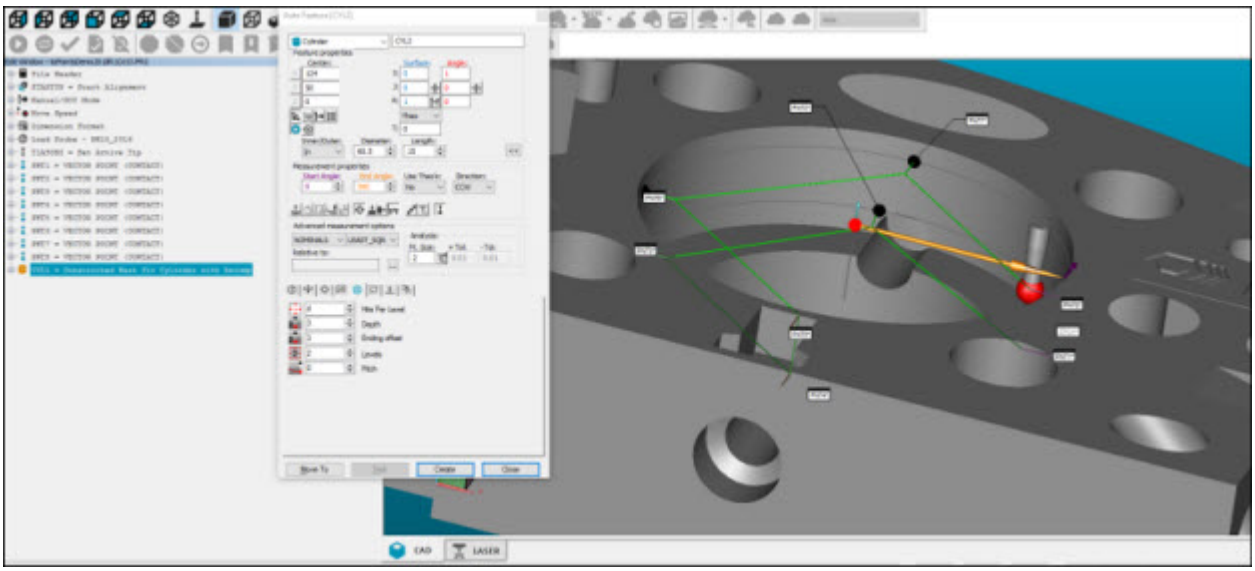
PC-DMIS applies the Best Fit Recompensate algorithm to the Vector Points to compute the constructed feature. Following is an example for a Circle feature:



Example of a circle feature generated with the To Points option enabled.

For more information about the Best Fit Recompensate algorithm, see the "Constructing a Best Fit or Best Fit Recompensate Circle" topic.

If the **Void Detection** function is enabled for the Auto feature, or if you drag the hits (user-defined hits), the resulting Vector Points will take care of it. For example:



Example of a cylinder feature generated with the Void Detection and To Points options enabled.

The **To Points** function also works with QuickFeatures. It is feature-type dependent. For example, you can enable the icon for circle features and disable it for cylinder and plane features.

After you create the measurement routine, the "optimize path" task can be used to assign the probe head orientation and to re-order the vector points. This feature enables PC-DMIS to measure all of the points with the same tip in one step even if they belong to different features. For more information about path optimization, see the "Optimizing the Path" topic.

PC-DMIS ignores some Auto feature measurement options (such as auto wrist, circular moves, clearance plane, and sample hits).

PC-DMIS moves some of the measurement options from the original Auto feature to the resulting Vector Points (such as the coordinate types and the thickness).

The **Test** button is unavailable if the **To Points** option is enabled.

Extended Sheet Metal Options Area

The **Extended sheet metal options** area of the **Auto Feature** dialog box contains some of the less frequently-used sheet metal options for supported Auto features.

Showing Extended Sheet Metal Options

The **Extended sheet metal options** area is hidden until the following conditions are met:

- You must select the **Show Extended Sheet Metal Options** check box on the **General** tab in the **Setup Options** dialog box (select **Edit | Preferences | Setup** or press the F5 key to access this dialog box.
- The type of Auto feature must support the extended options.

> Except for Corner Point and High Point, all other Contact and laser Auto features support the **Extended Sheet Metal Options** area.

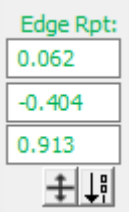
- You must select the **>>** button in the **Auto Feature** dialog box for Show Advanced Sheet Metal Options.
- You must select the **>>** button in the **Auto Feature** dialog box for Show Extended Sheet Metal Options. This button appears only for supported features.

This displays the **Auto Feature** dialog box with all of the available options visible.

Extended Sheet Metal Options

Depending on the feature you selected, the following items appear in the **Extended sheet metal options** area.

IJK Edge Report boxes



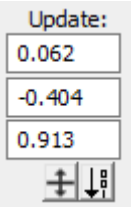
These boxes display the vector used for reporting deviation. This is a user-supplied I, J, K vector.

For the Line and Edge Point features, these boxes display the vector for the **RT** computation.

The Location Dimension option **RT** displays the deviation calculated along this vector. After the new value is created, PC-DMIS will normalize the vector, making its length one unit.

Supported Auto features: **Line, Edge Point**

IJK Update boxes

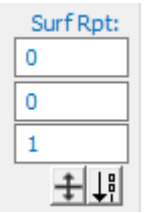


These boxes display the updated vector that PC-DMIS uses to pierce the CAD surface, if the option is active by setting the SET NOM AX mode. This is a user-supplied I, J, K normal vector.

The I, J, K should always point away from the surface. After the new value is created, PC-DMIS will normalize the vector, making its length one unit.

Supported Auto features: **Vector Point**

IJK Surf Rpt boxes



These boxes display the vector used for reporting deviation. This is a user-supplied I, J, K vector.

- For the Line and Edge Point features, these boxes display the vector for the **RS** computation.
- For the Angle Point feature, these boxes display the vector for the **RT** computation. You can use these boxes to deviate from the S and T deviations on the dimension for an Angle Point feature.
- For features that display only the **IJK Surf Rpt** boxes (they do not display the **IJK Edge Report** boxes), these boxes display the vector for the **RT** computation.

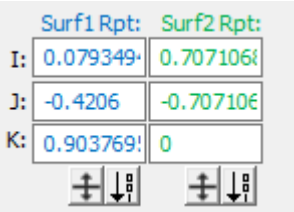
After the new value is created and you exit the dialog box, PC-DMIS will normalize the vector, making its length one unit.

The Location dimension's **RS** check box displays the deviation calculated along this vector.

The Location dimension's **RT** check box displays the measured deviation in the surface vector direction along this defined report vector.

Supported Auto features: **All features, except Corner Point, High Point, and Sphere**

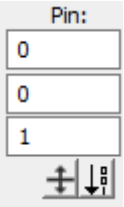
IJK Surf1 Rpt and Surf2 Rpt boxes



These boxes display the surface vectors for the **RS** (deviation along surface report) computation. You can use these boxes to deviate from the S and T deviations on the dimension for an Angle Point feature.

Supported Auto features: **Angle Point**

IJK Pin boxes

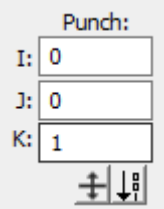


These boxes define the vector of the stud through the hole formed by the punch.

When pins are created on a sheet metal surface, they are not always perpendicular to the surface. This naturally creates an elliptical shape at the surface of the metal, even though a round pin may have been used. The **Pin** vector then allows for more accurate measurement and data analysis in this situation.

Supported Auto features: **Circle, Square Slot, Round Slot**

IJK Punch boxes



These boxes define the vector of the punch through the sheet metal. This vector is positioned at the XYZ center, plus half the thickness along the surface normal vector.

The punches used to place holes in sheet metal are not always perpendicular to the surface. This naturally creates an elliptical shape at the surface of the metal, even though a round punch may have been used. The **Punch** vector then allows for more accurate measurement and data analysis in this situation.

For the Circle feature, the diameter of the feature is also along this vector.

Supported Auto features: **Circle, Square Slot, Round Slot**

For more information on these vector boxes, see "Extended Sheet Metal Vector Diagrams".

Icon	Description
	Flip Vector
	Reset Vector to Surface Vector

Reset Vector to Surface Vector

The **Reset Vector to Surface Vector** icon will set the vector to match the normal Surface vector values.

Extended Sheet Metal Vector Diagrams

This topic contains various diagrams illustrating the vectors that you can use when you choose the show extended sheet metal feature options:

Normal Vector: The normal vector is the vector perpendicular to the surface at a point feature's location. See the diagram below:

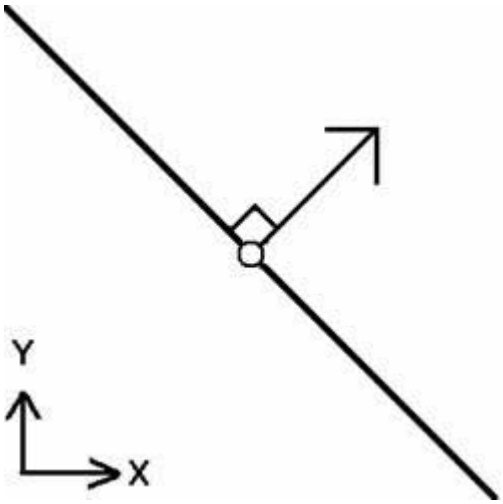


Diagram of a Normal Vector

	= Normal Vector
	= Surface
	= Point Location

Update Vector: The update vector is the vector used to determine the direction to follow when updating a point to a new surface. This update vector is derived from the reference line used in the initial creation of the feature. See the diagram below:

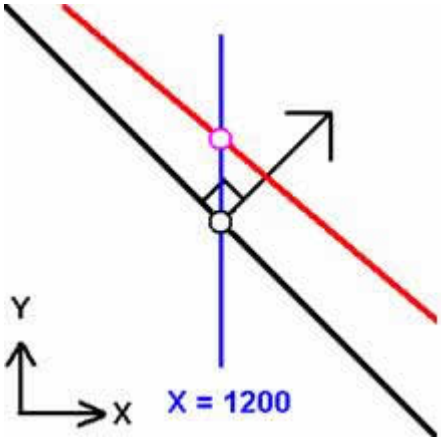


Diagram of an Update Vector

	= Updated or new surface
	= Original or old surface
	= Original point location
	= Updated point location
	= Reference line used to generate point location. Also known as the update vector.

Report Vector: The report vector lets you control the direction of the deviation by allowing you to define a vector onto which the deviation along the surface vector is then projected. The report vector may differ from the surface normal, such as along a specified axis (shown below as either Yr or Xr). See the diagrams below:

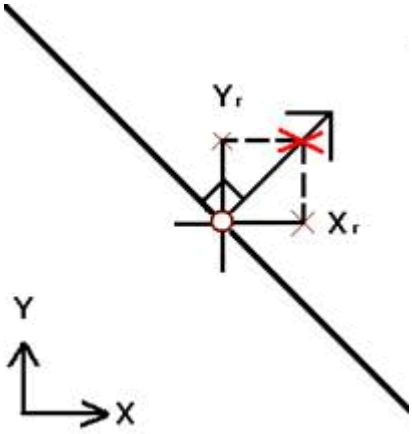


Diagram of a Report Vector

———	**= Theoretical surface**
o	**= Nominal point location**
×	**= Actual point location**
X_r	**= Deviation in X axis**
Y_r	**= Deviation in Y axis**

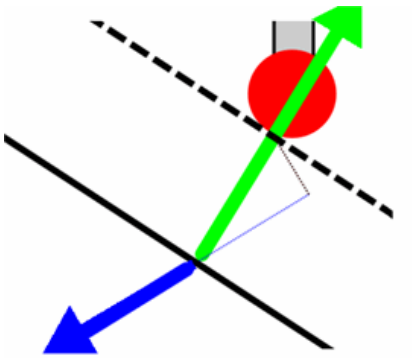


Diagram depicting a surface vector deviation projected onto the report vector

	= Theoretical surface
	=Actual surface
	=Surface vector
	=Report vector
	=Surface vector deviation projected to the report vector
	=Probe tip contacting actual surface

Pin Vector: Applied to slots and holes, the pin vector specifies the vector of the pin as it locates the product. See the diagram below:

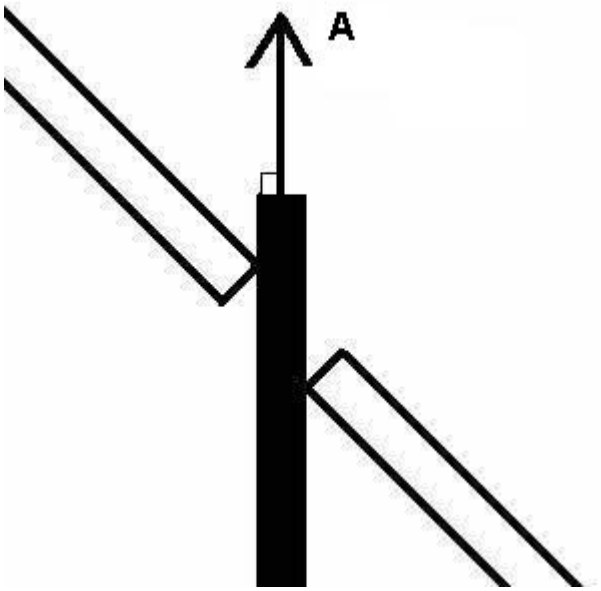


Diagram of a Pin Vector
***A** = Pin Vector*

Punch Vector: Applied to slots and holes, the punch vector specifies the direction of the punch used to create the feature. This vector is usually close to the surface normal within a few degrees. See the diagram below:

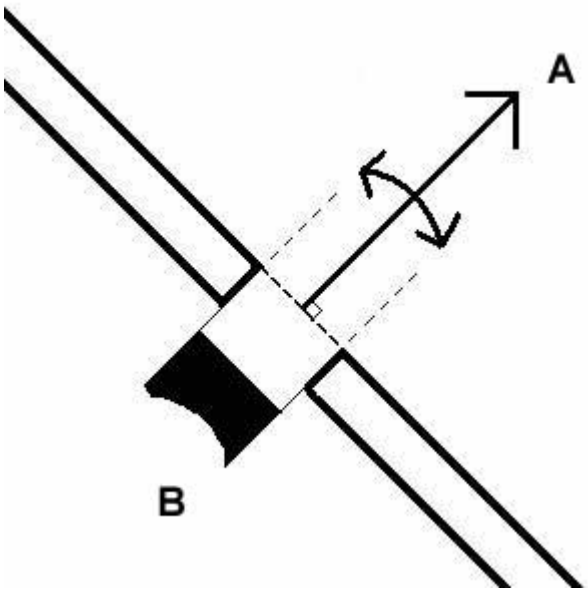
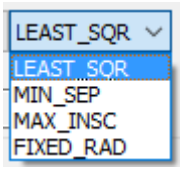


Diagram of a Punch Vector
***A** = Punch Vector. Close to the surface normal within +/- 5 degrees*
***B** = The Punch*

Advanced Measurement Options Area

The **Measurement Properties** area of the **Auto Feature** dialog box contains some or all of the following items, depending on the Auto feature you selected.

Calculation List



You can use the **Calculation** list in the **Advanced measurement options** area to specify how you would like the feature calculated from the measured hits. This list is available for the Circle, Cylinder, and Plane Auto features.

PC-DMIS dynamically filters the available options. The list of options depend on the type of feature that you select and whether the feature is an Inner (hole) or Outer (stud) feature type.

The available options for Circle and Cylinder features are:

LEAST_SQR, MIN_SEP, MAX_INSC, MIN_CIRCSC, and FIXED_RAD

The available options for the Plane feature are:

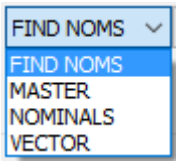
LEAST_SQR and MIN_SEP

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

These calculation types are already discussed in detail in the "Best Fit Type" topic in the "Constructing New Features from Existing Features" chapter.

Nominals Mode List



The **Mode** list in the **Advanced Measurement Options** area determines how PC-DMIS computes the nominals for a given point. PC-DMIS allows you to choose between FIND NOMS, MASTER, NOMINALS, VECTOR, and SET NOM AX.

- If `MODE = FIND NOMS` is active, PC-DMIS pierces the CAD model to find the closest location on a CAD edge (or surface) to the measured point and sets the nominals to that location on the CAD element.
- If `MODE = MASTER` is active, PC-DMIS uses the measured feature as the nominal, but PC-DMIS does not update the X, Y, Z and diameter data in the dialog box.
- If `MODE = NOMINALS` is active, PC-DMIS compares the measured feature with the theoretical data in the dialog box. It uses the measured data for calculations.
- If `MODE = VECTOR` is active, PC-DMIS uses your first three hits to calculate the appropriate vector to use for the feature. PC-DMIS does not adjust the location of the feature. You must take the hits with the **Auto Features** dialog box active.
- If `MODE = SET NOM AX` , PC-DMIS updates the nominal point location and the nominal approach direction before each execution.

For each mode, see the more detailed description in the table below if you need more information.

If a nominal value is not known, select **FIND NOMS** from the list. While this option is active, every time you select a value to change, PC-DMIS prompts you to take the appropriate measurements on the part to obtain the values needed.

FIND NOMS
When you select **FIND NOMS** from the **Mode** list, PC-DMIS pierces the CAD model to find the closest location on a CAD edge (or surface) to the measured point, and sets the nominals to that location on the CAD element.

To find the nominals using CAD data:

1. Verify that PC-DMIS is set to **Curve** mode or **Surface** mode (available from the Graphic Modes toolbar), as is appropriate for the particular measurement routine.
2. Click on the desired CAD element in the Graphic Display window. PC-DMIS pierces the CAD element to obtain the location, but does not create a feature. Once the desired elements have been selected, PC-DMIS automatically fills in the X, Y, Z and I, J, K values.
3. If the values are satisfactory, select the **Create** command button.

If you select the **Measure** check box, PC-DMIS measures the feature during creation. If you have not added clearance moves, this may cause a probe collision with the part.

If you take a hit on the part while in **FIND NOMS** mode, PC-DMIS searches through the CAD elements and finds the nominal CAD information that is closest to the measured point. If necessary, PC-DMIS prompts you to take any additional hits on the part.

The next time you measure the part, PC-DMIS sets the nominal data to the nearest CAD element it can find. The mode switch is then reset to **NOMINALS**.

Using Find Noms with Fixed Probes

A Faro or Romer arm using a fixed probe doesn't generate good approach vectors. Because of this, PC-DMIS cannot easily determine where to look for the surfaces.

To improve vectors from fixed probes:

1. Place the fixed probe on the part.
2. Press the **Hit** button.

3. Move the probe away from the part along the approximate surface vector.
4. Press the **End** button.

PC-DMIS calculates and then uses the vector between the hit and the probe tip's position.

Also, because the default vector from a Faro arm uses the axis of the fixed probe, the more you position the vector normal to the surface, the more useful the vector is for Find Noms operations.

MASTER
If you create a point when the **Mode** list is set to **MASTER**, then the next time you measure the part, PC-DMIS sets the nominal data equal to the measured data. The **Mode** list is then reset to **NOMINALS**.

NOMINALS
The **NOMINALS** option also requires that you have nominal data before you start the measurement routine. PC-DMIS compares the measured feature with the theoretical data in the dialog box. It uses the measured feature for any necessary calculations.

VECTOR
The **VECTOR** option allows you to update only the feature's vector during learn mode; it does not update the nominal XYZ values.

This option is *only* available for the Vector Point and Surface Point Auto features.

This option helps you to set a feature's vector that you may not be able to obtain otherwise. With the dialog box open, take three hits on the feature. This determines its vector.

You can use this mode as long as the dialog box remains open. Once you close the dialog box, the option is not available for the feature in the Edit window.

Supported Features: Vector Point, Surface Point, Corner Point, Line, Plane, Circle, Ellipse, Round Slot, Square Slot, Notch Slot, Polygon, Cylinder, Cone, Sphere

SET NOM AX
The **SET NOM AX** option updates (or "sets") the nominal point location and the nominal approach direction before each execution. If you select an axis check box from the **Location** area, PC-DMIS pierces the CAD surfaces along that axis. Otherwise, PC-DMIS uses the updated vector.

This option is *only* available for the Vector Point and Surface Point Auto features.

The axis (or vector) selected tells PC-DMIS along which axis (or vector) to pierce the CAD surface to find a new theo and target.

To select the **SET NOM AX** option:

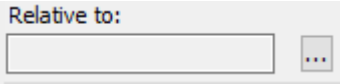
1. Make sure the **Show Extended Sheet Metal Option** check box is selected. See "Show Extended Sheet Metal Options" in the "Setting Your Preferences" chapter.
2. If desired, select one of the axis check boxes from the **Location** area.
3. Click **SET NOM AX** from the **Mode** list in the dialog box.
4. When you are done defining the rest of the Auto feature, click the **Create** button. The FIND NOM AXIS is then set to either the axis or the vector selected.

The Edit window for this option reads: FIND NOM AXIS = TOG

TOG represents the axis or vector to which FIND NOM AXIS is set. Available options for TOG include XAXIS, YAXIS, ZAXIS, VEC, and NONE.

If you don't select any axis, then the result reads FIND NOM AXIS = VEC by default.

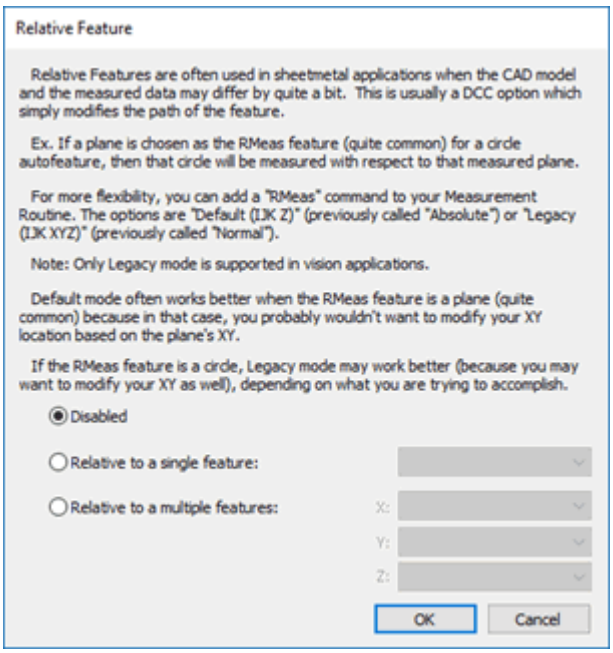
Relative To



Using the **Relative to** portion of the **Auto Feature** dialog box, you can keep the relative position and orientation between a chosen feature or features and the current auto feature. Before selecting a relative feature, make sure you choose a relative measure mode. For information on how to set up the relative measure mode, see "Setting Up Relative Measure".

The relative feature or features you choose must already exist in the measurement routine.

You can click the **…** button to display a **Relative Feature** dialog box and choose the relative feature (or features):



Relative Feature Dialog Box

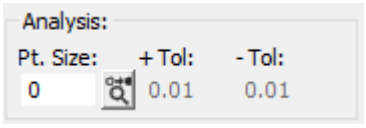
From this dialog box, you can choose to select either:

- A single feature from the list to the right
- Multiple features (one for each axis) from the lists to the right

After you select the relative feature or features and click **OK**, the **Relative To** box displays what you chose.

This option supports the DMIS V3.0 RMEAS formats 1, 3, and 6.

Analysis Area



The **Analysis** area allows you to determine how each measured hit/point is displayed.

This functionality was originally created for PC-DMIS Vision. For in-depth information on its usage, see the **Analysis area** section of the "Advanced Measurement Options Area" topic in the PC-DMIS Vision documentation.

A Note on Point and Arrow Vector Size

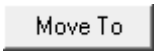
If your vector arrows for your Auto Point features seem too small, increasing the **Pt. Size** value increases both the size of the point in the Graphic Display window and the corresponding size of the vector arrow. Since Auto features are used on widely differing

machines, no one default size works for everyone. However, you can decide what size works best for you. PC-DMIS then defaults to your last entered value.

If your point appears like a large blob on your screen, set the **Pt. Size** value to 0. This generally produces a good-looking point size and vector arrow.

Auto Feature Command Buttons

Move To button



This moves the field of view in the Graphic Display window and centers it on the current feature's XYZ location.

In some cases, such as a Vector Point, the XYZ location is actually offset along the normal vector by the pre-hit value. This prevents the probe from crashing into the part.

If a feature is composed of more than one point (such as a line), then clicking this button switches between the points making up the feature.

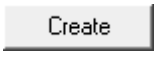
Test button

This tests a feature's creation and previews its dimensional data before it's actually created. It performs a measurement using the current parameters.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

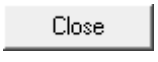
You can change parameters and click **Test** repeatedly until you have an acceptable measurement. Then when you click **Create,** the software converts the temporary feature into a normal feature in the measurement routine.

Create button



This inserts the defined Auto feature into the Edit Window at the current position.

Close button



This closes the **Auto Feature** dialog box.

Show Advanced Measurement Options button

This extends the **Auto Feature** dialog box and displays any available advanced measurement options. The button then changes to the **Hide Advanced Measurement Options** button.

Hide Advanced Measurement Options button

This shortens the **Auto Feature** dialog box and only shows the basic options for that Auto feature. The button then changes to the **Show Advanced Measurement Options** button.

Show Extended Sheet Metal Measurement Options button

For supported features, this displays the **Extended Sheet Metal Options** area. The button then changes to **Hide Extended Sheet Metal Measurement Options** button.

Hide Extended Sheet Metal Measurement Options button

This hides the **Extended Sheet Metal Options** area. The button then changes to the **Show Extended Sheet Metal Measurement Options** button.

Inserting Auto Features

The PC-DMIS configurations that support Auto features differ in which Auto features they support as well as how those features are created. Because of this, information on creating and inserting Auto Features into your measurement routine is not covered here. Instead, consult the documentation set for your PC-DMIS configuration from the list below:

Contact (PC-DMIS CMM)	Vision (PC-DMIS Vision)	Laser (PC-DMIS Laser)
Auto Vector Point		
Auto Surface Point	Auto Surface Point	Auto Surface Point
Auto Edge Point	Auto Edge Point	Auto Edge Point
Auto Angle Point		
Auto Corner Point		
Auto High Point		Auto High Point
Auto Line	Auto Line	
Auto Plane		Auto Plane
Auto Circle	Auto Circle	Auto Circle
Auto Ellipse	Auto Ellipse	
Auto Square Slot	Auto Square Slot	Auto Square Slot
Auto Round Slot	Auto Round Slot	Auto Round Slot
Auto Notch Slot	Auto Notch Slot	
Auto Polygon	Auto Polygon	Auto Polygon

Auto Cylinder		Auto Cylinder
Auto Cone		Auto Cone
Auto Sphere		Auto Sphere
		Auto Flush and Gap
Probe Toolbox Information		
Contact Probe Toolbox	Vision Probe Toolbox	Laser Probe Toolbox

Once an Auto feature is created, its command appears inside the Edit window (see "Auto Feature Field Definitions"). You can then mark the command and have PC-DMIS execute it like any other command or feature.

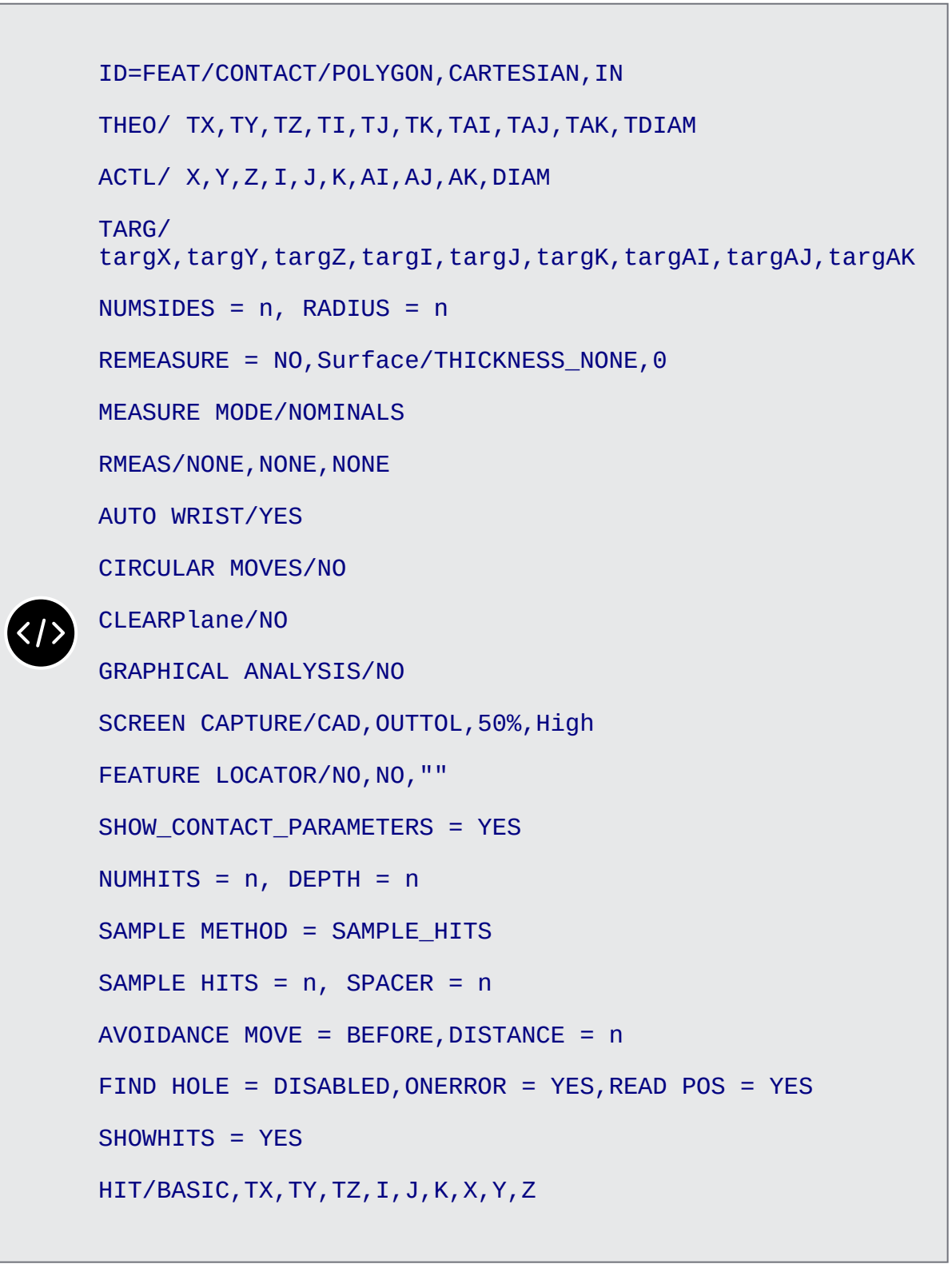
Auto Feature Field Definitions

When you create an Auto feature, PC-DMIS inserts the command for that feature into the Edit window. This topic documents the different fields that may appear in the Command mode of the Edit window for the different features.

In the table below, locate the field or command line used in your Auto Feature to see what it does.

Auto Polygon Field Definitions

The Edit window command line for an Auto Polygon feature would read:



```
ID=FEAT/CONTACT/POLYGON,CARTESIAN,IN
THEO/ TX,TY,TZ,TI,TJ,TK,TAI,TAJ,TAK,TDIAM
ACTL/ X,Y,Z,I,J,K,AI,AJ,AK,DIAM
TARG/
targX,targY,targZ,targI,targJ,targK,targAI,targAJ,targAK
NUMSIDES = n, RADIUS = n
REMEASURE = NO,Surface/THICKNESS_NONE,0
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n, DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED,ONERROR = YES,READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Sphere Field Definitions

The Edit window command line for an Auto Sphere feature would read:

```
ID=FEAT/CONTACT/SPHERE,CARTESIAN,IN,LEAST_SQR
THEO/ TX,TY,TZ,TI,TJ,TK,TDIAM
ACTL/ X,Y,Z,I,J,K,DIAM
TARG/ targX,targY,targZ,targI,targJ,targK
START ANG1 = n, END ANG1 = n
START ANG2 = n, END ANG2 = n
Angle VEC = I, J, K
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n,NUMLROWS = n,
SAMPLE HITS = n,
AVOIDANCE MOVE = BEFORE,DISTANCE = n
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Cone Field Definitions

The Edit window command line for an Auto Cone feature would read:

```
ID=FEAT/CONTACT/CONE,CARTESIAN,IN
THEO/ TX,TY,TZ,TI,TJ,TK,TAngle,TLENGTH,TDIAM
ACTL/ X,Y,Z,I,J,K,Angle,LENGTH,DIAM
TARG/ targX,targY,targZ,targI,targJ,targK
START ANG = n, END ANG = n
Angle VEC = I, J, K
Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n,NUMLEVELS = n, STARTING DEPTH = n, ENDING
DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
ONERROR = NO, READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Cylinder Field Definitions

The Edit window command line for an Auto Cylinder feature would read:

```
ID=FEAT/CONTACT/CYLINDER,CARTESIAN,IN,LEAST_SQR
THEO/ TX,TY,TZ,TI,TJ,TK,TDIAM,TLENGTH
ACTL/ X,Y,Z,I,J,K,DIAM,LENGTH
TARG/ targX,targY,targZ,targI,targJ,targK
START ANG = n, END ANG = n
REMEASURE = NO, USE THEO = YES
Angle VEC = I, J, K
DIRECTION = CCW
Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n,NUMLEVELS = n, STARTING DEPTH = n, ENDING
DEPTH = n, PITCH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED, ONERROR = NO, READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Notch Slot Field Definitions

The Edit window command line for an Auto Notch Slot feature would read:

```
ID=FEAT/CONTACT/NOTCH SLOT,CARTESIAN
THEO/ TX,TY,TZ,TI,TJ,TK,TAI,TAJ,TAK,TWIDTH,TLENGTH
ACTL/ X,Y,Z,I,J,K,AI,AJ,AK,WIDTH,LENGTH
TARG/ targX,targY,targZ,targI,targJ,targK
REMEASURE = NO
Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED, ONERROR = NO, READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Square Slot Field Definitions

The Edit window command line for an Auto Square Slot feature with extended options would read:

```
ID=FEAT/CONTACT/SQUARE SLOT,CARTESIAN,IN
THEO/ TX,TY,TZ,TI,TJ,TK,TAI,TAJ,TAK,TWIDTH,TLENGTH
ACTL/ X,Y,Z,I,J,K,AI,AJ,AK,WIDTH,LENGTH
TARG/
targX,targY,targZ,targI,targJ,targK,targAI,targAJ,targAK
MEAS WIDTH = YES, RADIUS = n
REMEASURE = NO
PUNCH = I,J,K,PIN = I,J,K Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED, ONERROR = NO, READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Round Slot Field Definitions

The Edit window command line for an Auto Round Slot feature with extended options would read:

```
ID=FEAT/CONTACT/ROUND SLOT,CARTESIAN,IN
THEO/ TX,TY,TZ,TI,TJ,TK,TAI,TAJ,TAK,TWIDTH,TLENGTH
ACTL/ X,Y,Z,I,J,K,AI,AJ,AK,WIDTH,LENGTH
TARG/
targX,targY,targZ,targI,targJ,targK,targAI,targAJ,targAK
MEAS Angle = n
REMEASURE = NO
PUNCH = I,J,K,PIN = I,J,K Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n, DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED, ONERROR = NO, READ POS = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Ellipse Field Definitions

The Edit window command line for an Auto Ellipse feature would read:

```
ID=FEAT/CONTACT/ELLIPSE,CARTESIAN,IN
THEO/ TX,TY,TZ,TI,TJ,TK,TDIAM,TDIAM2,TAI,TAJ,TAK
ACTL/ X,Y,Z,I,J,K,DIAM,DIAM2,AI,AJ,AK
TARG/
targX,targY,targZ,targI,targJ,targK,targAI,targAJ,targAK
START ANG = n,END ANG = n
Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n, DEPTH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
ONERROR = NO, READ POS = NO
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Circle Field Definitions

The Edit window command line for an Auto Circle feature with extended options would read:

```
ID=FEAT/CONTACT/CIRCLE,CARTESIAN,IN,LEAST_SQR
THEO/ TX,TY,TZ,TI,TJ,TK,TDIAM,TANG1,TANG2
ACTL/ X,Y,Z,I,J,K,DIAM,ANG1,ANG2
TARG/ targX,targY,targZ,targI,targJ,targK
AngleVEC = I,J,K
DIRECTION = CCW
REMEASURE = NO
PUNCH = I,J,K, PIN = I,J,K
Surface/ACTL_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CIRCULAR MOVES/NO
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n, DEPTH = n, PITCH = n
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
FIND HOLE = DISABLED, ONERROR = NO, READ POS = NO
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Plane Field Definitions

The Edit window command line for an Auto Plane feature would read:

```
ID=FEAT/CONTACT/PLANE,CARTESIAN,TRIAngle,LEAST_SQR
THEO/ TX,TY,TZ,TI,TJ,TK
ACTL/ X,Y,Z,I,J,K
TARG/ targX,targY,targZ,targI,targJ,targK
AngleVEC = I, J, K, Square
Surface/THEO_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
NUMHITS = n, NUMROWS = n
SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
SHOWHITS = YES
USE BOUNDARY OFFSET=YES, OFFSET=n
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Line Field Definitions

The Edit window command line for an Auto Line feature would read:

```
ID=FEAT/CONTACT/LINE,CARTESIAN

THEO/
TX,TY,TZ,TI,TJ,TK,TLI,TLJ,TLK,TEI,TEJ,TEK,TSI,TSJ,TSK,TL
ENGTH

ACTL/ X,Y,Z,I,J,K,LI,LJ,LK,EI,EJ,EK,SI,SJ,SK,TLENGTH

TARG/ targX,targY,targZ,targI,targJ,targK

BOUNDED

REPORT VEC = I,J,K

Edge/THEO_THICKNESS,n

Surface/THEO_THICKNESS,n

MEASURE MODE/NOMINALS

RMEAS/NONE,NONE,NONE

AUTO WRIST/YES

CLEARPlane/YES

GRAPHICAL ANALYSIS/YES,n,n,n

SCREEN CAPTURE/CAD,OUTTOL,50%,High

FEATURE LOCATOR/NO,NO,""

SHOW_CONTACT_PARAMETERS = YES

NUMHITS = n, DEPTH = n

AVOIDANCE MOVE = BEFORE,DISTANCE = n

SHOWHITS = YES

HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto High Point Field Definitions

The Edit window command line for an Auto High Point feature would read:

```
ID=FEAT/CONTACT/HIGH POINT,CARTESIAN

THEO/ TX,TY,TZ,TI,TJ,TK

MEAS/ X,Y,Z,I,J,K

TARG/ targX,targY,targZ,targI,targJ,targK

INCREMENT = n, TOL = n, CIRCULAR,OUTER RADIUS = n, INNER
RADIUS = n

CENTER = X,Y,Z

Surface/THEO_THICKNESS,n

MEASURE MODE/NOMINALS

RMEAS/NONE,NONE,NONE

AUTO WRIST/YES

CLEARPlane/NO

GRAPHICAL ANALYSIS/NO

SCREEN CAPTURE/CAD,OUTTOL,50%,High

FEATURE LOCATOR/NO,NO,""

SHOW_CONTACT_PARAMETERS = YES

AVOIDANCE MOVE = BEFORE,DISTANCE = n
```

Auto Corner Point Field Definitions

The Edit window command line for an Auto Corner Point feature would read:

```
ID=FEAT/CONTACT/CORNER POINT,CARTESIAN
THEO/ TX,TY,TZ,TSI,TSJ,TSK,TS2I,TS2J,TS2K,TS3I,TS3J,TS3K
MEAS/ X,Y,Z,I,J,K,SI,SJ,SK,S2I,S2J,S2K,S2I,S2J,S2K
TARG/ targX,targY,targZ,targI,targJ,targK
EXTERIOR
Surface2/THEO_THICKNESS,n
Surface3/THEO_THICKNESS,n
Surface/THEO_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
SPACER = n, INDENT1 = n, INDENT2 = n, INDENT3 = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
ONERROR = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Angle Point Field Definitions

The Edit window command block for an Auto Angle Point would read:

```
ID=FEAT/CONTACT/ANGLE POINT,CARTESIAN
THEO/ TX,TY,TZ,TI,TJ,TK,TSI,TSJ,TSK,TS2I,TS2J,TS2K
MEAS/ X,Y,Z,I,J,K,SI,SJ,SK,S2I,S2J,S2K
TARG/ targX,targY,targZ,targI,targJ,targK
EXTERIOR
Surface2/THEO_THICKNESS,n
Surface/THEO_THICKNESS,n
MEASURE MODE/FINDNOMS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
SAMPLE HITS = n, SPACER = n, INDENT1 = n, INDENT2 = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
ONERROR = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Edge Point Field Definitions

The Edit window command block for an Auto Edge Point feature with extended options would read:

```
ID=FEAT/CONTACT/EDGE POINT,CARTESIAN
THEO/ TX,TY,TZ,TI,TJ,TK
ACTL/ X,Y,Z,I,J,K
TARG/ targX,targY,targZ,targI,targJ,targK
Edge/THEO_THICKNESS,n
REPORT = I, J, K, Surface_REPORT = I, J, K
MEASURE ORDER = Surface
/THEO_THICKNESS,n
MEASURE MODE/FINDNOMS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
DEPTH = n
SAMPLE HITS = n, SPACER = n, INDENT1 = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
ONERROR = YES
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Surface Point Field Definitions

The Edit window command block for an Auto Surface Point feature with extended options would read:

```
ID=FEAT/CONTACT/SURFACE POINT,CARTESIAN
THEO/ TX,TY,TZ,TI,TJ,TK
ACTL/ X,Y,Z,I,J,K
TARG/ targX,targY,targZ,targI,targJ,targK
REPORT VEC = I,J,K
Surface/THEO_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/YES
GRAPHICAL ANALYSIS/YES,n,n,n
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
SAMPLE METHOD = SAMPLE_HITS
SAMPLE HITS = n, SPACER = n
AVOIDANCE MOVE = BEFORE,DISTANCE = n
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Auto Vector Point Field Definitions

The Edit window command block for an Auto Vector Point feature with extended options would read:

```
ID=FEAT/CONTACT/VECTOR POINT,CARTESIAN
THEO/ TX,TY,TZ,TI,TJ,TK
ACTL/ X,Y,Z,I,J,K
TARG/ targX,targY,targZ,targI,targJ,targK
REPORT VEC = I,J,K, UPDATE VEC = I,J,K
Surface/THEO_THICKNESS,n
MEASURE MODE/NOMINALS
RMEAS/NONE,NONE,NONE
AUTO WRIST/YES
CLEARPlane/NO
GRAPHICAL ANALYSIS/NO
SCREEN CAPTURE/CAD,OUTTOL,50%,High
FEATURE LOCATOR/NO,NO,""
SHOW_CONTACT_PARAMETERS = YES
AVOIDANCE MOVE = BEFORE,DISTANCE = n
SHOWHITS = YES
HIT/BASIC,TX,TY,TZ,I,J,K,X,Y,Z
```

Fields or commands in this documentation that are marked with ● relate to Extended Fields ⓘ.

A red bullet (●) in the documentation below indicates that this field only appears in the Edit window if you have enabled the **Show Extended Sheet Metal Options** check box found on the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup** or press F5 to access the dialog box). For more information, see "Show Extended Sheet Metal Options".

ID
This field shows the feature's identification label. See "ID box".

FEAT/CONTACT
This command displays the type of probing and the Auto feature type. See "Auto feature Type list".

POLAR or CARTESIAN
This field switches between POLAR and CARTESIAN and displays the X, Y, Z, I, J, K values in the selected coordinate system. See "Polar / Cartesian Toggle".

TRIANGLE or OUTLINE
For a Plane feature, this field switches between TRIANGLE or OUTLINE. It determines how PC-DMIS displays the plane in the Graphic Display window. See "Display list".

Used only on the **Plane** feature.

IN or **OUT**
This field switches between IN and OUT. It determines whether or not the feature is an internal feature (such as a hole) or an external feature (such as a stud). See "Inner / Outer".

Used only on these features: **Circle**, **Ellipse**, **Round Slot**, **Square Slot**, **Cylinder**, **Cone**, **Sphere**, **Polygon**

LEAST_SQR
This field determines the calculation routine used to create the feature from the measured hits. It can toggle between LEAST_SQR, MIN_SEP, MAX_INSC, MIN_CIRCSC, and FIXED_RAD. See "Calculation list".

The Plane feature can toggle between LEAST_SQR and MIN_SEP only.

Used only on these features: **Circle**, **Cylinder**, **Plane**, **Sphere**

THEO/
This stands for "theoretical".

TX, TY, TZ, TI, TJ, TK represent the theoretical (or nominal) hit location and vector.
TLI, TLJ, TLK represent the theoretical line vector.
TEI, TEJ, TEK represent the theoretical edge vector.
TSI, TSJ, TSK represent the surface theoretical vector.
TS2I,TS2J,TS2K represent the theoretical vector for the second surface.
TLENGTH represents the feature's theoretical length.
TDIAM represents the feature's theoretical diameter. For ellipses, this is the major diameter.
TDIAM2 is an ellipse's minor diameter. TANG1 represents the feature's theoretical starting angle. TANG2 represents the feature's theoretical ending angle.

TAI,TAJ,TAK represents the theoretical angle vector.
TWIDTH and TLENGTH represent the theoretical width and length of the feature.
TAngle represents the feature's angle.

ACTL/
This stands for "actual".

X, Y, Z, I, J, K represent the actual measured hit location and vector.
SI, SJ, SK, represent the measured vector for the surface.
LI, LJ, LK represent the measured line vector.
EI, EJ, EK represent the measured edge vector.
LENGTH represents the feature's measured length.
DIAM represents the feature's measured diameter. ANG1 represents the feature's actual starting angle. ANG2 represents the feature's actual ending angle.
AI, AJ, AK represents the measured angle vector.
WIDTH and LENGTH represent the measured width and length of the feature.
Angle represents the feature's angle.

TARG/
This stands for "target".

The fields targX, targY, targZ, targI, targJ, targK allow you to control the measurement location and vector approach direction for execution while having the ability to have a completely different THEO value.

The fields targAI, targAJ, targAK let you modify the target's angle IJK vector.

MEAS/
This stands for "measured".

The fields X, Y, Z, I, J, K represent the actual measured hit location and vector.

SI, SJ, SK represent the measured vector for the surface. S2I, S2J, S2K represent the measured vector for the second surface.

NUMSIDES
This editable value must be an integer of three or higher. It defines how many sides are in the polygon. See "Num Sides list".

Used only on the **Polygon** feature.

RADIUS
This editable value defines a radius for each corner in the polygon or square slot. When taking hits, PC-DMIS moves in along the side by that amount before taking hits. This helps to avoid taking hits directly in the corner. See "Corner Rad box".

Used only on these features: **Polygon**, **Square Slot**

NUMSIDES
Used only on the **Polygon** feature.

START ANG
This field defines the feature's starting angle. See "Start and End Angles".

Used only on these features: **Ellipse**, **Cone**, **Cylinder**

START ANG1
This field defines the feature's starting angle horizontally, around the equator of a sphere. See "Start and End Angles".

Used only on the **Sphere** feature.

START ANG2
This field defines the feature's starting angle vertically, around the poles of a sphere. See "Start and End Angles".

Used only on the **Sphere** feature.

END ANG
This field defines the feature's ending angle. See "Start and End Angles".

Used only on these features: **Ellipse**, **Cone**, **Cylinder**

END ANG1
This field defines the feature's ending angle horizontally, around the equator of a sphere. See "Start and End Angles".

Used only on the **Sphere** feature

END ANG2
This field defines the feature's ending angle vertically, around the poles of a sphere. See "Start and End Angles".

Used only on the **Sphere** feature.

MEAS Angle
This field defines an angle value determining how much of the arc making up the rounded edges of the Round Slot to measure. See "Meas Angle box".

Used only on the **Round Slot** feature

INCREMENT
This field defines the increment distance away from the start point that the probe moves when following its search pattern. See "Increment box".

Used only on the **High Point** feature.

TOL
This defines the tolerance value to use during the search process. See "Tolerance box".

Used only on the **High Point** feature.

CIRCULAR or **BOX**
This field switches between CIRCULAR or BOX. It defines the search region. See "Box / Circular list".

Used only on the **High Point** feature.

OUTER RADIUS
For a CIRCULAR search region, this field defines the search region's outer radius. See "Inner/Outer list".

Used only on the **High Point** feature.

INNER RADIUS
For a CIRCULAR search region, this field defines the search region's inner radius. See "Inner/Outer list".

Used only on the **High Point** feature.

WIDTH
For a BOX search region, this field defines the width of the rectangular search region. See "Width box".

Used only on the **High Point** feature.

LENGTH
For a BOX search region, this field defines the length of the rectangular search region. See "Length box".

Used only on the **High Point** feature.

BOUNDED or **UNBOUNDED**
This field appears for Line features. It determines the type of Line feature. It switches between BOUNDED and UNBOUNDED. See "Bounded list".

Used only on the **Line** feature.

EXTERIOR or **INTERIOR**
This field switches between EXTERIOR and INTERIOR; it describes the type of angle. See "Interior / Exterior list".

Used only on these features: **Angle Point**, **Corner Point**

REPORT VEC -
This command indicates the vector used for reporting deviation. See "Extended Sheet Metal Options area".

Used only on these features: **Line**, **Surface Point**, **Vector Point**

UPDATE VEC -
This command indicates the update vector that is used to pierce the CAD surface. See "Extended Sheet Metal Options area".

Used only on the **Vector Point** feature,

Angle VEC
This defines the angle vector for the feature. See "IJK Angle boxes".

REMEASURE
This field, if set to YES, re-measures the feature against the feature's measured values. See "Measure Now and Re-Measure Toggle".

Used only on these features: **Circle**, **Notch Slot**, **Square Slot**, **Round Slot**, **Cylinder**, **Polygon**

PUNCH -
This field indicates the direction of the punch through the sheet metal. It is an editable value. See "Extended Sheet Metal Options area".

Used only on these features: **Circle**, **Square Slot**, **Round Slot**

PIN -
This field indicates the direction of the point through the hole formed by the punch. See "Extended Sheet Metal Options area".

Used only on these features: **Circle**, **Square Slot**, **Round Slot**

REPORT -
This field indicates the vector used for reporting deviation. See "Extended Sheet Metal Options area".

Used only on the **Edge Point** feature.

Surface_REPORT -
This field indicates the vector used for reporting deviation. See "Extended Sheet Metal Options area".

Used only on the **Edge Point** feature.

MEASURE ORDER
This toggle field displays the order of measuring the sample hits. The available choices are Surface, Edge, or BOTH. See "Measure Order list".

Used only on the **Edge Point** feature.

Square or **RADIAL**
For a Plane feature, this field switches between Square and RADIAL. It defines the hit pattern for hits making up the feature. See "Pattern list".

Used only on the **Plane** feature.

Surface/, **Surface2/**, **Surface3/**
Any of these command lines switches between either THEO_THICKNESS, ACTL_THICKNESS, or THICKNESS_NONE. If the latter, then no thickness is used.

Otherwise, this displays the thickness of the part for the positive or negative values can be used. See "Use Thickness".

Edge/
For a Line feature, this command determines the edge thickness of the line. See "Use Thickness".

Used only on the **Line** feature.

MEASURE MODE
This command switches between these measuring modes: FINDNOMS, Vector, NOMINAL, MASTER, and SET NOM AX. See "Nominals Mode list".

RMEAS/
This command has three fields separated by commas. If you have a single RMEAS feature (relative feature), it occupies all three fields. If you have an RMEAS feature, one for each axis, they occupy the three fields left to right: X axis RMEAS feature, Y axis RMEAS feature, and Z axis RMEAS feature. See "Setting Up Relative Measure (RMEAS)".

The relative feature or features in this command must already exist in the measurement routine.

DIRECTION
This command defines the direction in which hits are taken. It switches between CCW (counterclockwise) and CW (clockwise). See "Direction List".

Used only on these features: **Circle**, **Cylinder**

CIRCULAR MOVES/
This command has a single YES/NO toggle field. If set to yes, then PC-DMIS moves the probe in a circular motion. See "Circular Moves".

Used only on these features: **Circle**, **Ellipse**, **Round Slot**, **Square Slot**, **Notch Slot**, **Cylinder**, **Cone**, **Sphere**, **Polygon**

AUTO WRIST/
This command has a single YES/NO toggle field. If set to YES, then during feature creation PC-DMIS automatically chooses the best probe angle to use for measuring the feature and inserts the appropriate TIP/ command before the feature. See "Auto Wrist".

CLEARPlane/
This command has a single YES/NO toggle field. If set to YES, then during feature creation PC-DMIS automatically inserts a `MOVE/CLEARPlane` command (relative to the current coordinate system and part origin) before the feature. See "Clearance Plane".

GRAPHICAL ANALYSIS/
This command has a YES/NO toggle field. If set to YES, then PC-DMIS displays a graphical analysis of the feature in the Graphic Display window. Three other fields become

enabled. These three fields, from left to right, allow you to determine the Graphical Analysis Point Size, the Plus Tolerance, and the Minus Tolerance. See "Analysis area".

FEATURE LOCATOR/
Initially, this command looks like this: `FEATURE LOCATOR/NO,NO,"<text instructions>"`

The left-most YES/NO toggle field indicates whether or not the **Feature Locator** tab displays a bitmap image. If you set this to YES, an additional field bounded by quotation marks becomes enabled, allowing you to type the full pathway to the bitmap image to display:

```
FEATURE LOCATOR/YES,"<pathway to bitmap file>",NO,"<text instructions>"
```

The next YES/NO toggle field indicates whether or not the **Feature Locator** tab plays an audio file (.wav). If you set this to YES, an additional field bounded by quotation marks becomes enabled, allowing you to type the full pathway to the audio file to play:

```
FEATURE LOCATOR/YES,"<pathway to bitmap file>",YES,"pathway to audio file","<text instructions>"
```

The final field, "<text instructions>", lets you display textual instructions in the **Feature Locator** tab. See "Feature Locator Tab" topic in the PC-DMIS Vision documentation.

SHOW_CONTACT_PARAMETERS
This YES/NO toggle field determines whether or not PC-DMIS displays additional contact parameters used with the Auto feature in the Edit window. Setting this to YES displays the following fields if applicable to the Auto feature: NUMHITS, NUMROWS, PITCH, DEPTH, STARTING DEPTH, ENDING DEPTH, SAMPLE HITS, SPACER, INDENT, AVOIDANCE MOVE, FIND HOLE, ON ERROR, READ POS.

SAMPLE METHOD
This toggle field determines whether surface sampling is done by using the hits from an existing feature or by sample hits.

- If SAMPLE METHOD = SAMPLE_HITS then the SAMPLE HITS and SPACER fields appear in the command block.
- If SAMPLE METHOD = SAMPLE_FEATURE then the SAMPLE FEATURE field appears in the command block, and the SAMPLE HITS and SPACER fields are hidden.

For details, see the "Working with Contact Sample Hits Properties" topic in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Cone**, **Cylinder**, **Ellipse**, **Polygon**, **Notch**, **Round Slot**, **Square Slot**, **Surface Point**, **Line**

SAMPLE FEATURE
If the SAMPLE METHOD = SAMPLE_FEATURE, this field appears. It determines the feature to use for surface sampling.

For details, see the "Working with Contact Sample Hits Properties" topic in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Cone**, **Cylinder**, **Ellipse**, **Polygon**, **Notch**, **Round Slot**, **Square Slot**, **Surface Point**, **Line**

SAMPLE HITS
For features that support sample hits, this value defines the number of sample hits to take during feature measurement. Acceptable values depend on the type of feature.

A single sample hit adjusts only the location (offset) where PC-DMIS measures the feature. Three or more sample hits also adjust the feature's vector (orientation).

For details, see the "Working with Contact Sample Hits Properties" topic in the PC-DMIS CMM documentation.

Used only on these features: **Angle Point**, **Circle**, **Cone**, **Corner Point**, **Cylinder**, **Edge Point**, **Ellipse**, **Line**, **Notch Slot**, **Polygon**, **Round Slot**, **Sphere**, **Square Slot**, **Surface Point**

SPACER
This field defines the distance from the nominal point location (or locations) that PC-DMIS uses to measure a sample plane if sample hits are specified.

For details, see the "Working with Contact Sample Hits Properties" topic in the PC-DMIS CMM documentation.

Used only on these features: **Surface Point**, **Edge Point**, **Angle Point**, **Corner Point**, **Plane**, **Circle**, **Ellipse**, **Round Slot**, **Square Slot**, **Notch Slot**, **Cylinder**, **Cone**, **Polygon**, **Line**

INDENT1, **INDENT2**, **INDENT3**
For all but Line, this defines offset distance from the feature's point or center location to the first sample hit.

For the Line feature, INDENT2 defines the offset distance from the line's end points to the sample hits for points two and three when three sample hits are used. INDENT1 defines the offset distance for point 1 when one or three sample hits are used.

For details, see the "Working with Contact Sample Hits Properties" topic in the PC-DMIS CMM documentation.

Used only on these features: **Edge Point**, **Angle Point**, **Corner Point**, **Notch Slot**, **Line**

NUMHITS
This field determines the number of hits to take when measuring the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Line**, **Plane**, **Circle**, **Ellipse**, **Round Slot**, **Cylinder**, **Cone**, **Sphere**, **Polygon**

NUMROWS
This field determines how many rows of hits to use when measuring the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on the **Plane** feature.

NUMLEVELS
This field determines how many levels of hits to use when measuring features with multiple levels. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Cone**, **Cylinder**

DEPTH
This field determines the offset distance either below a surface or up from the bottom of a feature where PC-DMIS measures the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Edge Point**, **Line**, **Circle**, **Ellipse**, **Round Slot**, **Square Slot**, **Notch Slot**, **Polygon**

STARTING DEPTH
This field defines the starting depth of the first level of hits for features with multiple levels. This depth is offset from the top of the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Cone**, **Cylinder**

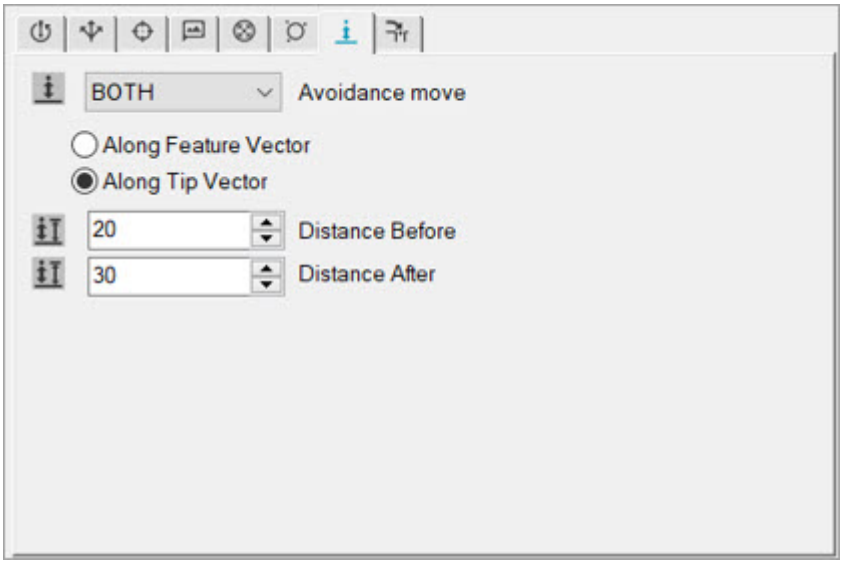
ENDING DEPTH
This field defines the ending depth of the last level of hits for features with multiple levels. This depth is offset from the top of the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Cone**, **Cylinder**

PITCH
This field determines the distance between threads along the axis of the feature. For more information, see "Working with Contact Path Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Cylinder**

AVOIDANCE MOVE/
You define the `AVOIDANCE MOVE` command from the **Contact Auto Move Properties** tab of the **Probe Toolbox**. From this tab you define the type of avoidance move from the **Avoidance move** list, the approach you want the tip to take and the amount to move before and after you create the Auto feature.



Probe Toolbox - Contact Auto Move Properties Tab

This tab becomes visible when you have the **Auto Feature** dialog box open and you enable a contact probe.

The **Avoidance move** list contains these options:

NO - PC-DMIS does not perform any avoidance move. This sets the `PTP_AvoidMove` setting entry to 0 (zero).

BOTH - PC-DMIS performs both the **Distance Before** and **Distance After** moves. This sets the `PTP_AvoidMove` setting entry to 1.

- The probe moves to a distance defined by the value set by the `PTP_AutoMoveDistance` setting entry above the centroid *before* it takes the first hit of the feature being created.
- The probe moves to the distance defined by the value set by the `PTP_AutoMoveDistance2` setting entry *after* it takes the last hit of the feature being created.

BEFORE - PC-DMIS only performs the **Distance Before** move where the probe moves to a distance defined by the `PTP_AutoMoveDistance` entry above the centroid *before* it takes the first hit of the feature being created. This sets the `PTP_AvoidMove` setting entry to 2.

AFTER - PC-DMIS only performs the **Distance After** move where the probe moves to the distance defined by the `PTP_AutoMoveDistance2` entry *after* it takes the last hit of the feature being created. This sets the `PTP_AvoidMove` setting entry to 3.

You must verify any measurement routine that uses the **Distance Before** and **Distance After** settings for collision detection if you intend on running it on an earlier version of PC-DMIS that only supports the **Distance Before** option. PC-DMIS uses the smaller of the two values for versions that only support the **Distance Before** option, which may result in a hardware crash.

For details on the Auto Move settings in the Settings Editor application, see "`PTP_AutoMove`" in the PC-DMIS Settings Editor documentation.

Select the tip's approach:

Along Feature Vector - PC-DMIS applies the avoidance move along the feature's vector.

Along Tip Vector - PC-DMIS applies the avoidance move along the tip's vector.

Depending on the option you select from the **Avoidance move** list, you can type the **Distance Before** and/or **Distance After** values. These let you define the avoidance move's distance before and after you create the Auto feature. For more information, see "Working with Contact Auto Move Properties" in the PC-DMIS CMM documentation.

USE BOUNDARY OFFSET
This YES/NO toggle field appears if VOID DETECTION=YES and if the current Auto feature is a plane. It determines whether or not a user-defined boundary offset distance is used for void detection. If this is set to YES, then the OFFSET field appears defining the minimum distance. If it is set to NO, then the OFFSET field is hidden, and the software uses a default distance of the radius of the current tip. For more information, see "Working with Contact Auto Move Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Cylinder**, **Round Slot**, **Square Slot**, **Notch Slot**, **Polygon**

FIND HOLE
This toggle field determines the method PC-DMIS uses to find hole features. Available options include: DISABLED, CENTER, SINGLE HIT, or NOCENTER. For more information, see "Working with Contact Find Hole Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Cylinder**, **Polygon**

ONERROR
This YES/NO field determines whether or not PC-DMIS uses improved error checking when it detects an unexpected or missed hit.

If set to YES and a machine error (such as an unexpected hit) occurs, then PC-DMIS displays the **Read Position** dialog box. You can then use your jog box to move the machine to the feature's location and attempt to re-measure it.

If set to NO, the usual "Movement Interrupted" message appears instead.

For more information, see "Working with Contact Find Hole Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Angle Point**, **Circle**, **Edge Point**, **Corner Point**, **Ellipse**, **Round Slot**, **Square Slot**, **Notch Slot**, **Cylinder**, **Cone**, **Polygon**

READ POS
This YES/NO field determines whether or not PC-DMIS pauses execution above the surface feature displaying a message, asking if you want to use the current data. For more information, see "Working with Contact Find Hole Properties" in the PC-DMIS CMM documentation.

Used only on these features: **Circle**, **Ellipse**, **Round Slot**, **Square Slot**, **Notch Slot**, **Cylinder**, **Cone**, **Polygon**

SHOWHITS
This YES/NO toggle field determines whether or not PC-DMIS displays all the hits making up the feature. If set to YES, then PC-DMIS displays a HIT/BASIC command line for each hit.

If you want to see the hits in the Graphic Display window, see "Show Hit Targets".

HIT/BASIC
This command line displays the theoretical XYZ, the theoretical IJK, and the measured XYZ values for the hit.

Setting Up Relative Measure (RMEAS)

To set the relative measurement mode for Auto features, select **Insert | Parameter Change | Auto Relative Measure**. When you select this menu option, PC-DMIS inserts an `RMEAS` command into the Edit window. By default, PC-DMIS sets the mode to use the Default (I,J,K, T) mode. This mode (previously called Absolute mode) uses the RMEAS feature’s measured position and vector. It applies any positional offsets along that vector. For more information about this mode, see the "Default (I,J,K, T) RMEAS Mode" topic.

You can change the mode to the Legacy (I,J,K, X,Y,Z) mode if desired. This mode (previously called Normal mode) uses the deviation of the RMEAS feature’s position and orientation. To change the mode, select **Insert | Parameter Change | Auto Relative Measure**. For more information about this mode, see the "Legacy (I,J,K, X,Y,Z) RMEAS Mode" topic.

The **Insert | Parameter Change | Auto Relative Measure** menu option toggles between Default (I,J,K, T) mode and Legacy (I,J,K, X,Y,Z) mode.

Before you use RMEAS, ensure that the `RMEAS_modeDefaultForPlane` entry is correctly set. For details, see the "RMEAS_modeDefaultForPlane" topic in the "USER_AutoFeatures" section of the PC-DMIS Settings Editor documentation.

Also, if you are using a dual-arm machine, remember to also check the same setting in the USER_AutoFeatures_CMM2 section.

For information on how to use PC-DMIS Settings Editor, see the "Modifying Setting Entries" appendix.

You should insert the RMEAS command into the measurement routine before you use the **Relative to** option in the **Advanced measurement options** area in the **Auto Feature** dialog box. If no RMEAS command appears in the measurement routine, PC-DMIS uses the default RMEAS algorithm to compute the RMEAS. For more information, see the "Relative To" topic.

Default (I,J,K, T) RMEAS Mode

When an Auto feature has a relative planar measure feature associated with it (see "Relative To"), PC-DMIS measures the Auto feature in a location that is adjusted according to the following rules:

- The measuring orientation of the Auto feature is corrected by the rotation offset that exists between the orientation of the nominal and the actual RMEAS feature.
- The measuring location of the Auto feature is corrected by the positional offset that exists between the location of the nominal and the actual RMEAS feature.

The Edit window command for this option is:

```
RMEAS/DEFAULT (I,J,K, T)
```

Math Process of the Default (I,J,K, T) RMEAS Mode

The Default (I,J,K T) RMEAS mode controls I, J, K, T, and therefore works well with RMEAS features such as planes.

Using the numerical values taken from the example features in the table below, follow these steps to understand how RMEAS/DEFAULT (I,J,K, T) functions when the RMEAS feature is "plane reducible".

A "reducible" feature is a feature that also contains information to be used as another feature. For example, a circle feature is point reducible because a point feature can be automatically extracted from the circle's centroid. It is also line reducible because a line can be drawn along the vector and through the centroid. It is plane reducible because a plane can be drawn intersecting all of the circle's hits.

1. Create a coordinate system (roto-translation matrix) given the nominal RMEAS feature XYZ IJK and the intersection vector between the nominal and the actual RMEAS feature.
2. Move the nominal Auto feature XYZ and IJK into the RMEAS coordinate system.
3. Zero-out the T value and rotate the nominal Auto feature XYZ on the plane of the actual RMEAS feature.
4. Offset the transformed Auto feature XYZ back to the original T offset plus the distance between the actual and the nominal RMEAS feature.
5. Move the transformed Auto feature XYZ and IJK back into the PART coordinate system.
6. Use the new nominal XYZ and IJK to measure the Auto feature.

Example Feature	XYZ	IJK
Nominal RMEAS Feature	0, 0, 2	0, 0, 1
Actual RMEAS Feature	-1, 0, 1	-0.7071, 0, 0.7071
Nominal Auto Feature	2, 1, 0	0.7071, 0, 0.7071
New Nominal Auto Feature	1.4142, 1, 0.4142	0, 0, 1

Example with Translation only	XYZ	IJK
Nominal RMEAS Feature	124, 50, 0	0, 0, 1
Actual RMEAS Feature	123, 50, -1	0, 0, 1
Nominal Auto Feature	93.5, 19.5, 0	0, 0, 1
New Nominal Auto Feature	93.5, 19.5, -1	0, 0, 1

Legacy (I,J,K, X,Y,Z) RMEAS Mode

The Legacy (I,J,K, X,Y,Z) RMEAS mode takes the position and the orientation of the reference feature into account.

To understand this mode, examine an Auto Circle with three sample hits. PC-DMIS first takes three hits on the surface around the circle and then measures the circle based on the plane's location and the surface normal vector's orientation. Therefore, if the plane is at a 45-degree angle, PC-DMIS also measures the Auto Circle at a 45-degree angle.

Similarly, if the relative measure feature is rotated from its original orientation, then the associated feature is also measured with the same rotational offset.

Before you use Legacy RMEAS mode, ensure that the `RMEAS_modeDefaultForPlane` entry is correctly set. For details, see the "RMEAS_modeDefaultForPlane" topic in the "USER_AutoFeatures" section of the PC-DMIS Settings Editor documentation.

For information on how to use PC-DMIS Settings Editor, see the "Modifying Setting Entries" appendix.

This process makes Legacy mode slightly smarter than Default mode, because when it looks at the reference feature, it only offsets the main feature by valid axes for the reference feature type. For example, it makes more sense to shift in a plane's normal vector instead of in all directions, which Legacy mode now does.

The Edit window command line for this option is:

```
RMEAS/LEGACY (I,J,K, X,Y,Z)
```

Math Process of the Legacy (I,J,K, X,Y,Z) RMEAS Mode

The Legacy (I,J,K, X,Y,Z) RMEAS mode controls I, J, K, X,Y, Z (T) and therefore works well with 3D RMEAS features, such as circles with sample hits.

Use the numerical values from the example features in the table below, and follow these steps to understand how RMEAS/LEGACY (I,J,K, X,Y,Z) functions.

1. Create a coordinate system (roto-translation matrix) given the nominal RMEAS feature XYZ IJK.
2. Move the nominal Auto feature XYZ and IJK into the RMEAS coordinate system.
3. Create a new coordinate system given the actual RMEAS feature XYZ IJK.
4. Now, move the roto-translated Auto feature XYZ and IJK back into the PART coordinate system using the new RMEAS coordinate system.
5. Use the new nominal XYZ and IJK to measure the Auto feature.

Example Feature	XYZ	IJK
Nominal RMEAS Feature	0, 0, 2	0, 0, 1
Actual RMEAS Feature	-1, 0, 1	-0.7071, 0, 0.7071
Nominal Auto Feature	2, 1, 0	0.7071, 0, 0.7071
New Nominal Auto Feature	1.8284, 1, 1	0, 0, 1

Example with Translation Only	XYZ	IJK
Nominal RMEAS Feature	124, 50, 0	0, 0, 1
Actual RMEAS Feature	123, 50, -1	0, 0, 1
Nominal Auto Feature	93.5, 19.5, 0	0, 0, 1
New Nominal Auto Feature	92.5, 19.5, -1	0, 0, 1

Creating Measured Features

Creating Measured Features: Introduction

Whenever you take hits on a part, PC-DMIS interprets those hits into different features depending on the number of the hits, their vectors, and so forth. PC-DMIS creates measured features from these hits. PC-DMIS supports the following measured features:

- Measured Point
- Measured Line
- Measured Plane
- Measured Circle
- Measured Sphere
- Measured Torus
- Measured Cone
- Measured Cylinder
- Measured Round Slot
- Measured Square Slot

These features are described in detail in the "Understanding the Command Format" topic.

The following major topics are covered in this chapter.

- Understanding the Command Format
- Inserting a Measured Feature
- Editing a Measured Feature
- Overriding a Guessed Measured Feature
- Creating Measured Feature Sets

You can also use insert measured features into your measurement routine by using the **Quick Start** toolbar. For information, see the "Quick Start: Measure Toolbar" topic in the "Using Toolbars" chapter.

Understanding the Command Format

All measured features are displayed in the following format in the Command Mode of the Edit window. There will be slight variations that are explained more thoroughly in the following sections. The actual Edit report will be displayed using all CAPS.

For example:

```
feature_name=FEAT/FEATURE_TYPE,TOG1,TOG4
THEO/TX,TY,TZ,TI,TJ,TK,TD,TMD,TL
ACTL/X,Y,Z,I,J,K,D,MD,L
MEAS/TOG2,TOG3,
   HIT/TOG5,TX,TY,YZ,TI,TJ,TK,X,Y,Z,USE_THEO=TOG6
ENDMEAS/
```

feature_name - This displays the name of the feature. This is a user editable field.

FEATURE_TYPE - This denotes the type of feature and is the same as TOG2 but is non-editable and cannot be toggled.

..... - This indicates that what follows is unique to each feature.

TOG1 - This field is present in all features and can be toggled between RECT / POLR. If you set this to RECT, PC-DMIS displays all the points in the Cartesian system (x, y, z). If you set this to POLR, PC-DMIS displays all the points in the Polar system (x_radius, y_angle, z_height). Vectors do not change.

***TOG2* -** This field is present in all measured features and can be switched between:

> **CIRCLE / CONE / CURVE / CYLINDER / LINE / PLANE / POINT / SET / SLOT / SPHERE / TORUS**
>
> POINT is the default feature type when the measured command line is opened for the first time in a measurement routine. The default feature type then becomes the last measured feature type.

TOG3 - This field represents the number of hits to take, according to the type of feature displayed in *TOG2*. The HIT line is varied, based on the number of hits you define. For example, 1 hit = 1 HIT/BASIC line, 3 hits = 3 HIT/BASIC lines. The required minimum number is indicated as the default.

***TOG4* -** This field also takes unique values according to the type of feature represented in TOG2. Linear features allow you to toggle between BOUND/ UNBOUND. Circular features allow you to switch between IN/OUT. Angular features allow you to switch between LENGTH/ANGLE.

TOG5 - This field represents the type of hit that is to be taken:

BASIC

TOG6 - This field allows you to determine whether or not to use the theoretical vector as the approach vector for the hit. This defaults to YES. If set to NO, PC-DMIS calculates the approach vector by subtracting the theoretical hit point (X,Y,Z) from the current probe position.

TX,TY,TZ,TI,TJ,TK - These numbers (TX,TY,TZ) represent the theoretical target vector and the theoretical approach vector (TI,TJ,TK) and you can edit them.

TD - For circular features, this number is the theoretical diameter.

TMD - For a torus feature, this number is the theoretical minor diameter.

X,Y,Z,I,J,K - These numbers are non-editable and represent the measured point and measured approach vector.

D - For circular features, this number is the measured diameter.

MD - For a torus feature, this number is the minor diameter.

TL - This is the theoretical length of features that have a length.

L - This is the measured length of features that have a length.

Basic Measurement Format for a Point

A single hit is required for a point.

```
Feature_name=FEAT/POINT,TOG1
THEO/TX,TY,TZ,TI,TJ,TK
ACTL/X,Y,Z,I,J,K
MEAS/POINT,TOG3
   HIT/…
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This list the number of hits that PC-DMIS should take to measure this feature. Since a point can only have one point, you cannot edit this value for this feature.

... - A single hit is entered at the ellipsis.

Basic Measurement Format for a Line

The minimum number of hits for a line is two.

```
Feature_name=FEAT/LINE,TOG1,TOG4
THEO/TX,TY,TZ,TI,TJ,TK
ACTL/X,Y,Z,I,J,K
MEAS/LINE,TOG3,TOG7
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user-editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 2 hits = 2 HIT/BASIC lines, 3 hits = 3 HIT/BASIC lines.

TOG4 - This field can be toggled between BND / UNBND. If set to BND, the THEO and ACTL fields, no longer display the vector information, but instead show the XYZ values for the second point along with the first.. See "Line Definition Format" later in this chapter.

TOG7 - This field switches between the various reference types. These include FEATURE, 3D, WORKPLANE, XPLUS, YPLUS, ZPLUS, XMINUS, YMINUS, ZMINUS.

... - As many hits as are necessary are entered at the ellipsis.

Measured Lines and Workplanes

When creating a measured line, PC-DMIS expects the hits for the line to be taken at a vector perpendicular to the current workplane.

For example, if your current workplane is ZPLUS (with a vector 0,0,1), and you have a block-like part, the hits for the measured line must be on a vertical face of that part, such as the front or side.

If you then wanted to measure a line feature on the top surface of the part, you would need to switch the workplane to XPLUS, XMINUS, YPLUS, or YMINUS, depending on the direction of the line.

Basic Measurement Format for a Plane

The minimum number of hits for a plane is three.

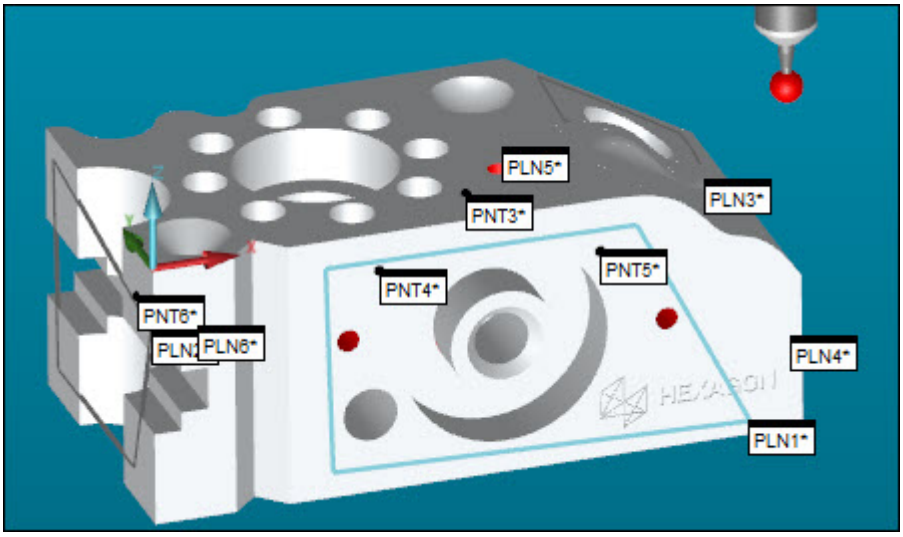
```
Feature_name=FEAT/PLANE,TOG1
THEO/TX,TY,TZ,TI,TJ,TK
ACTL/X,Y,Z,I,J,K
MEAS/PLANE,TOG3
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 3 hits = 3 HIT/BASIC lines, 8 hits = 8 HIT/BASIC lines.

... - As many hits as are necessary are entered at the ellipsis.

When you create a plane feature, PC-DMIS shows the outline of the plane from the plane's hits in the Graphic Display window:



Sample plane from four hits taken on the Y- face

Basic Measurement Format for a Circle

The minimum number of hits for a circle is three.

```
Feature_name=FEAT/CIRCLE,TOG1,TOG4,TOG6
THEO/TX,TY,TZ,TI,TJ,TK,TD
ACTL/X,Y,Z,I,J,K,D
MEAS/CIRCLE,TOG3,TOG7
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 3 hits = 3 HIT/BASIC lines, 5 hits = 5 HIT/BASIC lines.

TOG4 - This field can be toggled between IN / OUT.

TOG6 - This field switches between the various Best Fit types available. These include: LEAST_SQR, MIN_SEP, MAX_INSC, MIN_CIRCSC, FIXED_RAD

TOG7 - This field switches between the various reference types. These include FEATURE, 3D, WORKPLANE, XPLUS, YPLUS, ZPLUS, XMINUS, YMINUS, ZMINUS.

... - As many hits as are necessary are entered at the ellipsis.

Basic Measurement Format for a Sphere

The minimum number of hits for a sphere is four.

```
Feature_name=FEAT/SPHERE,TOG1,TOG4
THEO/TX,TY,TZ,TI,TJ,TK,TD
ACTL/X,Y,Z,I,J,K,D
MEAS/SPHERE,TOG3
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 4 hits = 4 HIT/BASIC lines, 7 hits = 7 HIT/BASIC lines.

TOG4 - This field can be toggled between IN / OUT.

... - As many hits as are necessary are entered at the ellipsis.

Basic Measurement Format for a Torus

The minimum number of hits for a torus is seven.

```
Feature_name=FEAT/TORUS,TOG1,TOG4
THEO/TX,TY,TZ,TI,TJ,TK,TD,TMD
ACTL/X,Y,Z,I,J,K,D,MD
MEAS/TORUS,TOG3
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 7 hits = 7 HIT/BASIC lines, 9 hits = 9 HIT/BASIC lines.

TOG4 - This field can be toggled between IN / OUT.

... - As many hits as are necessary are entered at the ellipsis.

Basic Measurement Format for a Cone

The minimum number of hits for a cone is six.

```
Feature_name=FEAT/CONE,TOG1,TOG4,TOG5
THEO/TX,TY,TZ,TI,TJ,TK,TL,TD1,TD2
ACTL/X,Y,Z,I,J,K,L,D1,D2
MEAS/CONE,TOG3
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 6 hits = 6 HIT/BASIC lines, 11 hits = 11 HIT/BASIC lines.

TOG4 - This field can be toggled between IN / OUT.

TOG5 - This field can be toggled between LENGTH / ANGLE.

... - As many hits as are necessary are entered at the ellipsis.

TD1,TD2 - These are the cone's two theoretical diameters.

D1,D2 - These are the actual measured values for the cone's two diameters.

Basic Measurement Format for a Cylinder

The minimum number of hits for a cylinder is six.

```
Feature_name=FEAT/CYLINDER,TOG1,TOG4,TOG6
THEO/TX,TY,TZ,TI,TJ,TK,TD,TL
ACTL/X,Y,Z,I,J,K,D,L
MEAS/CYLINDER,TOG3
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 6 hits = 6 HIT/BASIC lines, 11 hits = 11 HIT/BASIC lines.

TOG4 - This field can be toggled between IN / OUT.

TOG6 - This field switches between the various Best Fit types available. These include: LEAST_SQR, MIN_SEP, MAX_INSC, MIN_CIRCSC, FIXED_RAD

... - As many hits as are necessary are entered at the ellipsis.

If you create a measured cylinder with more than two rows, and later change the **Number of Hits** in the **Measured Cylinder** dialog box (**Insert | Feature | Measured Feature**), PC-DMIS will keep the number of rows the same.

Basic Measurement Format for a Round Slot

The minimum number of hits for a round slot is six.

```
Feature_name=FEAT/SLOT,TOG1,TOG2
THEO/TX,TY,TZ,TI,TJ,TK,SI,SJ,SK,TW,TL
ACTL/X,Y,Z,I,J,K,MI,MJ,MK,W,L
MEAS/SLOT,TOG3,TOG4
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG2 - This field can be toggled between IN / OUT.

TX,TY,TZ - Theoretical XYZ

TI,TJ,TK - Theoretical IJK vector

SI,SJ,SK - Theoretical IJK slot vector

TW - Theoretical width

TL - Theoretical length

X,Y,Z - Actual measured XYZ

I,J,K - Actual measured IJK

MI, MJ,MK - Measured IJK slot vector

W - Actual measured width

L - Actual measured length

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 6 hits = 6 HIT/BASIC lines, 8 hits = 8 HIT/BASIC lines.

TOG4 - *This value lets you choose the reference type for the slot.*

... - As many hits as are necessary are entered at the ellipsis.

Basic Measurement Format for a Square Slot

The minimum number of hits for a square slot is five.

```
Feature_name=FEAT/SLOT,TOG1,TOG2
THEO/TX,TY,TZ,TI,TJ,TK,SI,SJ,SK,TW,TL
ACTL/X,Y,Z,I,J,K,MI,MJ,MK,W,L
MEAS/SLOT,TOG3,TOG4
   ...
ENDMEAS/
```

TOG1 - This field can be toggled between RECT / POLR.

TOG2 - This field can be toggled between IN / OUT.

TX,TY,TZ - Theoretical XYZ

TI,TJ,TK - Theoretical IJK vector

SI,SJ,SK - Theoretical IJK slot vector

TW - Theoretical width

TL - Theoretical length

X,Y,Z - Actual measured XYZ

I,J,K - Actual measured IJK

MI, MJ,MK - Measured IJK slot vector

W - Actual measured width

L - Actual measured length

TOG3 - This value is user editable and is determined by the feature being measured. The HIT line is varied, based on the number of hits set up by the user. For example, 5 hits = 5 HIT/BASIC lines.

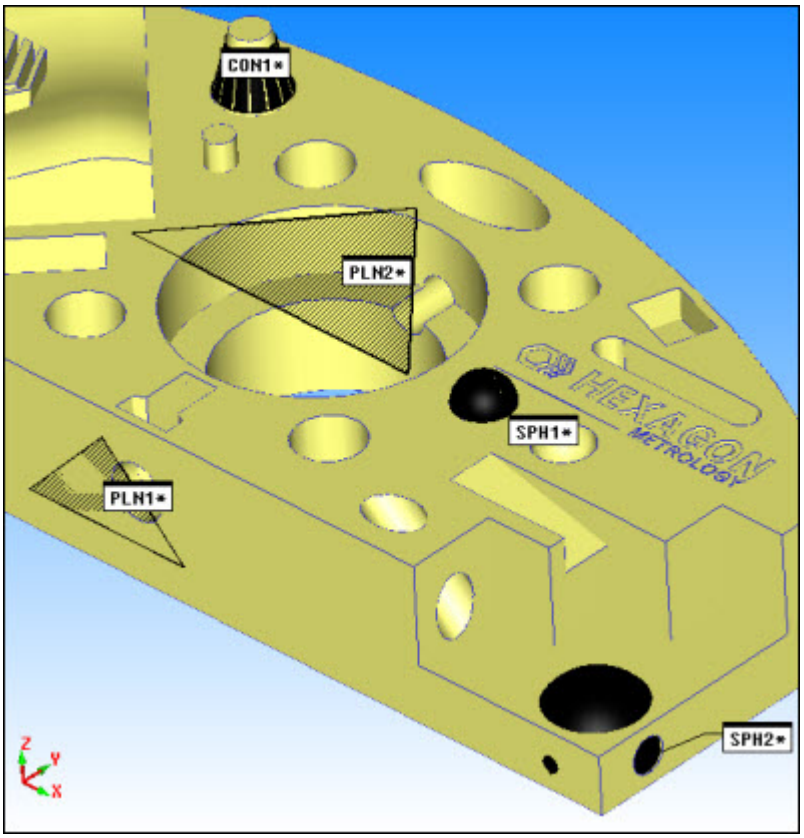
TOG4 - This value lets you choose the reference type for the slot.

... - As many hits as are necessary are entered at the ellipsis.

Inserting a Measured Feature

PC-DMIS allows you to insert measured features into your measurement routine by interpreting your probe hits and guessing the correct feature type. If PC-DMIS guesses incorrectly, you can force PC-DMIS to choose the correct feature type as long as you have taken the minimum number of hits for the desired feature.

As hits are taken and once the feature is created, PC-DMIS draws the measured feature on the screen. For 3D measured features (Torus, Cylinder, Sphere, Cone) and for the 2D Plane feature, PC-DMIS will draw the feature with a shaded surface.



Example showing the measured features displayed with shaded surfaces.

Hiding Shaded Plane Features

You can hide shaded planes by setting the **None** option in the **Display** area of the **Measured Plane** dialog box (**Insert | Feature | Measured Feature**). You can also globally hide all drawn shaded planes for future plane features by marking the **Do Not Display Plane** check box in the **Setup Options** dialog box.

Changing Feature Color

If desired, you can modify the feature color used during feature creation by using the **ID Setup** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). See the **Color** check box that appears after choosing **Features** under the **Labels for** item.

For information on creating the different feature types, consult the documentation set for your PC-DMIS configuration:

- PC-DMIS CMM
- PC-DMIS Portable

The procedures given in the following topic assume that you are working in Offline mode.

To Force the Insertion of a Certain Measured Feature Type

If PC-DMIS guesses the wrong measured feature type, you can cause it to insert the desired type by selecting the correct measured feature from the **Measured Features** toolbar or submenu *prior to pressing the END key*. Once you press the END key, PC-DMIS will insert the selected feature type into the Edit window.

Guessing a Measured Feature Type

If the **Measured Features** toolbar is set to **Measured Guess** ( ), then PC-DMIS is in Guess Mode. The following table shows the minimum number of hits PC-DMIS needs to guess a particular measured feature and that feature's dimension type.

Minimum # of Hits	Feature	Dimension
1	Point	1D
2	Line	2D
3	Plane	2D
3	Circle	2D
6	Cylinder	3D
6	Cone	3D
4	Sphere	3D
7	Torus	3D
6	Round Slot	2D
5	Square Slot	2D

Measured Point - Rules:

- Probe compensation is in the nearest axis (X, Y, or Z). This is determined based on the direction of machine motion at the time the hit is taken. Always approach normal to the surface.
- Only measure points on surfaces perpendicular to the X, Y, or Z axis to ensure accurate probe compensation. Use Vector or Surface points for other surfaces (Auto feature).
- Measured points are 1D; therefore, dimensional information is only valid in one axis (X, Y, or Z).

Measured Line - Rules:

- Measured lines are 2D; therefore, they are dependent on the active workplane for calculation purposes. You must activate the appropriate workplane before you measure a line.
- You control the direction of the measured line. The direction will go from the first hit towards the second hit.
- The dimensional information in the X, Y, or Z axis is from the centroid of the line.

Measured Circle - Rules:

- Measured circles are 2D; therefore, they are dependent on the active workplane for calculation purposes. You must activate the appropriate workplane before you measure a circle.
- A minimum arc of 90 degrees is required to accurately calculate a measured circle.
- The dimensional information in the X, Y, or Z axis is from the centroid of the circle, and the diameter is the average diameter by default (least squares).

Measured Plane - Rules:

- Measured planes are 3D; therefore, they are not dependent on the active workplane for calculation purposes.
- The IJK vector for the plane is perpendicular to the plane, and its direction points away from the material of the part.
- The dimensional information in the X, Y, or Z axis is from the centroid of the plane.

Measured Cylinder - Rules:

- Measured cylinders are 3D; therefore, they are not dependent on the active workplane for calculation purposes.
- You control the direction of the IJK vector for the cylinder. It goes from the first circular cross-section toward the second circular cross-section. When possible, take three circular cross-sections to increase the hit data used to calculate the cylinder.

- The dimensional information in the X, Y, or Z axis is from the centroid of the cylinder, and the diameter is the average by default (least squares).

Measured Cone - Rules:

- Measured cones are 3D; therefore, they are not dependent on the active workplane for calculation purposes.
- You do not control the direction of the IJK vector for the cone. The IJK vector always points away from the vertex of the cone. When possible, take three circular cross-sections to increase the hit data used to calculate the cone.
- The dimensional information in the X, Y, or Z axis is from the centroid of the cone.

Measured Sphere - Rules:

- Measured spheres are 3D; therefore, they are not dependent on the active workplane for calculation purposes.
- In some cases, it is not possible to take a hit on top of the sphere. In these cases, take three circular cross-sections. The IJK vector direction will go from the first toward the last cross-section.
- The dimensional information in the X, Y, or Z axis is from the centroid of the sphere, and the diameter is the average.

Measured Torus - Rules:

- Measured tori are 3D; therefore, they are not dependent on the active workplane for calculation purposes.
- The first three hits must be taken in a plane around the torus that is perpendicular to the center circle.
- The dimensional information in the X, Y, or Z axis is from the centroid of the torus.

Measured Round Slot - Rules:

- Six hits are required.
    - Option 1: Two points on each straight side and one point on each curve.
    - Option 2: Three points on each curve.

Measured Square Slot - Rules:

- Five hits are required: Two points on one of the long sides and one hit on each of the three remaining sides.
- The hits must be taken in a strictly clockwise or counterclockwise direction.

Using Guess Mode

1. Access the Edit window, and place it in either Command Mode or Summary Mode.
2. Place PC-DMIS into Program Mode, and set the appropriate probe depth.

3. Access the **Measured Features** toolbar and then click the **Guess Mode** icon .
4. Click on the model of the part at the location where you want to insert the feature.
    - *If you're using an imported wireframe CAD model*, PC-DMIS automatically uses the nearest wire as the basis for the feature you selected.
    - *If you're using an imported solid CAD model*, PC-DMIS counts each mouse click as an individual hit.
5. Press the End key. PC-DMIS interprets your hits and places the guessed feature into the Edit window.

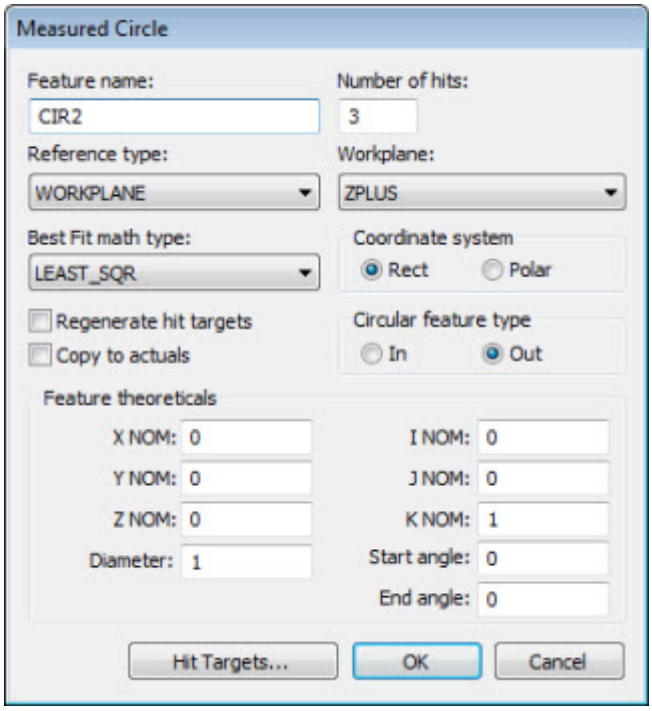
For information about the **Guess Mode** icon and the **Measured Features** toolbar, see "Measured Features Toolbar" in the "Using Toolbars" chapter.

Editing a Measured Feature

PC-DMIS will allow you to select the desired measured feature in the Edit window and make any corrections directly to the Edit window. The **Measured Feature** dialog box was created to offer a user friendly alternative to editing the Edit window.

To open a **Measured Feature** dialog box, follow these steps:

1. Place the cursor within the first four lines of the desired measured feature.
2. Press the **F9** key.



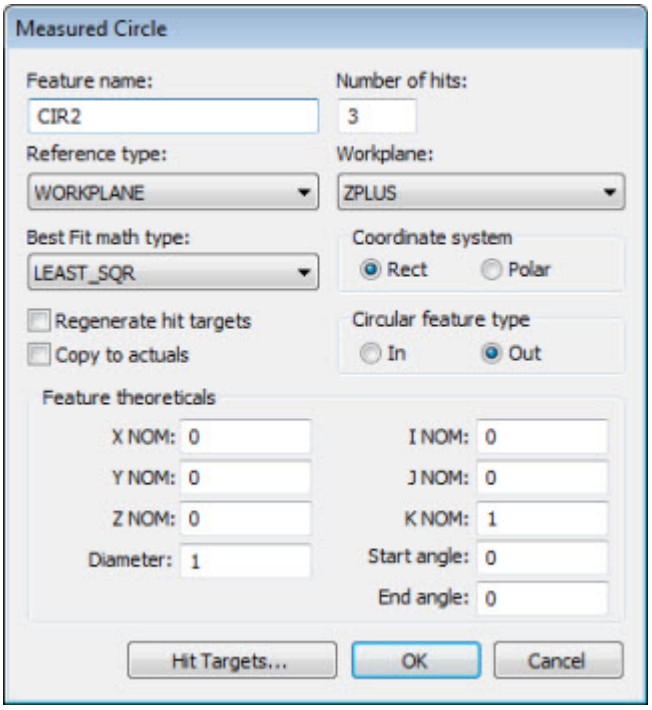
Measured Circle Dialog Box

Generally a **Measured Feature** dialog box opens based on the feature type. However, if you place the cursor on a listed hit (such as HIT/BASIC) within the measured feature and you press the **F9** key, PC-DMIS instead displays the **Measured Hits** dialog box for the selected hit.

Measured Feature Dialog Box Description

Measured features can be created or edited using the **Measured Feature** dialog box.

Select **Insert | Feature | Measured Feature** and then selecting an item from that menu to create your feature. Once created, place the cursor within the first four lines of the measured feature and press the F9 key on your keyboard to display the **Measured Feature** dialog box.



Measured Circle Dialog Box

The following topics describe the different elements of the **Measured Feature** dialog box. Some elements of the dialog box may only appear for certain feature types.

Feature Name

The **Feature Name** box allows you to change the name of the displayed feature.

You can also change the name in the Edit window. To do that, highlight the feature ID in the Edit window, type in a new identification and press the Enter key.

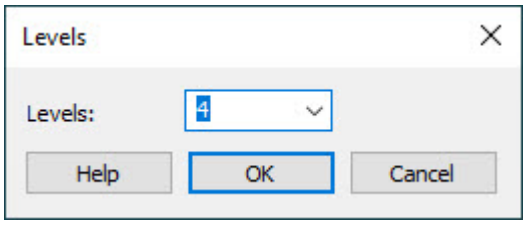
Number of Hits

The **Number of hits** box allows you to change the number of hits for a specified feature.

Suppose there are four hits in the edit box. If you delete a hit, the software removes the last hit (#4) in the feature list. If you delete two hits, the software removes the last two hits (#4 and #3). PC-DMIS renumbers the remaining hits based on its current numbering assignment.

If you add a hit, PC-DMIS increases the **Hit Targets** list by one, displaying an extra hit at the end of the hit list. This hit does not indicate any x, y, z, i, j, k values until the measurement routine is executed. At that time, PC-DMIS prompts you to take the additional hit. PC-DMIS asks if the hits are to be equally spaced.

- If you click **Yes**, PC-DMIS takes the hits in equally spaced intervals. For a cone, sphere, cylinder, or torus feature, PC-DMIS displays a **Levels** dialog box.



Levels Dialog Box

You can use this dialog box to specify the number of levels PC-DMIS uses to equally space the hits. The default is two.

- If you click **No**, PC-DMIS displays the hit values as 0, 0, 0, 0, 0, 1. You then need to manually enter the hit values.

Reference Type

Some measured features (circles, ellipses, lines, polygons, and slots) can be projected relative to a reference plane.

The **Reference Type** list controls whether or not the measured feature is 3D (not projected), projected relative to the current workplane, or projected relative to another specified plane. This affects mathematically how the final characteristics of the feature are derived.

For a detailed explanation on reference types, see the description of the **Reference Feature** area in the "Reference Feature" portion of "Using the Quick Start Dialog Box" topic in the "Using Other Windows, Editors, and Tools" chapter.

Best Fit Math Type

This **Best Fit Math Type** list indicates the type of math algorithm that will be used to measure the feature. LEAST_SQR is the standard Best Fit method.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

See "Calculation List" in the "Creating Auto Features" chapter for a discussion on these types.

Regenerate Hit Targets

The **Regenerate Hit Targets** check box regenerates this feature's `HIT/BASIC` commands so that they are updated with changed data from the **Feature** Theoreticals area.

Copy to Actuals

The **Copy to Actuals** check box copies any changes made to the **Feature Theoreticals** area of the **Measured Feature** dialog box (**Insert | Feature | Measured Feature**) and applies the same changes to the actual measured data.

Coordinate System

The **Coordinate System** area allows you to choose between rectangle and polar coordinates. If the Rect option is selected, all points are represented in Cartesian system (x, y, z). If Polar is selected, all points are displayed in polar system (x_radius, y_angle, z_height). Vectors do not change.

Angular Definition Format

If an angular feature is displayed, you can choose between LENGTH and ANGLE.

- *The length option* displays the diameters of two circles. It also displays the length between the two circles.
- *The angle option* will display the nominal (x, y, z) and vector (i, j, k) of the point. It also displays the angle value.

Circular Feature Type

If a circular feature is displayed, PC-DMIS allows you to choose between IN and OUT.

Line Definition Format

If a linear feature is displayed, PC-DMIS will allow you to choose between the Bound and Unbound options in the **Linear Definition Format** area.

Selecting **Bound** causes PC-DMIS to display the two end points that make up the line in the **Feature Theoreticals** area, like this:

X, Y, Z

and

X2 Nom, Y2 Nom, and **Z2**

For Bounded lines, the **Length** value is grayed out and unavailable for modification.

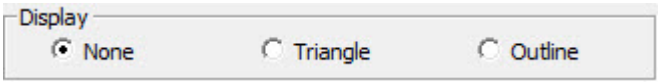
Selecting **Unbound** causes PC-DMIS to display the line's nominal information in the Feature Theoreticals area, like this:

X, Y, Z, and **length**

and

I Nom, J Nom, and **K**

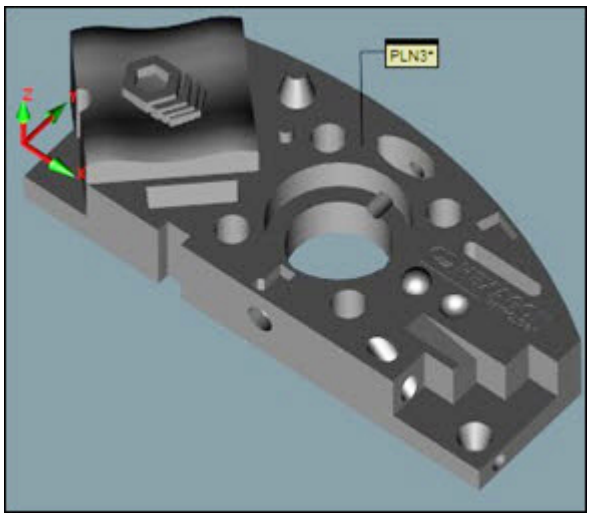
Using the Display Area



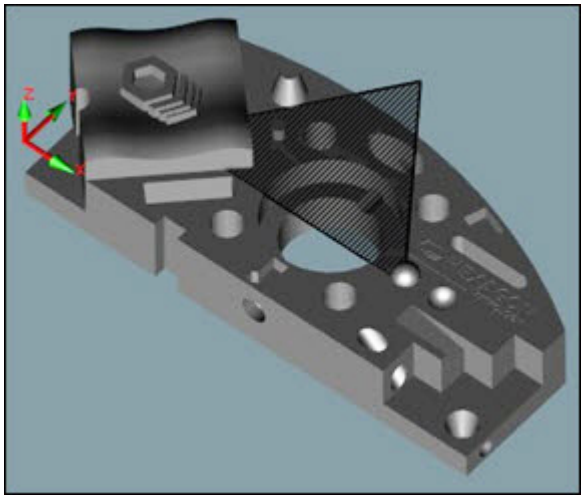
Display Area

This area defines how the plane feature draws in the Graphic Display window. It contains these options:

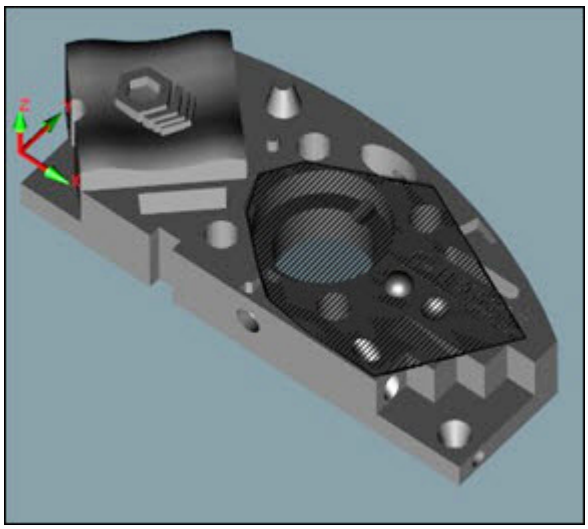
None - PC-DMIS only displays the ID of the constructed plane in the Graphic Display window. It does not display a drawing or outline of the constructed plane.



Triangle - PC-DMIS draws the constructed plane as a shaded triangle. The size of the constructed plane depends on the number of hits that make up the plane.



Outline - PC-DMIS draws the constructed plane as an outline from all the hits. The size depends on the number of hits that make up the constructed plane.



The Display Outline of Plane or Do Not Display Plane options on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**) define the default display state for future measured or constructed planes. They do not affect the display state of existing planes.

Feature Theoreticals

This area contains the theoretical data for the feature. You can update this data by making changes and clicking **OK**. Note that this will only update the theoretical data. If you want to affect the hits and the actual measured data, select the **Regenerate Hits** and **Copy to Actuals** check boxes described above.

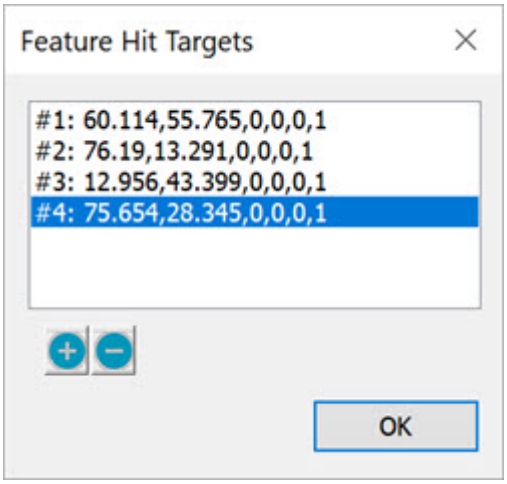
Hit Targets

Supported features:

- Measured Circle
- Measured Cylinder
- Measured Point
- Measured Plane
- Measured Sphere
- Measured Line
- Measured Cone

For supported measured features, the **Hit Targets** button allows you to view or change hit data for the feature.

Clicking the **Hit Targets** button opens the **Feature Hit Targets** dialog box.

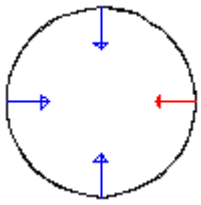


Feature Hit Targets Dialog Box

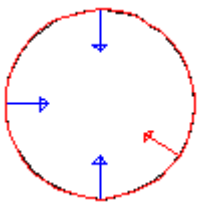
This dialog box lists the hit data by rows. The numbers to the left of the rows indicates the order of the hits. You can change the hit data by double-clicking on a row and editing its values in the resulting **Measured Hits** dialog box.

The **Feature Hits Targets** dialog box also lets you view and edit hit data from the Graphic Display window. As long as the **Feature Hit Targets** dialog box remains open, you can do the following:

- *View individual hits with red colored arrows.* Click any of the rows with hit data listed in the **Feature Hit Targets** dialog box and the associated arrow in the Graphic Display area turns red.



- *Click to move a hit to a new location.* To move a hit to a new location on the CAD, from the **Feature Hit Targets** dialog box, select one of the hits in the list. PC-DMIS changes the color of the vector for the selected hit to red in the Graphic Display window. Click a new location on the CAD in the **Graphic Display** window. The vector relocates to the new position and PC-DMIS recalculates the data and updates the values in the **Feature Hit Targets** dialog box for the selected hit.

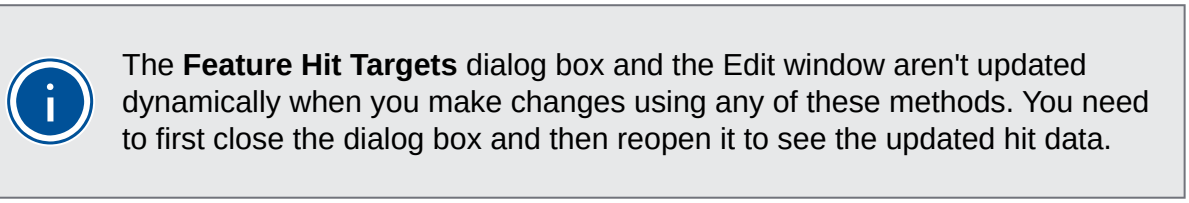


- *Insert new hits.* You can also insert new hits into an existing feature. To do this:

1. From the **Feature Hit Targets** dialog box, select a hit from the list.
2. Click the **+** button to generate a new hit. PC-DMIS generates a new hit at the same location as the selected hit.
3. Click a different location on the CAD in the Graphic Display window to relocate the new hit.
4. Click the **OK** button on the **Feature Hit Targets** dialog box.
5. Click the **OK** button on the **Measured Feature** dialog box. Once you click the **OK** button, PC-DMIS asks if you want to update the feature theoretical values.

   If you select **Yes**, PC-DMIS inserts the new hits you added for the current feature and also updates the feature's theoretical values with the new hits.

   If you select **No**, PC-DMIS inserts the new hits you added for the current feature but does not update the feature's theoretical values with the new hits.

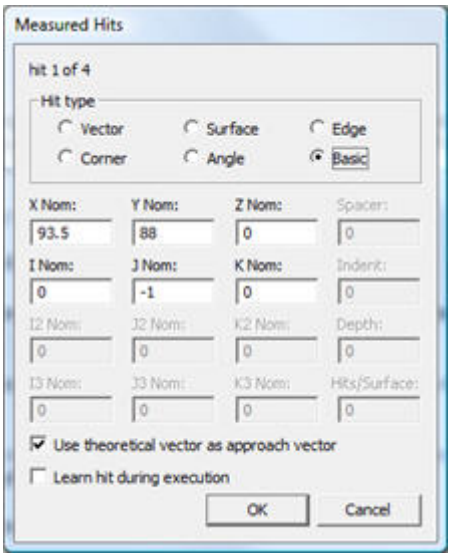


The **Feature Hit Targets** dialog box and the Edit window aren't updated dynamically when you make changes using any of these methods. You need to first close the dialog box and then reopen it to see the updated hit data.

- *Remove hits.* You can remove hits from an existing feature. To do this:

  1. From the **Feature Hit Targets** dialog box, select a hit from the list.
  2. Click the **-** button. PC-DMIS immediately deletes the selected hit.
  3. You can continue selecting and deleting hits for the selected feature this way. PC-DMIS, however, requires that a minimum number of hits remain for the feature type. So, for example, you can delete hits for a plane up until there are three hits remaining. The software will not allow you to delete any more hits for this feature type since a plane requires a minimum number of three hits to define it.

Measured Hits Dialog Box Description



Measured Hits Dialog Box

The **Measured Hits** dialog box allows you to edit individual hits of a measured feature. To access this dialog box, select one of the BASIC hits inside the Edit window and press F9. You can edit the following items:

- Hit Type
- X, Y, and Z position.
- Various vectors
- Spacer
- Indent
- Depth
- Hits per Surface

Some items are unavailable unless you select a specific hit type. To update hit data for the selected measured feature, edit the values in this dialog box, and click **OK**.

Editing Basic Hits by Probing:

You can also update Basic hits by probing the part in Online mode or clicking on the CAD in Offline mode with the **Measured Hits** dialog box open. PC-DMIS automatically updates the dialog box with the new results.

For Basic hits, these check boxes appear:

Use Theoretical Vector As Approach Vector
This check box allows you to specify whether or not the theoretical vector should be used as the approach vector. If deselected, PC-DMIS calculates the approach vector by subtracting the theoretical hit point (X,Y,Z) from the current probe position.

Learn Hit During Execution
This check box allows you to specify whether or not PC-DMIS should relearn the basic hit during measurement routine execution.

Overriding a Guessed Measured Feature

For information, see "Overriding Guessed Features" in the "Editing a Measurement Routine" chapter.

Creating Measured Feature Sets

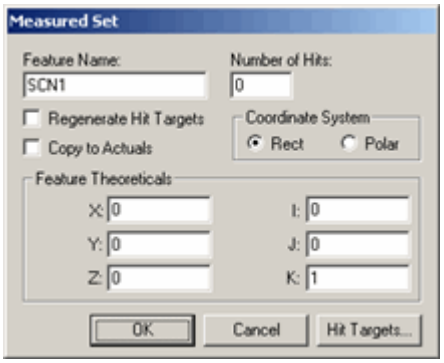
You can measure a single point multiple times as a measured feature set (also called a Point Set). The **Insert | Feature | Measured | Feature Set** menu item inserts a `FEAT/SET` command into the Edit window. This measured feature command creates a scan of a single point that measures the same point a defined number of times to obtain an averaged (and hopefully more accurate) representation of the point measurement.

Point Sets are most often used as inputs when performing Leapfrog alignment operations with a manual arm machine. See "Performing a Leapfrog Operation" in the "Creating and Using Alignments" chapter for more information.

In Command Mode, the Measured Feature Set looks like this:

```
SCN1=FEAT/SET,RECT
   THEO/0,0,0,0,0,1
   ACTL/0,0,0,0,0,1
   MEAS/SET,0
   ENDMEAS/
```

You can press F9 while your cursor is in the command to display the **Measured Set** dialog box to edit this command:



Measured Set Dialog Box

For information on this dialog box, see the "Editing a Measured Feature" topic above. The **Measured Feature** dialog box contains many of the same items that are already discussed in that topic.

To Create a Measured Feature Set (Point Set):

1. Select **Insert | Feature | Measured | Feature Set** to insert a measured feature set command.
2. Press F9 on the inserted measure feature set command.
3. In the dialog box, in the **Feature Theoretics** area, type in the point's theoretical XYZ location and IJK vector.
4. In the **Number of Hits** box, specify the number of measurements you want PC-DMIS to take on this point. The higher the number, the better the average you will receive.
5. Select other options in the dialog box as desired.
6. Click **OK**. PC-DMIS updates the command in the Edit window.

   For example, a finished point set scan with 5 iterations might look like this:

```
SCN1       =FEAT/SET,RECT
            THEO/107,11,21,0,0,1
            ACTL/0,0,0,0,0,1
            MEAS/SET,5
            HIT/BASIC,NORMAL,107,11,21,0,0,1,0,0,0,USE
THEO = YES
            HIT/BASIC,NORMAL,107,11,21,0,0,1,0,0,0,USE
THEO = YES
            HIT/BASIC,NORMAL,107,11,21,0,0,1,0,0,0,USE
THEO = YES
            HIT/BASIC,NORMAL,107,11,21,0,0,1,0,0,0,USE
THEO = YES
            HIT/BASIC,NORMAL,107,11,21,0,0,1,0,0,0,USE
THEO = YES
            ENDMEAS/
```

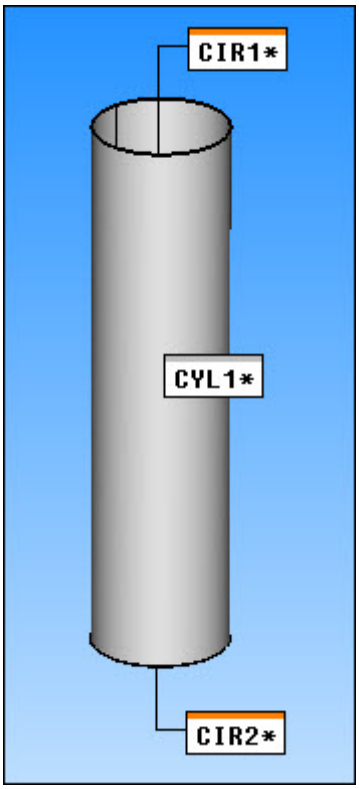
Constructing New Features from Existing Features

Constructing New Features from Existing Features: Introduction

Use the **Insert | Feature | Constructed** submenu to construct features when probing on a required element is not possible (for example, the intersection point of two edges).

The items on this menu enable you to create features (points, lines, circles, and other features) from existing features (ones already probed or constructed). The "Navigating the User Interface" chapter describes the several ways to specify input features. See the "Selecting Features Using the Graphic Display Window" topic in the "Editing the CAD Display" chapter for a description of the several ways to specify input features.

Once you construct the feature, PC-DMIS draws the new feature on the screen. For 3D features (Cylinder, Sphere, and Cone) and for a 2D plane, PC-DMIS draws the feature with a shaded surface.



Example of a cylinder feature (CYL1) drawn with shaded surfaces and constructed from two circles (CIR1 and CIR2).

You can use Hit arrays to define the inputs for constructed features. For details, see the "Using Hit Arrays to Define Constructed Feature Inputs" section of the "Hit Arrays" help topic in the "Using Expressions and Variables" chapter of the PC-DMIS Core documentation.

Hiding Shaded Plane Features

To hide shaded planes, set the **None** option in the **Display** area of the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**). To globally hide all drawn shaded planes for future plane features, select the **Do Not Display Plane** check box in the **Setup Options** dialog box.

Changing the Feature Color

If desired, you can use the **ID Setup** tab in the **Setup Options** dialog box to modify the feature color used during feature creation. See the **Color** check box that appears after you choose **Features** under the **Labels For** item.

You can construct a feature by selecting features in the dialog boxes or the Edit window (see "To Construct a Feature from the Edit Window" and "To Construct a Feature Using the Construction Dialog Boxes:" below). You can find specific rules to construct a feature from other features from the topics below. These rules apply to either method of construction.

The default method to construct a feature is **Auto**. In this case, PC-DMIS automatically determines the best construction type based on the input feature or features. The order of selection (unless noted otherwise) is usually not important. It is only necessary that you choose the correct feature type or types. For example, to construct an intersect point between a line and a slot, simply select the necessary line and slot. PC-DMIS creates a point where the line intersects the slot.

It is important to note that the documented conventions are just tools that allow you to become familiar with the rules that govern the Edit window. The actual Edit report displays in all capital letters.

Here are two command line examples that show a constructed point in the Edit window:

```
CONSTR/POINT, INTOF, line_id, slot_id
```

or

```
CONSTR/POINT, INTOF, slot_id, line_id
```

The main topics in this chapter include:

- Understanding the General Command Format
- Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions
- Specifying Feature Theoreticals
- Constructing a Point Feature

- Constructing a Line Feature
- Constructing a Plane Feature
- Constructing a Circle Feature
- Constructing an Ellipse Feature
- Constructing a Round Slot Feature
- Constructing a Square Slot Feature
- Constructing a Curve
- Constructing a Cylinder Feature
- Constructing a Cone Feature
- Constructing a Sphere Feature
- Constructing a Surface
- Constructing a Set of Features
- Constructing a Filter Set
- Constructing an Adjusted Filter
- Constructing a Width Feature

To Construct a Feature from the Edit Window

1. Access the Edit window (**View | Edit Window**).
2. Place the cursor where you want to construct the feature in the Edit window.
3. Type the command `CONST` , and press the Tab key.
4. In the Edit window, locate the `CONSTR/` line.
5. Click on that line, and press the Tab key until you highlight the feature type on the right side of the slash.
6. Press F7 and F8, and choose the type of feature to construct. (For an example of the command line for a constructed point, see "Constructing New Features from Existing Features: Introduction" above).

To Construct a Feature Using the Construction Dialog Boxes

1. Select the **Insert | Feature | Constructed** menu.

2. From the **Constructed** menu, select the type of feature to construct.
3. From the dialog box, select the input features.

You can also use the Graphical Hit Selection method to select inputs to your construction. For details, see "Graphical Hit Selection Method".

4. From the dialog box, choose other options as needed. For information on the available options, see the specific constructed feature type.
5. Click the **Create** button.

PC-DMIS constructs the feature and then displays it on the screen. It also displays the feature's center point in the Edit window. PC-DMIS constructs the requested feature, leaves the dialog box open, and adds the newly-constructed feature into the dialog box. This allows you to construct multiple features based on newly-constructed features.

Understanding the General Command Format

In the Edit window, all constructed features display in the following format. There are slight variations that are explained more thoroughly in the following sections.

The actual Edit report displays in all capital letters.

For example:

```
Feature_name=FEAT/TYPE OF FEATURE,TOG1,.....
THEO /x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,......
ACTL /x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,......
CONSTR/TOG2,TOG3,......
```

feature_name - The name of the feature. This is an editable field.

TYPE OF FEATURE - This field denotes the feature type. This feature type is the same as TOG2 but it is not editable, and it is not a toggle field.

..... : This indicates that what follows is unique to each feature. This is explained more thoroughly as needed in the appropriate topics.

TOG1 - This field is present in all features and can be switched between RECT / POLR. If this feature is RECT, all points are represented in the Cartesian system (x, y, z). If the field is POLR, all points display in the polar system (x_radius, y_angle, z_height). Vectors do not change.

TOG2 - This field is present in all features and can be switched between the different types of features through the following values:

> CIRCLE / CONE / CYLINDER / LINE / PLANE / POINT / SPHERE / CURVE / SURFACE / SET /
>
> POINT is the default feature type when the construct command line is opened for the first time in a measurement routine. The default feature type then becomes the last constructed feature type.

TOG3 - This field is also present in all features, but it takes unique values according to the type of feature. (See the specific feature type for additional information.)

Accessing the Construct [Feature] Dialog Box from a Constructed Feature

To access the corresponding construction dialog box from which an Edit window's feature was created:

1. Make sure the Edit window is open.
2. In the Edit window, click the mouse on the feature's name.
3. Press the F9 key to open the construction dialog box.

Use this dialog box to make any changes. When you click the **Create** button, the updated changes are made to the Edit window.

Decimal Placement Conventions

The following statements refer to metric measurements where the maximum number of decimal places is 6. For standard measurement (inches), the maximum number of decimal places is seven.

Please note the following information about decimal place conventions for constructed features:

- All vectors (i_vect, j_vect, k_vect) are represented by a maximum of six places to the right of the decimal.
- All lengths and distances (x_cord, y_cord, z_cord, diam, height, length, etc.) are represented by a maximum of six places to the right of the decimal.
- All angles are represented by a maximum of six places to the right of the decimal.
- If the decimal place is zero, then the number is represented as an integer.

You can change the decimal places through the **Dimension** tab (F5) in the **Setup Options** dialog box (**Edit | Preferences | Setup**). If you set the decimal place limit, PC-DMIS inserts a `DISPLAYPRECISION` command in the Edit window. All features that follow this command conform to the specified number of decimal places.

Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions

Best Fit Recompensate (BFRE) is only accurate when you construct a feature using actual surface data.

If the inputs for the construction are a mixture of features that have surface data and features that do not, you should use the Best Fit (BF) calculation method.

If none of the inputs for the construction have surface data and you choose BFRE, PC-DMIS uses the Best Fit calculation method internally, instead.

The table below shows a list of feature types that have actual surface data (left), and the feature types that don't have surface data (right).

Features That Contain Actual Surface Data	Features That Do Not Contain Surface Data
Measured Points	Generic Features (All Types)
Directly Referenced Hits of Measured Lines	Auto Vector Points – Self Centring Point Strategy
Directly Referenced Hits of Measured Planes	Auto Edge Points (All Sensor Types)
Directly Referenced Hits of Measured Circles	Auto Angle Points (All Sensor Types)
Directly Referenced Hits of Measured Cylinders	Auto Corner Points (All Sensor Types)
Directly Referenced Hits of Measured Cones	Auto High Points (All Sensor Types)
Directly Referenced Hits of Measured Spheres	Laser/Vision Auto Surface Points
Directly Referenced Hits of Measured Toruses	Vision Auto Lines
Directly Referenced Hits of Measured Sets	Laser Auto Planes
Probed Auto Vector Points	Laser/Vision Auto Circles
Probed Auto Surface Points	Laser Auto Cylinders
Directly Referenced Hits of probed Auto Lines (All Strategies)	Laser Auto Cones

Directly Referenced Hits of probed Auto Planes (All Strategies)	Laser Auto Spheres
Directly Referenced Hits of probed Auto Circles (All Strategies)	Laser/Vision Auto Ellipses
Directly Referenced Hits of probed Auto Round Slots	Laser/Vision Auto Polygons
Directly Referenced Hits of probed Auto Square Slots	Laser/Vision Auto Round Slots
Directly Referenced Hits of probed Auto Notch Slots	Laser/Vision Auto Square Slots
Directly Referenced Hits of probed Auto Cylinders (All Strategies)	Laser/Vision Auto Notch Slots
Directly Referenced Hits of probed Auto Spheres	Laser Auto Flush & Gaps
Directly Referenced Hits of probed Auto Ellipses	Vision Auto 2D Profiles
Directly Referenced Hits of probed Auto Polygons	Laser Pointcloud data
Directly Referenced Hits of probed Scans (except Basic Center scans)	Laser Mesh data
	Gages (all types)
	Constructed Features (all types)
	Basic Center Scans

	Uni-Scans
	Scans performed with a laser sensor

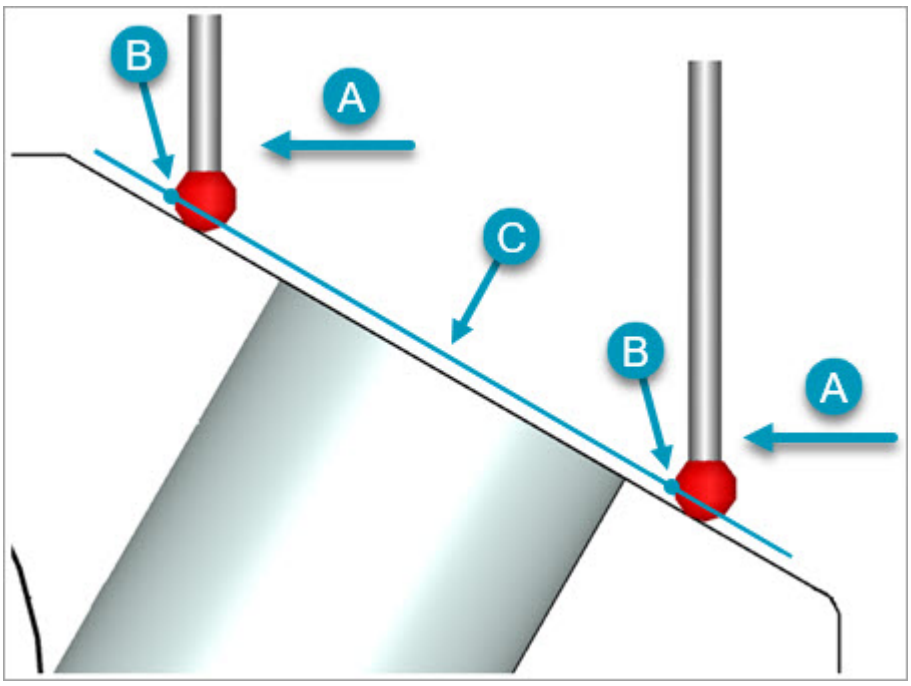
The Best Fit construction method takes the actual measured points in their pre-compensated state. The Best Fit Recompensate method takes the ball's center and applies further re-compensation after fitting the feature. In both cases, the average squared error is minimized in the Least Squares method and the maximal error is minimized in the Min/Max method. You can also choose to remove outliers or apply a Gaussian filter to the constructed Line, Plane or Circle.

- The Best Fit option uses the ball center data shifted one tip radius in the direction of the negated hit vector to perform the compensation prior to fitting.
- The Best Fit Recompensate option uses the ball center data and the tip compensation as part of the fitting process.

Consider the following example where the probe is not approaching normal to the surface. The resultant hits are compensated in the direction of travel (**A**), resulting in some amount of measurement error.

> This example is an extreme case, deliberately chosen for illustration purposes only.

If you construct a BF line from the points, it will also be in error because it passes through the pre-compensated hit locations.



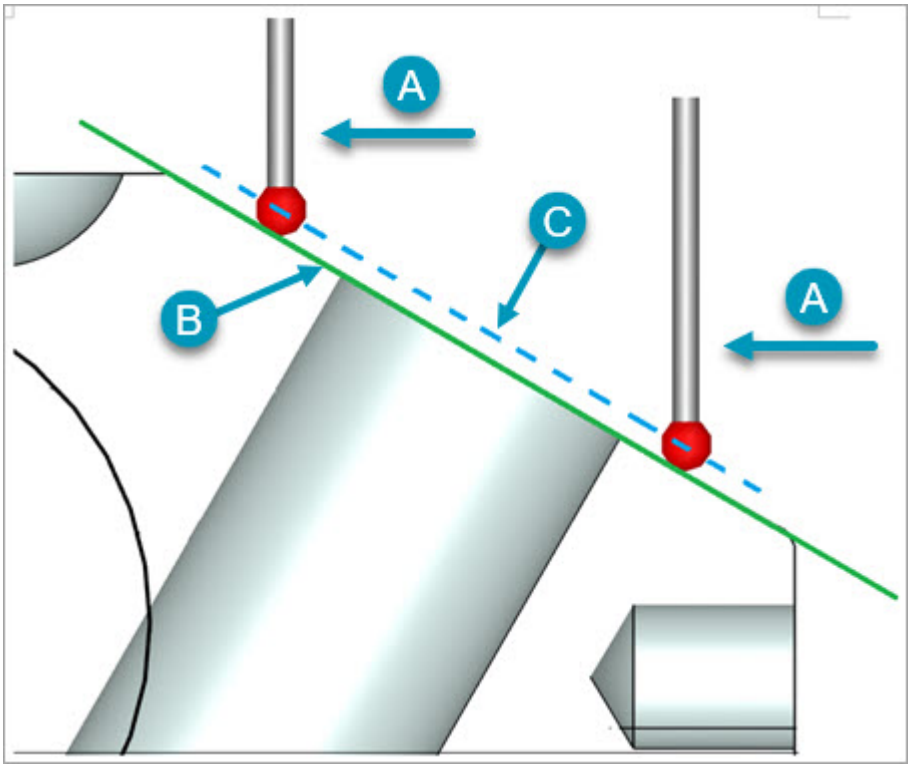
Example of probe not approaching normal to the surface using the BF method

A. *Probe direction*

B. *Compensated points*

C. *Constructed Best Fit Line passing through both pre-compensated points*

However, if you construct a BFRE line, it corrects for this error because PC-DMIS applies the radius compensation to the line ***AFTER*** it first fits it through the ball centers.



Example showing probe not approaching normal to the surface using the BFRE method

A. *Probe direction*

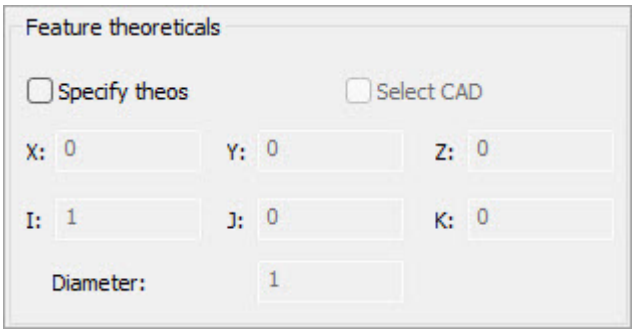
B. *BFRE Line, compensation applied normal to the line*

C. *Initial line fitted through the ball centers*

Specifying Feature Theoreticals

PC-DMIS provides the ability to specify theoretical information for most of the available constructed feature types. Traditionally, PC-DMIS uses theoretical values from the input features to calculate a theoretical value for the constructed feature. In some circumstances, however, you may not want this result. To make constructed features more flexible, you can override the traditional behavior and specify your own theoretical values for the feature.

The **Feature theoreticals** area is at the bottom-right of the constructed feature's dialog box (**Insert | Feature | Constructed**). This area is unavailable for selection until you select the **Specify theos** check box. This enables the other items in the area, which you can use to override the calculated theoretical values from the input features with the specified theoretical values.



Example of the Feature Theoreticals area for a constructed circle feature.

The items that appear in the **Feature theoreticals** area change, depending on the feature you are constructing. Each feature displays at least the **X**, **Y**, and **Z** boxes (the location) and the **I**, **J**, and **K** boxes (the orientation). For features with characteristic sizes, such as circles, slots, and cones, additional items appear in this area.

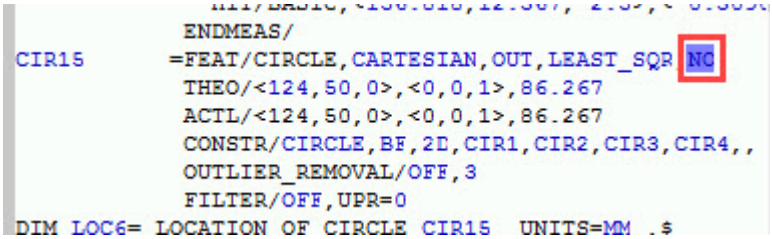
Overriding Feature Theoreticals

To change an existing feature from the traditional method to the new theoretical override:

1. From the Edit window, press **F9** to see the dialog box for that constructed feature.
2. Select the **Specify theos** check box to enable the **Feature theoretical** area.
3. Change the theoretical values by:
    - Manually editing the available items.
    - Select the **Select CAD** check box and then click the corresponding element on the CAD model.

4. Click the **Create** button when finished.

Alternately, you can edit the feature from the Edit window. To do this, in Command mode, set the last field on the first line of the feature to **YES** and then manually type the values in the `THEO` line.



Sample Constructed Feature in Command Mode

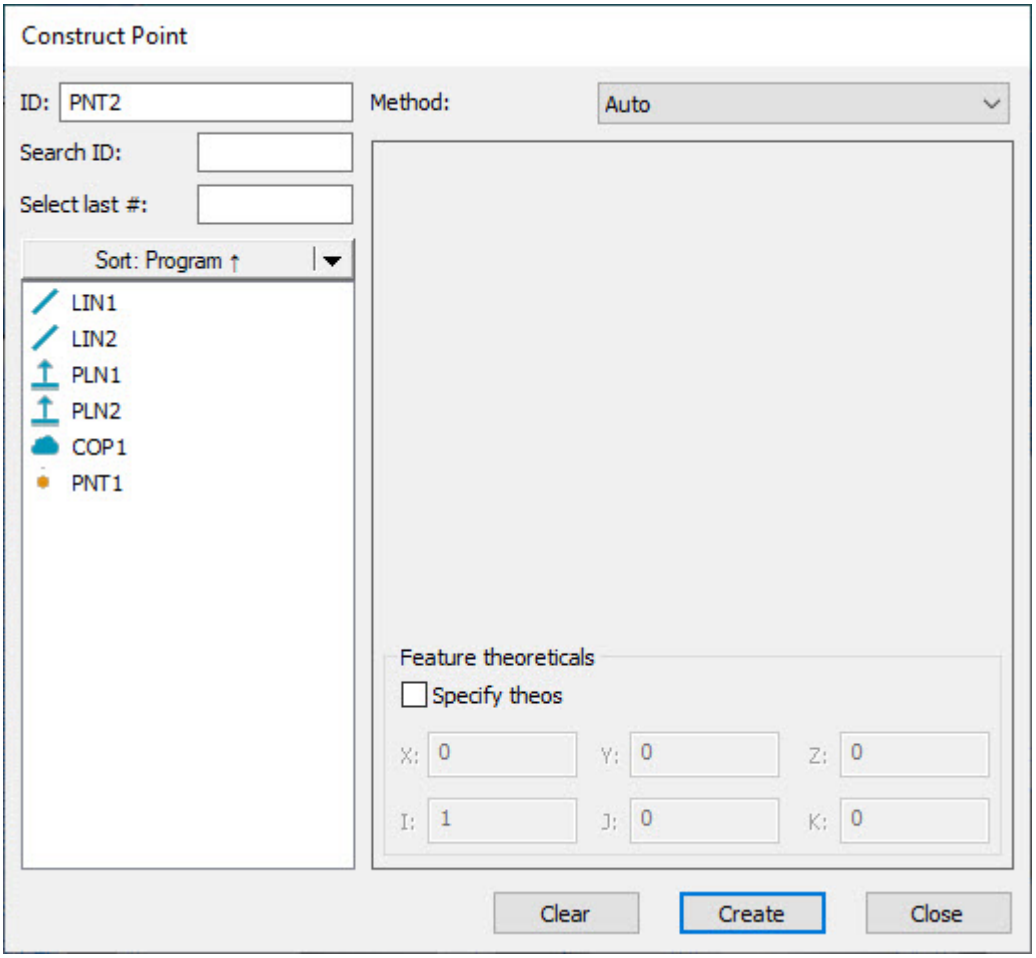
To have PC-DMIS automatically calculate the feature information based on the input features (the traditional method), clear the **Specify theos** check box in the dialog box or change the appropriate value in the Edit window. The features update accordingly.

Because you are editing feature theoretical values, PC-DMIS asks if you want to update your measured values and the theoretical values of any associated dimensions. This is standard PC-DMIS behavior.

All constructed features have this capability except for:

Specify Theos	Select CAD
Adjust Filter	Adjust Filter
Curve	Curve
Filter	Filter
Set	Line
Surface	Point
	Set
	Surface
	Width

Constructing a Point Feature



Construct Point Dialog Box

There are a number of ways to use PC-DMIS to construct a point. The following table lists the various types of constructed points along with their necessary inputs. Some features may require no inputs while others may require three inputs or more. In the table, the term "Any" indicates that the construction can take *any* type of feature as input for construction. PC-DMIS allows the features to be selected in any order.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Tertiary Feature	Comments
Auto Point	-	-	-	-	-	See "Auto Point Construction".
Intersect Point	INTOF	2	See topic below for valid features.	See topic below for valid features.	-	Constructs a point at the intersection of the linear attribute of two features.
Origin Point	ORIGIN	0	-	-	-	Constructs a point at the alignment origin.
Drop Point	DROP	2	Any	Cone, Cylinder, Line, Slot	-	First feature is dropped onto the second line feature.
Cast Point	CAST	1	Any	-	-	Constructs a point at the centroid of the input feature.
Mid Point	MID	2	Any	Any	-	Constructs a mid point between the centroids of the inputs.
Corner Point	CORNER	3	Plane	Plane	Plane	Constructs a point at the intersection of three planes.
Project Point	PROJ	1 or 2	Any	Plane	-	One input feature will project the point to the workplane.

Pierce Point	PIERCE	2	See topic below for valid features.	See topic below for valid features.	-	Constructs a point where one feature pierces the surface of another feature.
Vector Distance Point	VECT_DIST	2	Any	Any	-	Constructs a point at a distance from the second feature along the line from the two input features.
Offset Point	OFFSET	1	Any	-	-	Requires three offsets corresponding to X,Y, and Z.
Tertiary Datum Point	TERTIARY_DATUM	1	Point, Plane, Line, Set of Points	(Additional Point features if first feature is a point)	(Additional Point features if first feature is a point)	Constructs a point which simulates a tertiary datum that is external to material.
Extracted Edge Point	EXTRACTED_EDGE_POINT	1	COP or Mesh	-	-	Extracts and constructs a point from the selected COP or Mesh command.
Extracted Surface Point	EXTRACTED_SURFACE_POINT	1	COP or Mesh	-	-	Extracts and constructs a surface point from the selected COP or Mesh command.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

To construct a point, do the following:

1. Open the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. From the **Method** list, select the feature type to construct.
3. From the **Feature Selection** list, select the features to use to create the constructed point. If the feature type requires you to select more than one feature, select the features in the order you want PC-DMIS to process them in: Primary, Secondary and Tertiary.
4. In the **Parameters** area, select and enter the values you want to apply to the selected method. The available parameters are dependent on the selected method. For example, if you choose **Auto** from the **Method** list, you can select the **Specify theos** check box in the **Feature theoreticals** area and then enter the **X**, **Y**, **Z**, **I**, **J**, and **K** values.
5. Click the **Create** button.

The Edit window command line for a sample point construction would read:

```
feature_name=FEAT/POINT,TOG1
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
CONSTR/TOG2,TOG3,......
```

The actual Edit report displays in all capital letters.

TOG1 = POLR or RECT

TOG2 = POINT

TOG3 = CAST / CORNER / DROP / INTOF / MID / OFFSET / ORIG / PIERCE / PROJ / TERTIARY_DATUM

The first three lines that display in the Edit window are the same for all constructed points. The fourth line is slightly different according to the type of feature you are constructing. To switch between the different types of points, place the cursor on TOG3 and press F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

AUTO is the default method of construction. This option automatically determines the best way to construct a point using the input feature or features. See "Auto Point Construction".

The following topics describe the options for constructing a point.

Auto Point Construction

The list below indicates the type of point that the software constructs when you select certain inputs along with the **Auto** option. The order of the selected features is not important. If you select an incorrect input feature or features, PC-DMIS displays an error message and does not automatically construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Auto** option from the **Method** list.
3. Select the desired feature or features based on the information below.
4. To change the feature theoreticals values, click the **Specify theos** check box in the **Feature Theoreticals** section and update the **X**, **Y**, **Z**, **I**, **J**, and **K** values as necessary.
5. Click the **Create** button.

Input Feature List

Along with the table below, the following lists each input feature and what it can construct:

- 3 Points = Corner Point
- Any one feature = Cast Point

<table>
<tr><th></th><th>Circle</th><th>Cone</th><th>Cylinder</th><th>Ellipse</th><th>Line</th><th>Plane</th><th>Point</th><th>Set</th><th>Slot</th><th>Sphere</th></tr>
<tr><td>Circle</td><td>Mid Point</td><td colspan="2">Drop Point</td><td>Mid Point</td><td>Pierce Point</td><td>Project Point</td><td colspan="4">Mid Point</td></tr>
<tr><td>Cone</td><td rowspan="2">Drop Point</td><td colspan="2" rowspan="2">Intersection Point</td><td rowspan="2">Drop Point</td><td rowspan="2">Intersection Point</td><td rowspan="2">Pierce Point</td><td colspan="2" rowspan="2">Drop Point</td><td rowspan="2">Intersection Point</td><td rowspan="2">Drop Point</td></tr>
<tr><td>Cylinder</td></tr>
<tr><td>Ellipse</td><td>Mid Point</td><td colspan="2">Drop Point</td><td>Mid Point</td><td>Drop Point</td><td>Project Point</td><td colspan="4">Mid Point</td></tr>
<tr><td>Line</td><td>Pierce Point</td><td colspan="2">Intersection Point</td><td>Drop Point</td><td>Intersection Point</td><td>Pierce Point</td><td colspan="2">Drop Point</td><td>Intersection Point</td><td>Drop Point</td></tr>
<tr><td>Plane</td><td>Project Point</td><td colspan="2">Pierce Point</td><td>Project Point</td><td>Pierce Point</td><td colspan="5">Project Point</td></tr>
<tr><td>Point</td><td rowspan="3">Mid Point</td><td colspan="2" rowspan="2">Drop Point</td><td rowspan="3">Mid Point</td><td rowspan="2">Drop Point</td><td rowspan="3">Project Point</td><td colspan="4" rowspan="2">Mid Point</td></tr>
<tr><td>Set</td></tr>
<tr><td>Slot</td><td colspan="2">Intersection Point</td><td>Intersection Point</td><td colspan="2">Mid Point</td><td>Intersection Point</td><td>Mid Point</td></tr>
<tr><td>Sphere</td><td colspan="2">Drop Point</td><td>Drop Point</td><td colspan="4">Mid Point</td></tr>
</table>

Mesh and Pointcloud Feature List

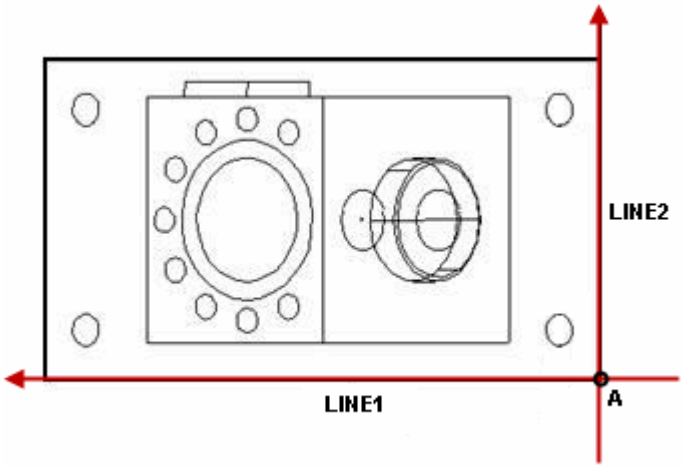
- You can extract and construct an Edge Point from a COP or Mesh command. For details, see "Constructing an Extracted Edge Point" in this document.
- You can extract and construct a Surface Point from a COP or Mesh command. For details, see "Constructing an Extracted Surface Point" in this document.

Constructing an Intersection Point

You can construct a point between two valid features:

First Valid Feature	Second Valid Feature
Plane	Line, Cylinder, Cone, or Curve
Line	Plane, Sphere, Cone, Cylinder, Curve, Line, Circle, or Slot
Cylinder	Plane, Line, Cone, Cylinder, Circle, or Slot
Cone	Plane, Line, Cone, Cylinder, Circle, or Slot
Curve	Plane, Line
Sphere	Line
Circle	Line, Cone, Cylinder, Circle, or Slot

The point is created where the lines (centerlines) of the two features intersect or where a linear feature pierces a planar feature.



Constructing an intersection point from two lines
***A** - Point constructed from two lines (LINE1 and LINE2)*

If you want to intersect or pierce a circle with a line, see "Constructing a Pierce Point".

To construct an Intersect Point:

1. Select **Insert | Feature | Constructed | Point** to display the **Construct Point** dialog box.
2. Select the **Intersection** option from the **Method** list.
3. Use the table above to select the first and second valid features types.

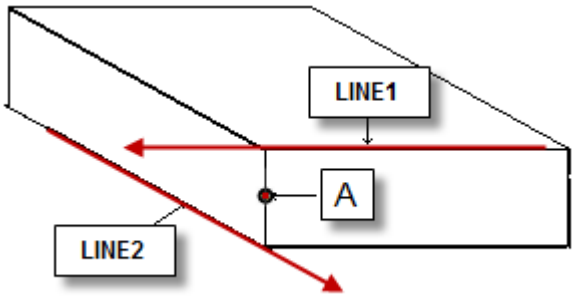
4. To change the feature theoreticals values, click the **Specify theos** check box in the **Feature Theoreticals** section and update the **X**, **Y**, **Z**, **I**, **J**, and **K** values as necessary.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,INTOF,feat_1,feat_2
```

If the two features do not intersect, the point is constructed midway between the two features at the apparent intersection. In other words, the intersect point is the mid-point of the shortest line connecting the two input features.

In this example, **A** indicates where the intersect point is constructed between LINE1 and LINE2:



Example of an Intersection Point constructed from two non-intersecting lines.

You can intersect two circle features and as long as they have the same (or very similar) vectors, PC-DMIS generates a constructed point feature at one of the intersection points. If you switch the order of the selected input features in the **Construct Point** dialog box, PC-DMIS constructs a point at the other intersection point.

You can intersect a line feature with a constructed curve. The line and curve are first projected onto the current workplane, where the intersection is computed. If there is no intersection, an error message appears. Even though there may be many intersection points, PC-DMIS only reports the one closest to the start of the curve. To get the other intersection points, it may be possible to subdivide the curve and compute intersections on the sub curves.

If you select a slot as one of your input features, PC-DMIS uses the slot centerline vector when constructing this point instead of the slot normal vector. If you want to use the older slot normal vector, you need to modify the value for the `UseLegacySlotVector` entry in PC-DMIS Settings Editor. For information on how to modify the value, see the "Modifying Setting Entries" topic in the PC-DMIS Core documentation.

Constructing a Point at the Origin

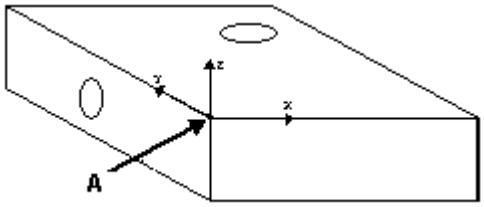
You can construct a point at the current alignment origin.

To construct an origin point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **At Origin** option.
3. Do *not* select any input features.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,ORIG
```



Example of a point feature constructed at the origin.

***A** - Point constructed at the origin (0,0,0)*

Constructing a Drop Point

You can construct a point from any feature and a line (cone, cylinder, or slot). PC-DMIS drops the centroid of the first feature onto the second feature. The "dropped" point is dropped on a line that is perpendicular to the line, centerline, or plane. If two line features are selected, PC-DMIS drops the centroid of the first line feature onto the second line feature.

For this construction method, you must select feature types in the correct order.

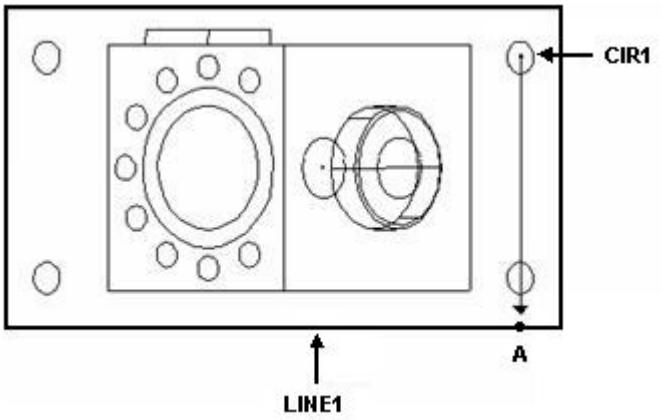
To construct a drop point:

1. Select **Insert | Feature | Constructed | Point** to display the **Construct Point** dialog box.
2. Select the **Drop Point** option from the list of options.
3. Select the first feature. It may be of any type.
4. Select the second feature. It must be cone, cylinder, line, or slot.

5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,DROP,feat_1,feat_2
```



A - Point is constructed by normally projecting circle, CIR1 (the designated point) to line, LINE1
Example showing a drop point constructed from a circle and a line.

If you select a slot as one of your input features, PC-DMIS uses the slot centerline vector when it constructs this point instead of the slot normal vector. If you want to use the older slot normal vector, you need to modify the value for the `UseLegacySlotVector` entry in PC-DMIS Settings Editor. For information on how to do this, see "Modifying Setting Entries".

Constructing a Cast Point

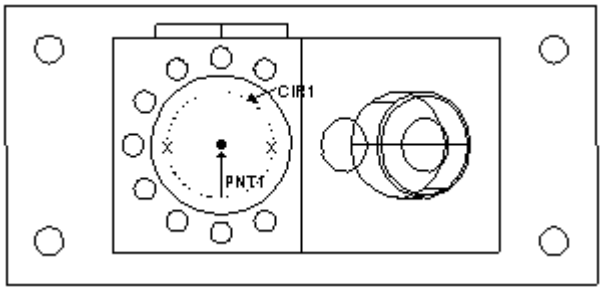
You can construct a point by changing any given feature into the point. PC-DMIS constructs the point at the centroid of the input feature.

To construct a cast point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Cast** option from the list of options.
3. Select any feature.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,CAST,feat_1
```



Constructing a cast point from a circle.

Constructing a Mid Point

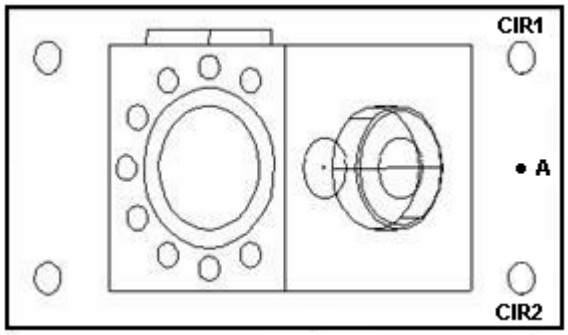
You can construct a point from any two features without direction. PC-DMIS creates a mid point between the centroids of the two input features.

To construct a mid point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Mid Point** option from the list of options.
3. Select any two features.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,MID,feat_1,feat_2
```



Example showing a mid point constructed from two circles.

***A** - Point constructed midway between two circles, CIR1 and CIR2*

Constructing a Corner Point

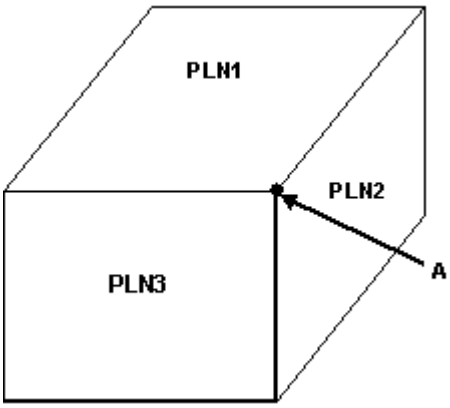
You can construct a point from three planes. PC-DMIS creates a point at their intersection. The vector of the constructed corner point is the cross product of the second input vector into the third input vector.

To construct a corner point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Corner Point** option from the list of options.
3. Select three different planes.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,CORNER,feat_1,feat_2,feat_3
```



A - Point constructed at the intersection of three planes (PLN1, PLN2, and PLN3)
Constructing a corner point from three planes

Constructing a Projected Point

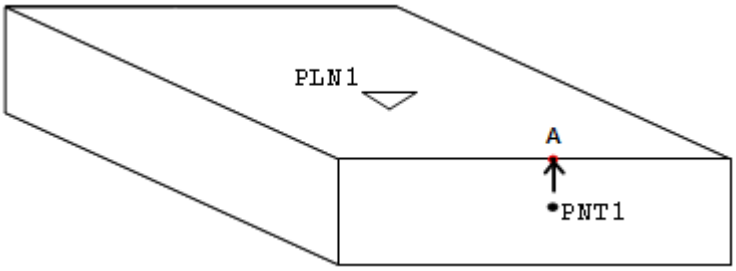
You can construct a point from any feature and a plane. PC-DMIS projects the point where the plane intersects the point. If there is only one input feature, the projection will be to the workplane.

To construct a projected point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Projection** option from the list of options.
3. Select a feature from which to create the projected point.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,PROJ,feat_1,(feat_2)
```



Example of a project point constructed from a point and a plane.

***A** - Projected point constructed from a point, PNT1, and a plane, PLN1*

Constructing a Pierce Point

You can construct a point where one feature pierces the surface of another feature:

First Valid Feature	Second Valid Feature
Line	Plane, Sphere, Cylinder, Circle, Cone, or Ellipse
Slot	Plane, Sphere, Cylinder, Circle, Cone, or Ellipse
Cone	Plane, Sphere, Cylinder, Circle, Cone, Ellipse, Line, or Slot
Cylinder	Plane, Sphere, Cylinder, Circle, Cone, Ellipse, Line, or Slot
Circle	Plane, Sphere, Cylinder, Circle, Cone, Ellipse, Line, or Slot
Ellipse	Plane, Sphere, Cylinder, Circle, Cone, Ellipse, Line, or Slot
Plane	Line, Slot, Cone, or Cylinder
Sphere	Line, Slot, Cone, or Cylinder

Normally, the first feature selected would be the surface to pierce, with these exceptions:

- If one of the inputs is a Plane, Sphere, Circle, or Ellipse, no matter what the input order is, this feature becomes the pierced surface.
- If the second feature is a cylinder and the first input is not a cylinder, circle, cone, or line (these are all line-reducible features), then the cylinder becomes the line-reducible feature, and the first feature becomes the pierced surface.

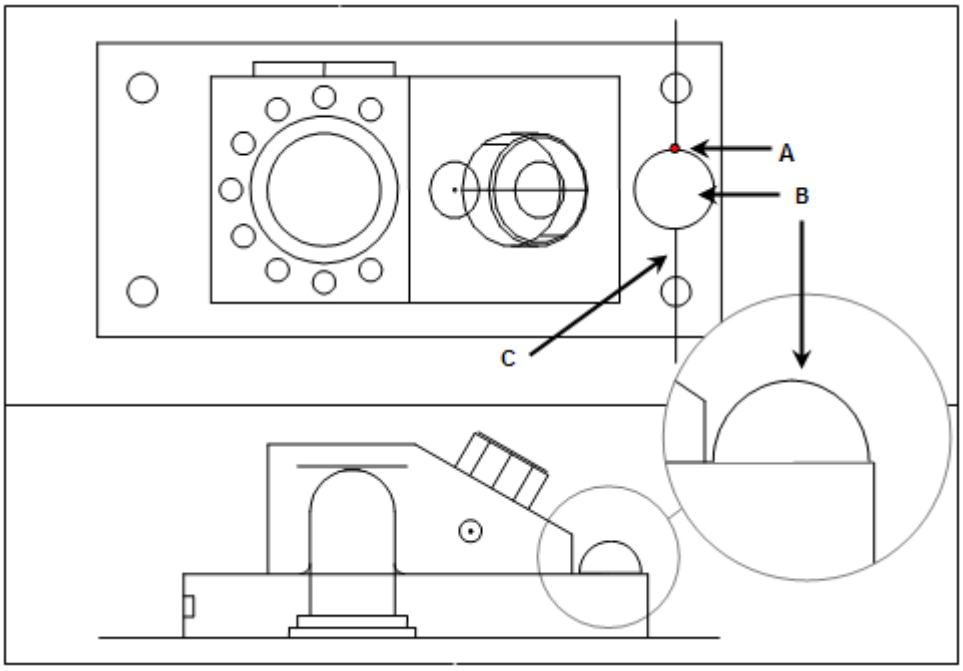
- If the second feature is a cone and the first feature is not a cylinder, circle, cone, line, or slot, then the cone becomes the line-reducible feature and pierces the surface of the first input.

To construct a pierce point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Pierce** option from the list of options.
3. Select a first valid feature.
4. Select a second valid feature.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/POINT,PIERCE,feat_1,feat_2
```



Constructing a pierce point from a Line and a Sphere
A *- Point constructed at the intersection of a line and sphere*
B *- Sphere feature*
C *- Line (please note the direction)*

Additional notes:

- If the pierced feature is a circle or an ellipse, PC-DMIS creates a cylindrical circumference about the centerline, and constructs the pierce point. If two similar features (two cylinders, for example) are provided, PC-DMIS pierces the second feature with the first feature.

- If the pierced feature is a sphere, circle, cone, or cylinder, the pierce point is constructed where the piercing input feature first intersects the surface of the pierced element. The first intersection point is determined by the direction of the line. It is important to know the direction in which the line was defined. If the wrong point is constructed, create a new, flipped line (see "Changing the Direction of a Line"), and use it to construct the point.

Constructing a Vector Distance Point

You can construct a point at a specified distance along an imaginary line created from two input features. PC-DMIS constructs the point along the line from the first input to the second input at a specified distance from the second input.

For example, if your two input features were in order, PNT1 and PNT2, and you defined a distance of 10 mm, PC-DMIS would construct the point (PNT3) like this:

Example of a constructed vector distance point (PNT3) created from input features (PNT1 and PNT2).

To construct a vector distance point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Vector Distance** option from the **Method** list.
3. Select the first feature.
4. Select the second feature.
5. Specify a distance in the **Distance** box. You can type a negative value to construct the point *between* the two input features.
6. To change the feature theoreticals values, click the **Specify theos** check box in the **Feature Theoreticals** section and update the **X**, **Y**, **Z**, **I**, **J**, and **K** values as necessary.
7. Click the **Create** button. PC-DMIS constructs a point at the specified distance from the second input feature along the line from the first feature to the second feature.

The Edit window command line for this option would read:

```
CONSTR/POINT,VECT_DIST,feat_1,feat_2,distance
```

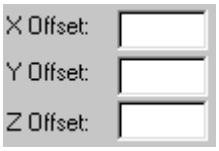
Constructing an Offset Point

You can construct a point a specified distance from any input feature.

To construct an offset point:

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Offset Point** option from the **Method** list.
3. Select a feature from which to create the offset point.
4. Enter the offset values in the **X offset**, **Y offset**, and **Z offset** boxes as necessary to create an offset from the origin. To create an offset point relative to a particular feature, select the desired feature and type the values in the **X offset**, **Y offset**, and **Z offset** boxes.
5. To change the feature theoreticals values, click the **Specify theos** check box in the **Feature Theoreticals** section and update the **X**, **Y**, **Z**, **I**, **J**, and **K** values as necessary.
6. Click the **Create** button.

XYZ Offset



These fields allow you to enter in offset distances for the X, Y, and Z, axes. These options remain unavailable until you select the **Offset Point** option from the **Method** list.

The Edit window command line for this option would read:

```
CONSTR/POINT,OFFSET,feat_1,dx,dy,dz
```

Constructing a Tertiary Datum Point

You can construct a Tertiary Datum Point from a plane, line, or set of points. The set of points can be multiple selected points, a constructed set of points, or a scan feature that contains multiple points.

The tertiary datum construction uses the points within the input feature (or features), as well as with the specified primary and secondary datums to create an "orientation constrained" and "external to material" point. You can use this point in alignments to simulate a tertiary datum.

- "Orientation constrained" means that the resulting point feature is solved with a nominal orientation, with respect to the specified primary and secondary datum.
- "External to material" means that the resulting point feature lies on the highest point on the input feature (or features), subject to the orientation constraint.

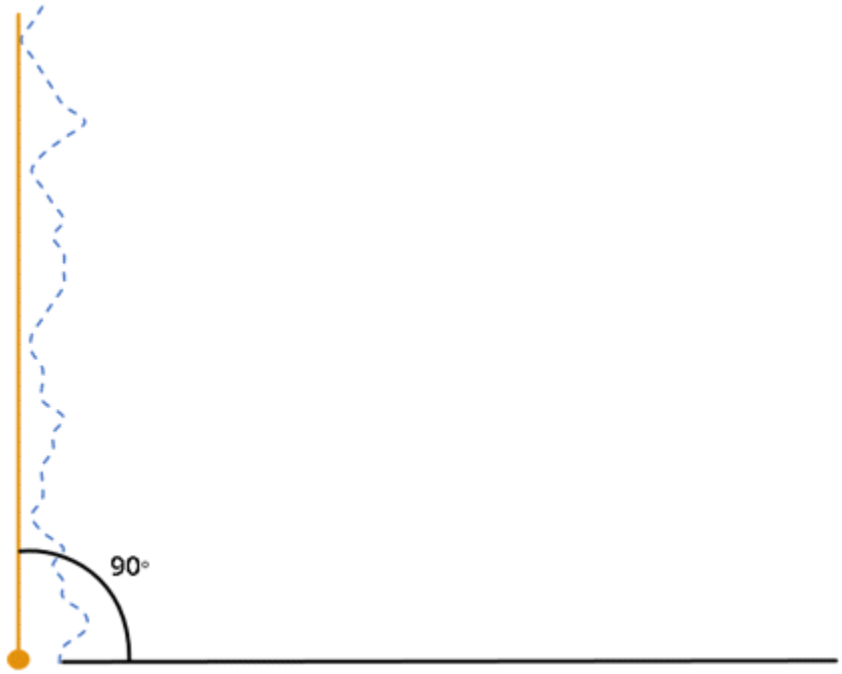
PC-DMIS constructs the tertiary datum point at the highest point along a vector that is oriented perpendicular to the specified primary datum, and at the keyed-in angle from the specified secondary datum. PC-DMIS places the point on the secondary datum line in the primary datum plane to simplify use in the alignment command.

Examples

The examples below showcase two different angles. The first angle is orthogonal, and the second angle is non-orthogonal:

Case 1 : 90 Degrees (Orthogonal)

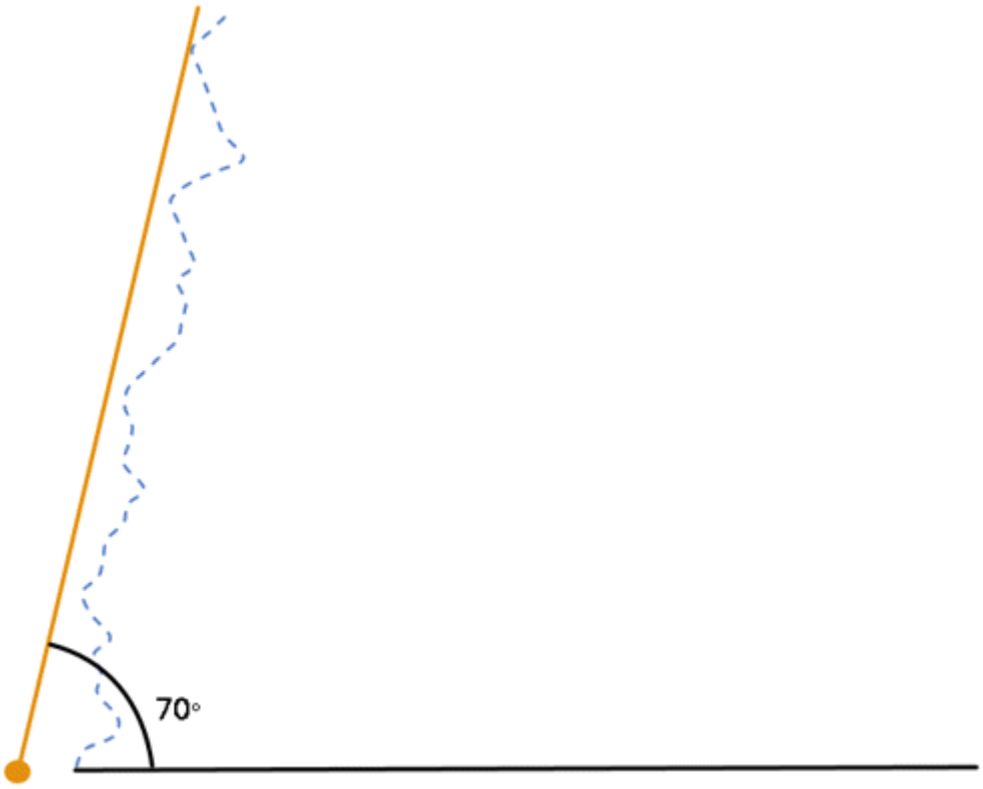
The yellow tertiary datum point below is constructed from the highest input point (where the yellow line contacts the blue dotted line) that is relative to the basic angle (90 degrees) from the specified secondary datum line in the primary datum plane's workplane.



Constructed Tertiary Datum Point feature from a 90-degree basic angle

Case 2 : 70 Degrees (Non-Orthogonal)

Similar to Case 1, the yellow tertiary datum point is again constructed from the highest input point relative to the specified datums. This example shows how a non-orthogonal tertiary datum is created.



Constructed Tertiary Datum Point feature from a 70-degree basic angle

How to Construct a Tertiary Datum Point

1. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).
2. Select the **Tertiary Datum** option from the **Method** list.
3. From the list of features, select the input feature (or features). For valid input feature types, see the table in the "Constructing a Point Feature" topic. The selected features determine the initial **Secondary-tertiary basic angle** value.
4. From the **Primary datum** list, select the primary datum. The feature must either be a plane or workplane relative to the alignment.
5. From the **Secondary datum** list, select the secondary datum. The feature must be a line.
6. From the **Compensation type** area, from the second drop-down list, select a probe compensation option. The available options are **BF** for Best Fit or **BFRE** for Best Fit REcompensate.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

7. If you need a different basic angle, type it in the **Secondary-tertiary basic angle** box.
8. To change the feature theoreticals values, click the **Specify theos** check box in the **Feature Theoreticals** section and update the **X**, **Y**, **Z**, **I**, **J**, and **K** values as necessary.
9. Click the **Create** button to insert the constructed point feature into your measurement routine.

The Edit window command line for this option is:

```
CONSTR/POINT,TERTIARY_DATUM,feat_1,feat_2, …
MATH_TYPE/BFRE
PRIMARY DATUM PLANE/FEATURE,PLN1
SECONDARY DATUM PLANE/FEATURE,LIN1
SECONDARY TERTIARY BASIC ANGLE/<angle>
```

<angle> = This is the basic angle between the secondary and tertiary datums.

Constructing an Extracted Edge Point

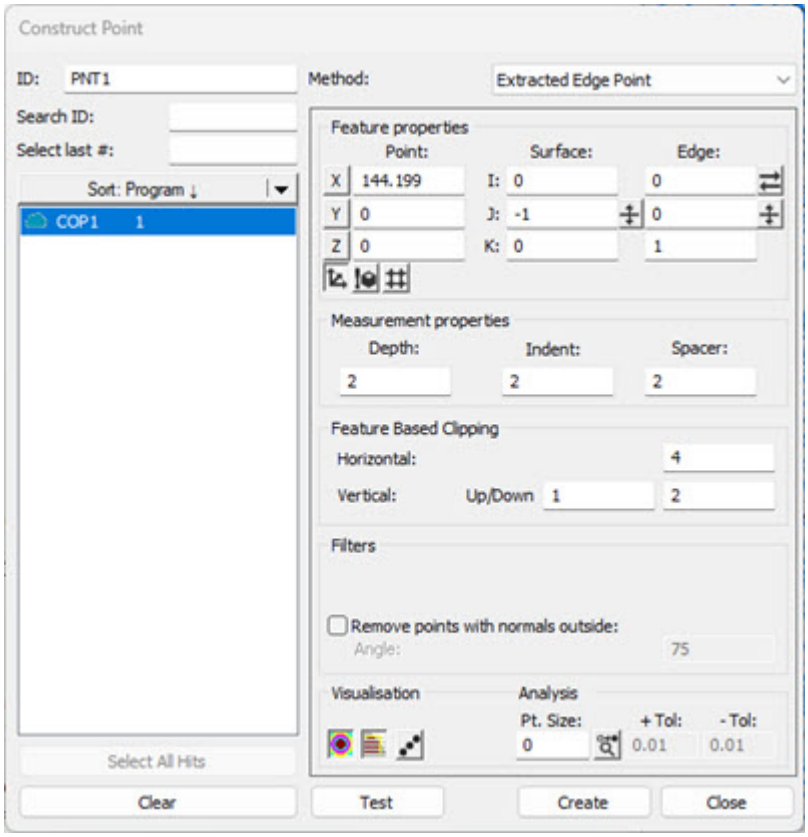
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct an edge point that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).



Construct Point Dialog Box - Extracted Edge Point Option

3. Select the **Extracted Edge Point** option from the **Method** list. This changes the **Construct Point** dialog box to show the options to create the extracted point.
4. From the **Reference** area, select the COP or Mesh.
5. Click on the CAD model or data to define the nominal (or from the **Feature properties** area, in the **X**, **Y**, and **Z** boxes, type the nominal location). A black line shows the current edge vector. The center of this line is the nominal location.

   PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the extraction zone that PC-DMIS uses for the edge point extraction. The yellow rectangle is the horizontal zone and the green rectangle is the vertical. The orange points are the candidate points that the extraction considers.
6. In the **I**, **J**, and **K** boxes, define the **Surface** and **Edge** vectors. The black line, the edge vector, needs to be in the same direction as your part's edge.

   You can use these controls to perform the associated functions:

Flip Vectors

Polar/Cartesian

Find Nearest CAD element

Snap to Grid

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation."

7. From the **Measurement properties** area, define the **Depth** at which you want to evaluate. Use the **Indent**, and **Spacer** values to determine the plane and the edge. For information on these properties, see the "The Dialog Box Options" sub-topic below.

8. From the **Feature Based Clipping** area, define the **Horizontal**, **Vertical Up/Down**, and **Vertical** values. This sets the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

9. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

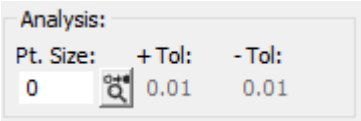
For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

10. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.



- The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the "Analysis Area" topic in the PC-DMIS Core documentation.

11. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

12. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
PNT1 =FEAT/POINT,CARTESIAN
   THEO/location,edge_vec,surf_vec
   ACTL/location2,edge_vec2,surf_vec2
   DEPTH=n1,INDENT=n2,SPACER=n3,
   HORIZONTAL CLIPPING=n4,VERTICAL CLIPPING=n5,
   REMOVE POINTS WITH NORMALS OUTSIDE=tog1,n6
   CONSTR/POINT,EXTRACTED_EDGE_POINT,ref_1
```

    location - This stands for the XYZ nominal location.

    edge_vec - This stands for the I, J, K edge vector.

    surf_vec - This stands for the I,J,K surface vector.

    location2, edge_vec2, surf_vec2 - These are the same as the items above but contain the actual measured values.

    n1 through **n6** - These numbers stand for the values for the different properties. **n6** is visible only when **tog1** is YES.

tog1 - This is either YES or NO based on the check box state.

ref_1 - This is the reference to the cloud of points (i.e. COP1).

For an example command, see the example below.

The Dialog Box Options

ID - Defines the extracted point feature's label.

Reference - Select the Pointcloud or Mesh object(s) from this list that PC-DMIS will use in the extraction.

Feature properties - This area defines the location for the point and the surface and edge vectors.

Point 1 - The **X**, **Y**, and **Z** boxes define the location of the point. PC-DMIS draws a black line for the edge vector in the Graphic Display window. The center of the black line is the point's location.

Surface - The **I**, **J**, and **K** boxes define the surface vector. PC-DMIS uses this vector along with the spacer value to calculate the surface. PC-DMIS draws the surface as a yellow plane in the Graphic Display window. When PC-DMIS creates the extracted point, it projects it to this surface.

Edge - The **I**, **J**, **K** boxes define the edge vector. PC-DMIS draws the edge vector as a black line in the Graphic Display window.

Measurement properties - This area defines properties that PC-DMIS uses to extract the point.

Depth - This defines the depth. PC-DMIS extracts the point from the considered points at this depth up to the edge. It then calculates the best point and projects it onto the plane. PC-DMIS shows an upside-down blue T to indicate the depth. One blue line extends down from the reference plane. Another blue line is drawn parallel to the black line at that depth. The **Depth** value must be within the **Vertical** value of the **Feature Based Clipping** area.

Indent - This defines how far from the edge to draw the center of the spacer. PC-DMIS shows a red line to indicate the indent distance.

Spacer - This defines the size of the circular area (a radius) that PC-DMIS uses to calculate the surface. PC-DMIS draws a purple circle to indicate the spacer.

Feature Based Clipping - This area defines the boundaries of the extraction zone. Consider part variability when you define the extraction zone.

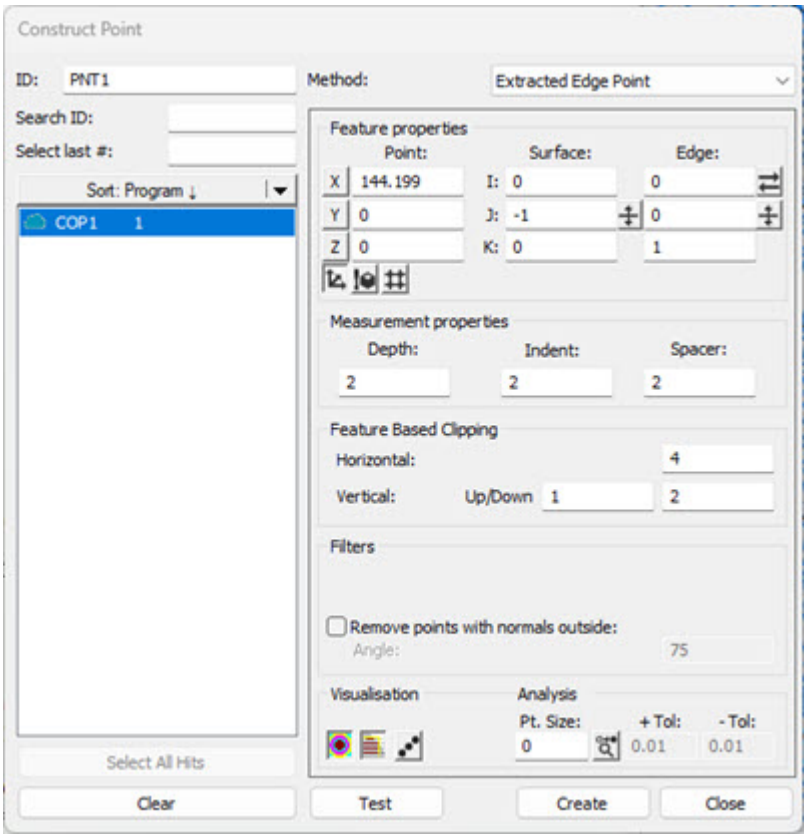
Horizontal - This defines the width of the extraction zone (yellow rectangle).

Vertical - This defines the height of the extraction zone (green rectangle). If you type a **Depth** value larger than this value, PC-DMIS increases this value to match the depth.

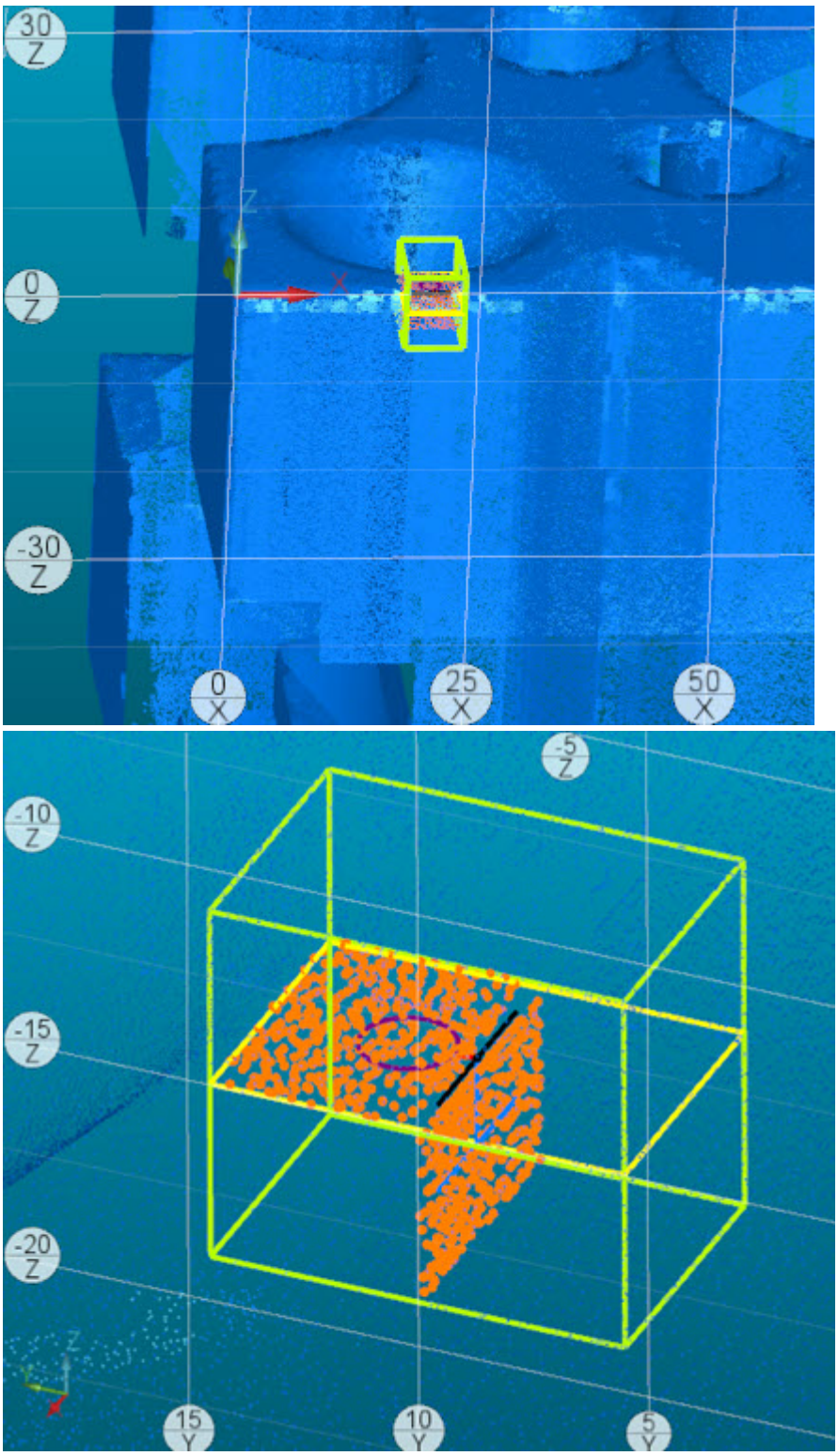
Filters - This area provides a check box and an angle box to filter out points beyond a defined incidence angle. This is useful if you have thin parts, like sheet a metal part, and the extraction zone may include scan data on both sides of the part. You can use the **Max incidence angle** box to define an angle. If you mark the check box, PC-DMIS compares the estimated normal of each scanned point to the feature theoretical normal. Points that are beyond that angle are removed from consideration.

An Example

Suppose that you have a pointcloud scan of the Hexagon demo block named COP1, and you have these values in the **Construct Point** dialog box:

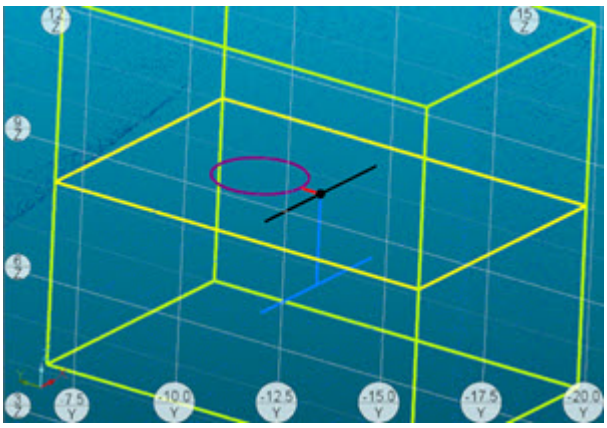


The result in the Graphic Display window while the dialog box is open, is this:



Example of a constructed edge point shown in a zoomed-out view (left) and a zoomed-in, rotated view (right).

Here's a view without the orange candidate points to provide a more visible example:



- The green box defines the extraction zone. The yellow plane defines the surface.
- The black line matches the edge vector.
- The center of the black line, the black dot, indicates the center location of the cube. This dot appears once you create the point. This is defined from the XYZ location.
- The red line shows the indent. In the above picture, you can see that it extends from the edge in the Y plus direction. This defines the spacer's distance from the edge.
- The purple ellipse shows the spacer.
- The blue line shows the depth.

When you click **Create**, PC-DMIS adds this point in the Edit window:

```
PNT1=FEAT/POINT,CARTESIAN
    THEO/<20,0,0>,<0,-1,0>,<0,0,1>
    ACTL/
<20,-0.064,0.054>,<0.0000197,-1,0.0002969>,<0.0042823,0.0002
97,0.9999908>
    DEPTH=2,INDENT=1.5,SPACER=1,
    HORIZONTAL
CLIPPING=2,VERTICAL CLIPPING=4,
    REMOVE POINTS
WITH NORMALS OUTSIDE=YES,0
            CONSTR/POINT,EXTRACTED_EDGE_POINT,COP1
```

Constructing an Extracted Surface Point

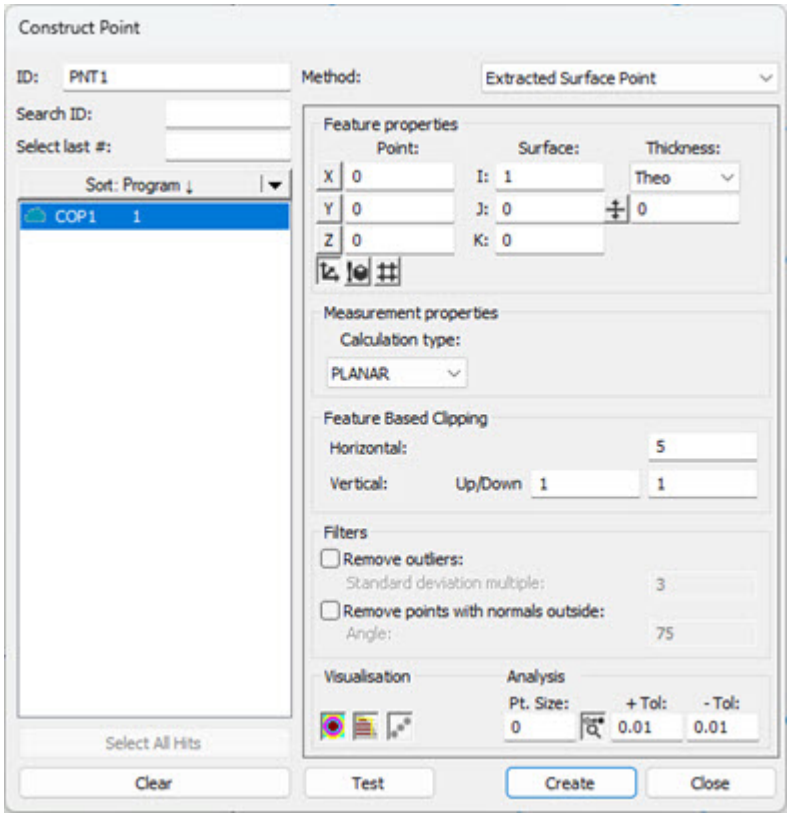
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a surface point that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Access the **Construct Point** dialog box (**Insert | Feature | Constructed | Point**).

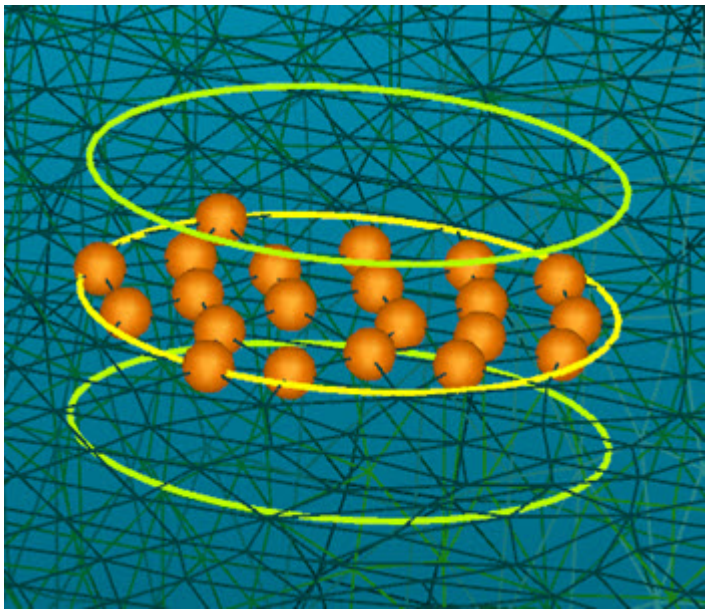


Construct Point Dialog Box - Extracted Surface Point Option

3. Select the **Extracted Surface Point** option from the **Method** list. This changes the **Construct Point** dialog box to show the options to create the extracted point.
4. From the **Reference** area, select the COP or Mesh.

5. Click on the CAD model or data to define the nominal (or from the **Feature properties** area, in the **X**, **Y**, and **Z** boxes, type the nominal location).

   PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the cylinder zone that PC-DMIS uses for the surface point extraction. The yellow cylinder is the surface. The yellow cylinder is also the horizontal zone and the green cylinder is the vertical zone. The orange points are the candidate points that the extraction considers.

   Example showing a constructed surface point with mesh displaying the candidate points.

   You can use these controls to perform the associated functions:

   Flip Vectors

   Polar/Cartesian

   Find Nearest CAD element

   Snap to Grid

   For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation."

6. From the **Measurement Properties** area, select the appropriate measurement method from the **Calculation type** list.

   The options are:

   - **PLANAR**
   - **SPHERICAL**

- **EXTENDED**

7. From the **Feature Based Clipping** area, define the **Horizontal**, **Vertical Up/Down**, and **Vertical** values. This sets the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

8. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

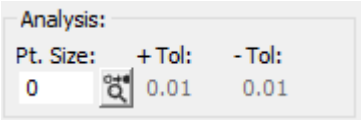
9. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

10. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.



- The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the "Analysis Area" topic in the PC-DMIS Core documentation.

11. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

12. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
PNT1 =FEAT/POINT,CARTESIAN,PLANAR
   THEO/<78.752,27.187,0>,<0,0,1>
   ACTL/
<78.752,27.187,0>,<-0.002493,-0.0020317,0.9999948>
   THEO_THICKNESS,0,
   HORIZONTAL CLIPPING=1,VERTICAL CLIPPING=2,
   USE OUTLIER REMOVAL=YES,0.1,
   REMOVE POINTS WITH NORMALS OUTSIDE=YES,5,
   CONSTR/POINT,EXTRACTED_SURFACE_POINT,REF=COP1
```

Constructing an Extreme Point

For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

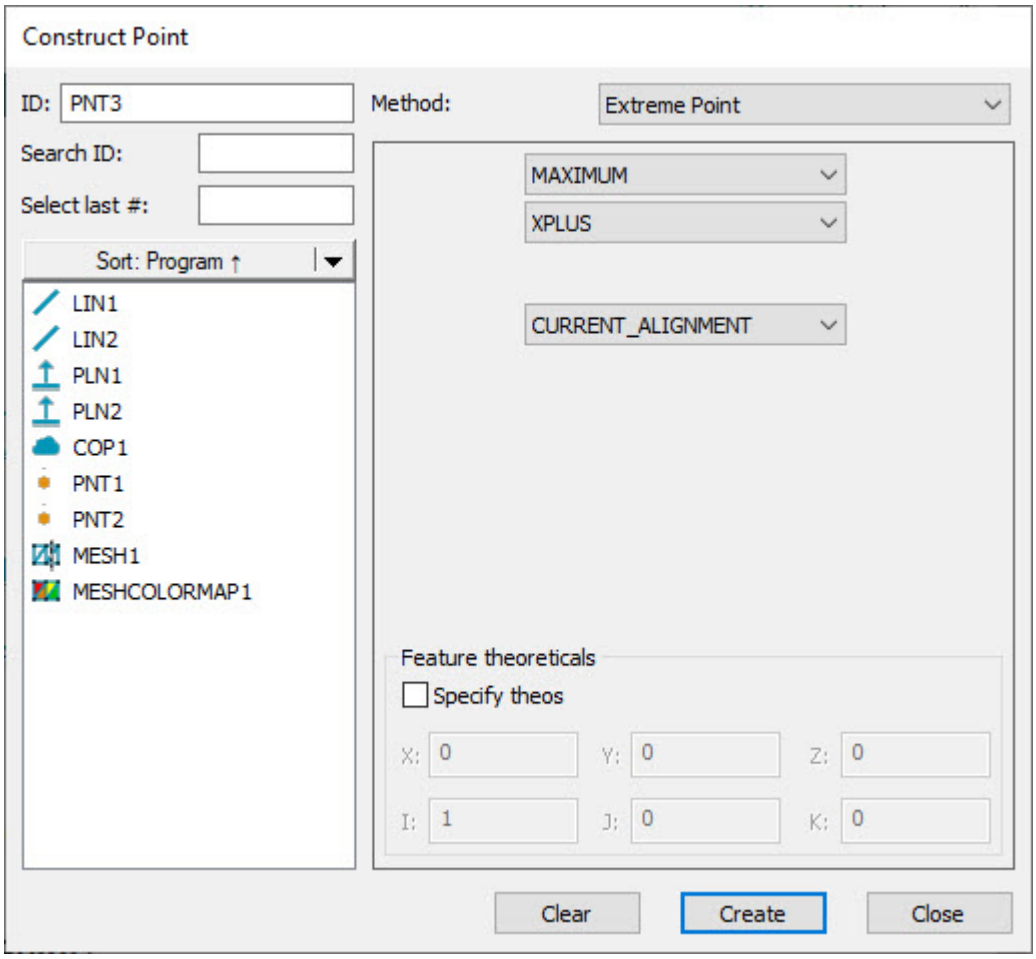
For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct an extreme point from a measured feature, scanned object, or from a scanned cloud of points (COP) or MESH.

To do this:

1. Ensure that your measurement routine has a measured feature, scanned object, or a Pointcloud (COP) or Mesh command.

2. Access the **Constructed Point** dialog box (**Insert | Feature | Constructed | Point**).



Construct Point Dialog Box - Extreme Point Option

3. Select the **Extreme Point** option from the **Method** list. This changes the **Constructed Point** dialog box to show the options to create an extreme point.

4. From the **Feature List** area, select the desired feature.

5. The **Options** area consists of three combo-boxes and the Feature theoreticals area. From the first combo-box, select either **MAXIMUM** or **MINIMUM**.

6. From the second combo-box, select the direction (**XPLUS**, **XMINUS**, **YPLUS**, **YMINUS**, **ZPLUS**, or **ZMINUS**) for the extreme point calculation. This is the direction that PC-DMIS uses to find the extreme point.

    - If you selected **MAXIMUM** in step 5, PC-DMIS defines the extreme point as:

        - If you selected **XPLUS**, **XMINUS**, **YPLUS**, **YMINUS**, **ZPLUS**, or **ZMINUS**, the largest value from the origin along the selected axis relative to the start point.

    - If you selected **2D_PR** or **3D_PR**, the longest polar radius from the origin relative to the start point.

- If you selected **MINIMUM** in step 5, PC-DMIS defines the extreme point as:

    - If you selected **XPLUS**, **XMINUS**, **YPLUS**, **YMINUS**, **ZPLUS**, or **ZMINUS**, the smallest value from the origin along the selected axis relative to the start point.

    - If you selected **2D_PR** or **3D_PR**, the shortest polar radius from the origin relative to the start point.

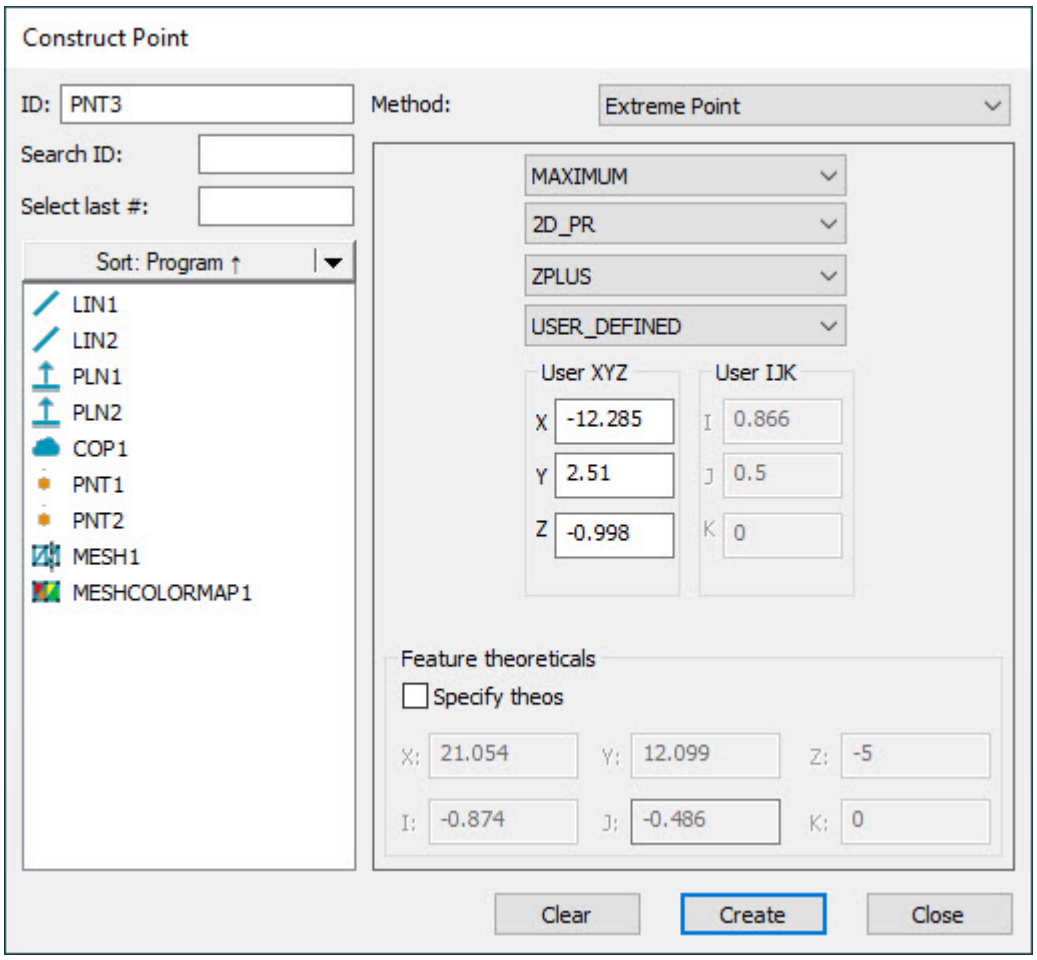
  If you select the **2D_PR** (2D-Polar Radius) option, PC-DMIS enables another combo-box just below the second combo-box. Use this to select the workplane (**ZPLUS**, **XPLUS**, **YPLUS**, **ZMINUS**, **XMINUS**, or **YMINUS**) in which to apply the 2D-Polar Radius.

  Select the **3D_PR** (3D-Polar Radius) option to define a 3D polar radius direction.

7. Use the third combo-box to define the start point that PC-DMIS uses to calculate the extreme point.

    - **CURRENT_ALIGNMENT** - Select this option to specify the origin (0, 0, 0) and orientation of the currently active alignment.

    - **USER_DEFINED** - Select this option to specify a new centroid starting point and starting axis vector. If you choose one of the polar radius calculations from the second combo-box (**2D_PR** or **3D_PR**), you can only specify the new centroid starting point.



When you select the **USER_DEFINED** option from the third combo-box, PC-DMIS creates a local alignment using the supplied X, Y, and Z values as the origin, the current workplane as the axis, and then rotates the primary axis using the supplied I, J, and K values. You can type the values into the **X**, **Y**, and **Z** boxes or click inside one of the boxes and then click a feature to use its centroid. Similarly, you can type the values into the **I**, **J**, and **K** boxes or click inside one of the boxes and then Alt + click a feature to use its direction.

8. From the **Feature theoreticals** area, you can select the **Specify theos** check box. Here you can set the theoretical **X**, **Y**, **Z** and **I**, **J**, **K** values for the constructed extreme point.

9. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS returns the calculated extreme point.

PC-DMIS created the command in the Edit window:

```
PNT1=FEAT/POINT,CARTESIAN,tog1
   THEO/<theoXyz>,<theoIjk>
   ACTL/<actXyz>,<actIjk>
   CONSTR/
POINT,EXTREME,feat1,tog2,tog3,optTog,tog4,udXyz,udIjk
```

Where:

tog1 – This is either `YES` or `NO` based on the values entered for the **Specify theos** option.

feat1 – This is the input feature selected in step 4 that PC-DMIS uses to find the extreme point.

tog2 – This is either **MAXIMUM** or **MINIMUM** selected in step 5.

tog3 – This is either **XPLUS**, **XMINUS**, **YPLUS**, **YMINUS**, **ZPLUS**, **ZMINUS**, **2D_PR** (2D-Polar Radius), or **3D_PR** (3D-Polar Radius). This specifies the direction to search for the extreme point.

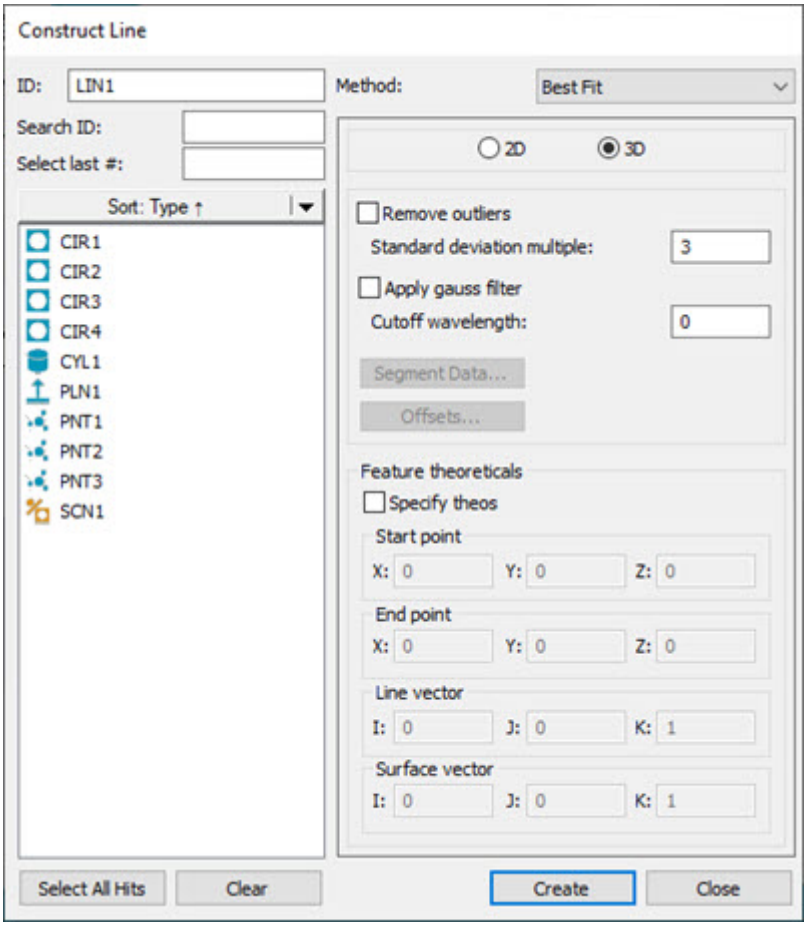
optTog – This is only available when **tog3** = **2D_PR**. This can be **ZPLUS**, **XPLUS**, **YPLUS**, **ZMINUS**, **XMINUS**, or **YMINUS**.

tog4 – This is either **CURRENT_ALIGNMENT** or **USER_DEFINED**. When you select **CURRENT_ALIGNMENT**, PC-DMIS calculates all extreme points from the origin (0, 0, 0). If you select **USER_DEFINED**, you can specify a new starting point and direction to calculate from. This is a way of creating a temporary alignment for the extreme point calculation.

udXyz – This is the user-defined X, Y, Z point that PC-DMIS uses to calculate the extreme point from.

udIjk – This is the user-defined I, J, K direction vector. This allows you to specify a new starting direction axis.

Constructing a Line Feature



Construct Line Dialog Box

There are many methods PC-DMIS uses to construct a line. The following table lists the various types of constructed lines along with their necessary inputs. Some features may require no inputs while others may require three or more inputs. In the table, the term "Any" indicates that the construction can take any type of feature as input for construction. PC-DMIS allows you to select the features in any order.

CONSTRUCT FEATURE TYPE	SYMBOL IN EDIT WINDOW	# OF INPUT FEATURES REQUIRED	FEATURE #1:	FEATURE #2:	COMMENTS
Auto Line	-	-	-	-	See "Auto Line Construction".
Alignment Line	ALIGN	0	-	-	Constructs a line through the alignment origin.
Best Fit Line	BF	At least 2 inputs are needed.	-	-	Constructs a best fit line using the inputs.
Best Fit with Recomp Line	BFRE	At least 2 inputs are needed. (1 must be a point)	-	-	Constructs a best fit line using the inputs.
Cast Line	CAST	1	Any	-	Constructs a line at the centroid of the input feature.
Intersect Line	INTOF	2	Plane	Plane	Constructs a line at the intersection of two planes.
Mid Line	MID	2	Line, Cone, Cylinder, Slot, Plane	Line, Cone, Cylinder, Slot, Plane	Constructs a mid line between the input features.
Offset Line	OFFSET	At least 2 inputs are needed.	Any	Any	Builds a line through the first feature and the offset from the second feature by a specified amount.

Parallel Line	PLTO	2	Any	Any	Constructs a line parallel to the first feature and passing through the second feature.
Perpendicular Line	PRTO	2	Any	Any	Constructs a line perpendicular to the first feature and passing through the second feature.
Project Line	PROJ	1 or 2	Any	Plane	Using one input feature will project the line to the workplane.
Reverse Line	REV	1	Line	-	Constructs a line passing through the input with a reversed vector.
Scan Segment Line	SCAN_SEGMENT	1	Scan	-	Constructs a line from a part of a Linear Open or Linear Close scan.
Secondary Datum Line	SECONDARY_DATUM	1	Point, Plane, Line, Set of Points	(Additional Point features if first feature is a point)	Constructs a line which simulates a secondary datum that is external to material.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

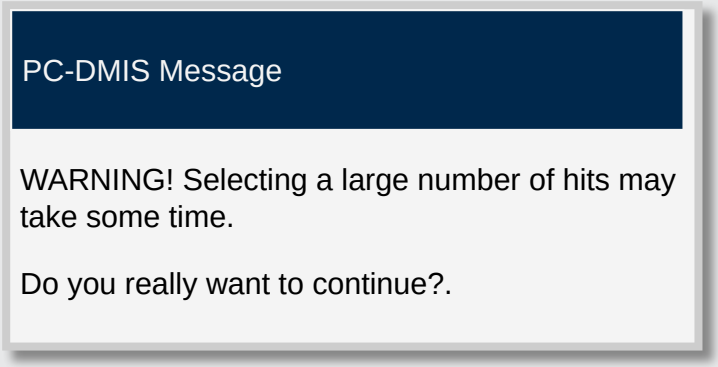
To construct a line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. Select the desired features from the **Feature** list.
3. From the **Method** list, select the method of the constructed line. The available options are:
    - Auto
    - Alignment
    - Best Fit
    - BF Recomp
    - Cast
    - Intersection
    - Mid
    - Parallel
    - Perpendicular
    - Projection
    - Reverse

- Scan Segment
- Offset
- Secondary Datum

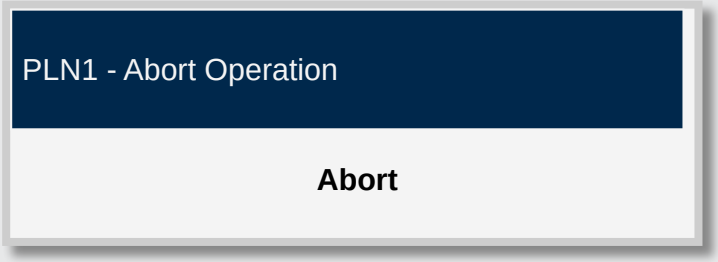
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

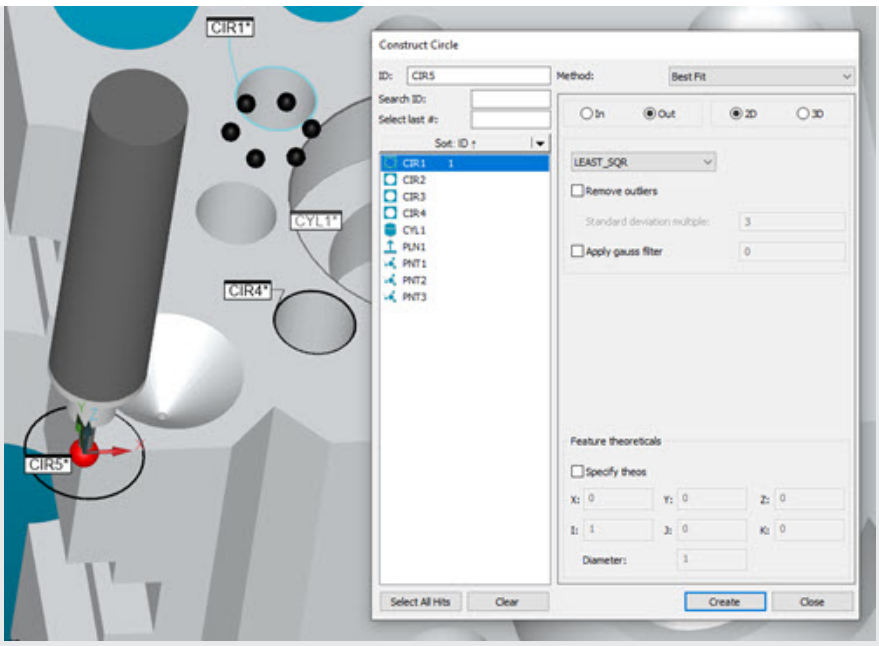


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

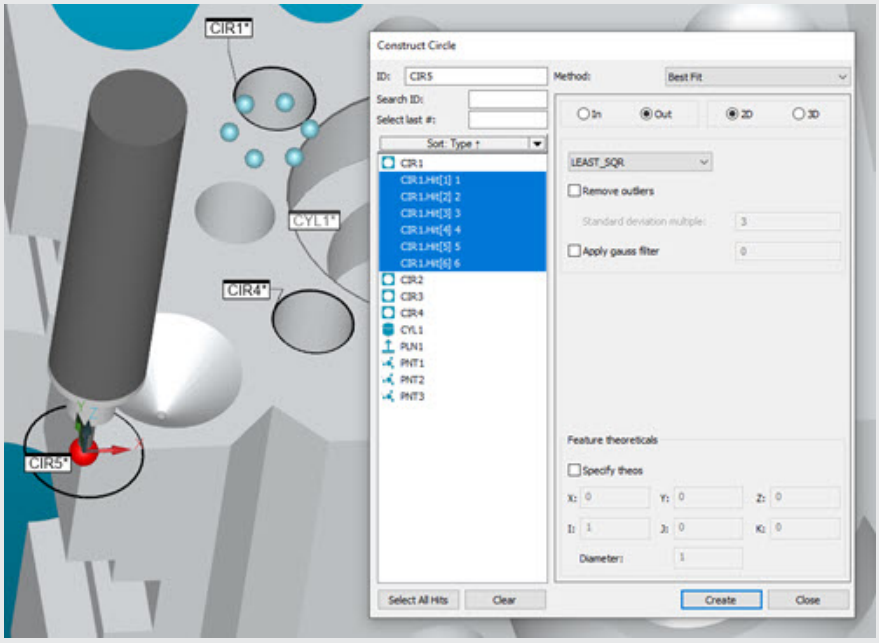
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

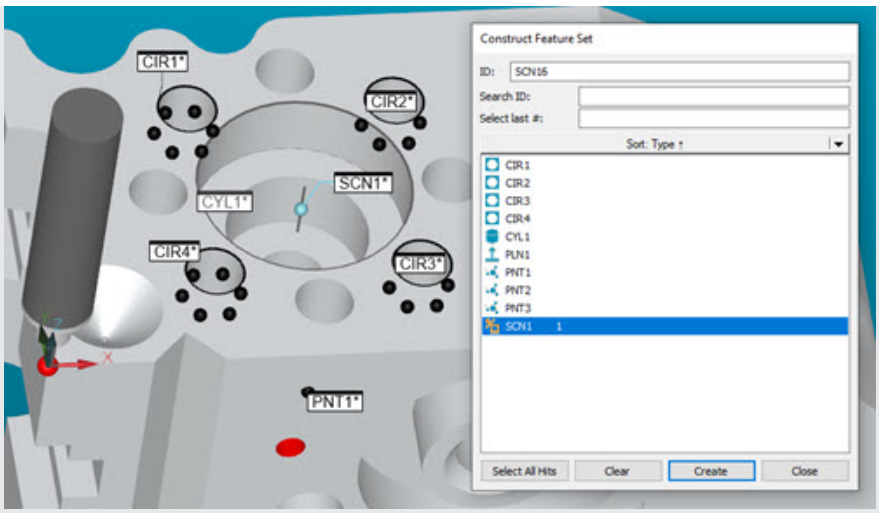
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

4. From the **Feature** list, use the table above to select the features based on the selected method.
5. Select either the **2D** or **3D** option.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for this option would read:

```
feature_name=FEAT/LINE,TOG1,TOG4
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length
CONSTR/TOG2,TOG3,...
```

If TOG2 = LINE and TOG3 = BF or BFRE then the command has the following format:

```
feature_name=FEAT/LINE,TOG1,TOG4
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length
CONSTR/LINE,TOG3
OUTLIER_REMOVAL/TOG5, stdDevMultiplier
FILTER/TOG5, WAVELENGTH=cutoffWavelength
```

The actual Edit report displays in all capital letters.

TOG1 = POLR or RECT

TOG2 = LINE

TOG3 = ALIGN / BF / BFRE / CAST / INTOF / MID / OFFSET / PLTO / PROJ / PRTO / REV / SCAN_SEGMENT / SECONDARY DATUM

TOG4 = BND / UNBND

TOG5 = OFF / ON

Length = This value represents the theoretical or actual length of the line.

stdDevMultiplier = This multiplier value determines whether or not a measured point is an outlier. If the point from the line is further than the standard deviation multiplied by this value, then it is an outlier and is removed if you selected the **Remove Outlier** option.

cutoffWavelength = This value controls the amount of data smoothing. The longer the wavelength, the more smoothing there is.

The first three lines that display in the Edit window are the same for constructed lines. The fourth line is slightly different, according to the type of feature you are constructing. To switch between the different types of lines, place the cursor on TOG3 and press F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

AUTO is the default method of construction. See "Auto Line Construction"

The following topics describe the available options for constructing a line:

Auto Line Construction

The list below indicates the type of line that the software constructs when you select certain input features along with the **Auto** option. The order of the selected features is not important. If you select an incorrect input feature or features, PC-DMIS shows an error message and will not construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Auto** option.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button.

Input Feature List

Input Feature(s)	Constructs
Three or more features	Best Fit Line
Any one feature (except line/set)	Cast Line
Any one set	Best Fit Line
Any two features + offset value	Offset Line
Circle + Circle	Best Fit Line
Circle + Ellipse	Best Fit Line
Circle + Point	Best Fit Line
Circle + Set	Best Fit Line
Circle + Slot	Best Fit Line

Circle + Sphere	Best Fit Line
Cone + Circle	Parallel to Line
Cone + Cone	Mid Line
Cone + Cylinder	Mid Line
Cone + Ellipse	Parallel to line
Cone + Point	Parallel to Line
Cone + Set	Parallel to Line
Cone + Sphere	Parallel to Line
Cylinder + Circle	Parallel to Line
Cylinder + Cylinder	Mid Line
Cylinder + Ellipse	Parallel to Line
Cylinder + Point	Parallel to Line
Cylinder + Set	Parallel to Line
Cylinder + Sphere	Parallel to Line
Ellipse + Ellipse	Best Fit Line
Ellipse + Set	Best Fit Line
Ellipse + Sphere	Best Fit Line
Line	Reverse Line
Line + Circle	Parallel to Line

Line + Cone	Mid Line
Line + Cylinder	Mid Line
Line + Ellipse	Parallel to Line
Line + Line	Mid Line
Line + Point	Parallel to Line
Line + Set	Parallel to Line
Line + Slot	Mid Line
Line + Sphere	Parallel to Line
Point + Ellipse	Best Fit Line
Point + Point	Best Fit Line
Point + Sphere	Best Fit Line
Point + Slot	Best Fit Line
Point + Set	Best Fit Line
Plane + any feature (except plane)	Project Line
Plane + Plane	Intersect Line
Slot + Cone	Mid Line
Slot + Cylinder	Mid Line
Slot + Ellipse	Parallel to Line
Slot + Slot	Best Fit Line

Constructing an Alignment Line

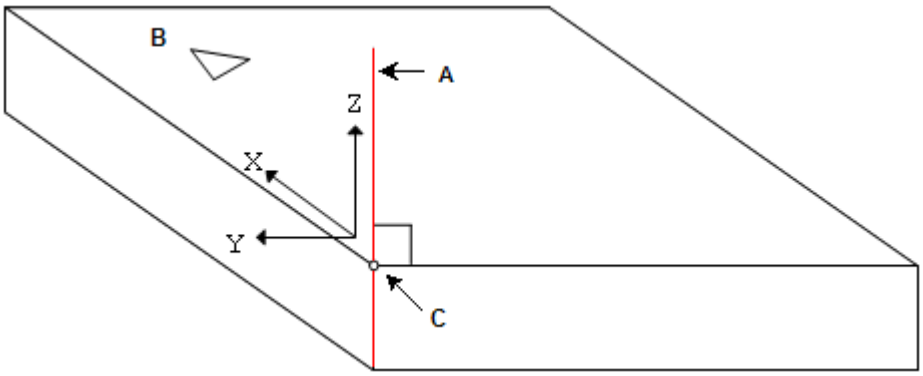
You can construct a line through the current origin, normal to the current workplane. (Input features do not need to be provided.)

To construct an alignment line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Alignment** option.
3. Do *not* select any features.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,ALIGN,length
```



Constructing a Line Normal to the Workplane

A. *Line through the current origin normal to the current workplane*

B. *Current workplane*

C. *Origin*

Constructing a Best Fit or Best Fit Recompensate Line

Best Fit Recompensate is only accurate when you use points to construct a line.

You can construct a "best fit" two or three-dimensional line from two or more features. The average squared error is minimized in the least squares method and the maximal error is minimized in the min/max method. You can also choose to remove outliers or apply a Gaussian filter to the constructed line.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

- The **Best fit** option compensates prior to fitting by using the ball center data shifted one tip radius in the direction of the negated hit vector.
- The **BF Recomp** option uses the ball center data and the tip compensation is part of the fitting process.

To construct a best fit or best fit recompensate line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. From the **Feature** list, select at least two features.
4. Select either the **2D** or **3D** option.
5. If desired, click the **Remove outliers** check box and specify a value in the **Standard deviation multiple** box.
6. If desired, click the **Apply gauss filter** check box and specify a value in the **Cuttoff wavelength** box.
7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click the **Create** button.

The Edit window command line for the **Best Fit** method would read:

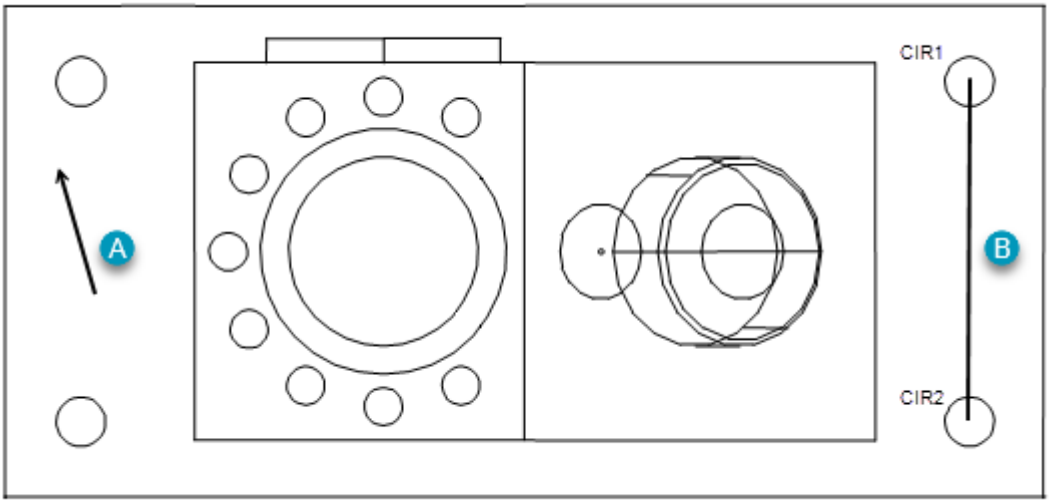
```
CONSTR/LINE,BF, feat_1,feat_2, …
OUTLIER_REMOVAL/(ON | OFF), stdDevMultiple
FILTER/(ON | OFF),WAVELENGTH=ctfoffWavelength
```

(This method uses the actual measured points for construction.)

The Edit window command line for the **Best Fit Recompensate** method would read:

```
CONSTR/LINE,BFRE,feat_1,feat_2, …
OUTLIER_REMOVAL/(ON | OFF), stdDevMultiple
FILTER/(ON | OFF),WAVELENGTH=cutoffWavelength
```

(This method uses the center of the probe for the measurement, and then recompensating after measuring the features.)



A *- Level*
B *- Line*
Constructing a Line from Two or More Features

Remove Outliers / Standard Deviation Multiple

For a best fit (BF) or best fit recompensate (BFRE) line, you can choose to remove outliers based on the distance from the best fit feature. This allows the removal of anomalies that arise in the measurement process.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

PC-DMIS first fits a line to the data and then determines which points are outliers based on the value in the **Standard Deviation Multiple** box. It then does the following:

- Recalculates the best fit line with those outliers removed.
- Checks for outliers again.
- Recalculates the best fit line.
- Continues to repeat this process until no more outliers exist or until PC-DMIS cannot compute the line. (PC-DMIS can't compute the line if there are fewer than three data points.)

For 2D lines, the deviation is calculated in a plane parallel to the workplane.

For 3D lines, the deviation is computed as the distance from the measured point to the nearest point along the line.

Apply Gauss Filter / Cutoff Wavelength

The Best Fit (BF) and Best Fit Recompensate (BFRE) constructed lines have the option of filtering the deviations of the measured data points from the best fit line computed from the measured data.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Apply Gauss Filter - If you select this check box, PC-DMIS applies a Gaussian filter which controls the smoothing by the cutoff wavelength. Generally, a longer cutoff wavelength produces smoother filtered data.

Remove Outliers - If you select this check box and you are filtering the data, it removes the outlier data prior to filtering.

For 2D lines, PC-DMIS filters the deviations in a plane parallel to the workplane.

For 3D lines, PC-DMIS filters the deviations in two planes perpendicular to each other, both planes containing the line. PC-DMIS filters these deviations in 3D.

Constructing a Cast Line

You can construct a line by changing any given feature into a line. PC-DMIS constructs the line at the centroid of the input feature.

You can modify the length of the line; this changes the line from DEPENDENT to INDEPENDENT. This means that when the line is executed, the length does not change based on the input feature, but is independent of the input feature, while the position and vector remain dependent on the input feature. This allows you to have control over the line length in cases where the input feature doesn’t really have a length, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field that you may change.

To change the length of a line:

1. Open the Edit window.
2. Click on the line feature.
3. Press the Tab key until the length field is selected.
4. Type in a new length.
5. Press the Tab key. PC-DMIS updates the length.

PC-DMIS uses this length value for all calculations instead of a default length.

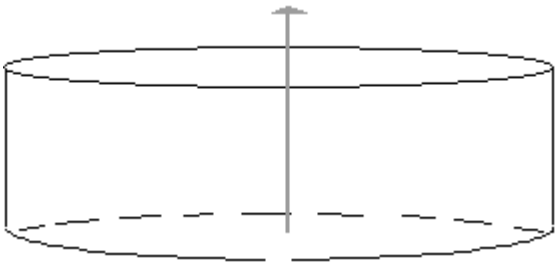
To construct a cast line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,CAST,feat_1,length,(DEPENDENT | INDEPENDENT)
```



Constructing a Line from a Cylinder

Constructing an Intersect Line

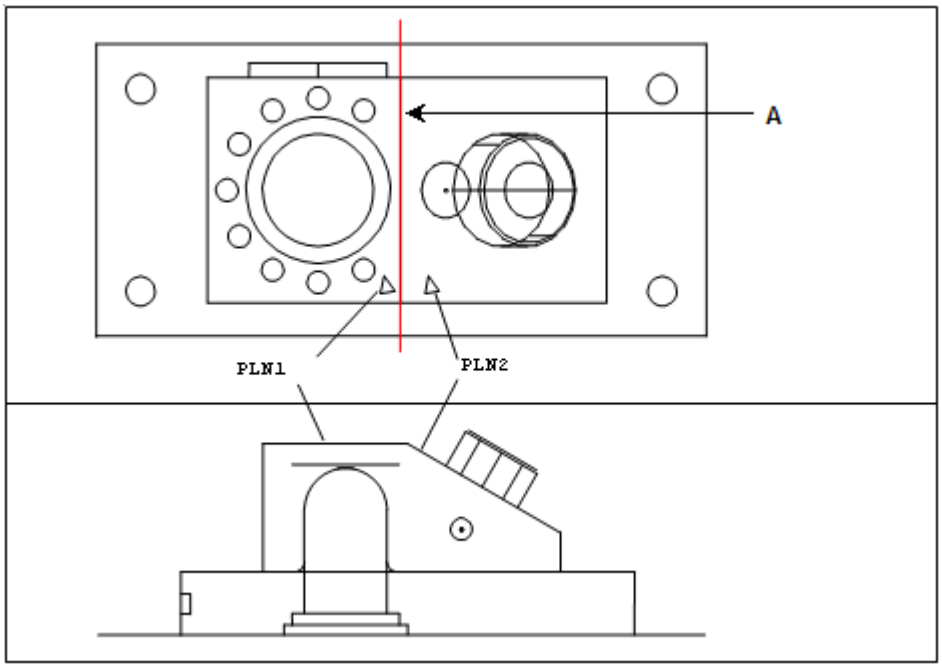
You can construct a line at the intersection of two planes.

To construct an intersect line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Intersection** option.
3. From the **Feature** list, select the first feature. It must be a plane.
4. From the **Feature** list, select the second feature. It must be a plane.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,INTOF,feat_1,feat_2,length
```



Constructing a Line from Two Planes

***A** - Line constructed at the intersection of two planes (PLN1 and PLN2)*

Constructing a Mid Line

You can construct a mid line between a line (cone, slot, cylinder, or plane) and a line (cone, slot, cylinder, or plane). PC-DMIS creates a line (mid line) such that each of its points is the same distance from both input lines. You can construct mid lines from lines that are parallel as well as lines with any angle between them. The lines do not have to intersect.

The mid line centroid is a point on the line segment between the centroids of the input lines that is the same distance from both lines (it is not necessarily the midpoint of the line segment).

The direction of the mid line vector depends on the location of the mid line centroid and on the two input vectors, with the sense of the mid line vector determined by the first line vector. Expressed mathematically, if the first line vector is V1 and the second line vector is V2 then the direction of the mid line vector is generally either V1 + V2 or V1 - V2.

To construct a mid line:

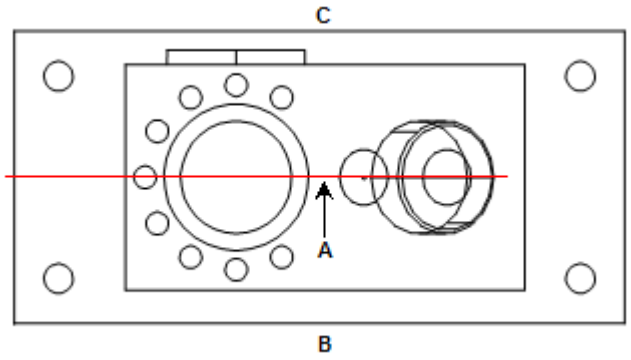
1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Mid** option.
3. From the **Feature** list, select the first feature. It must be either a line, cone, cylinder, or slot.
4. From the **Feature** list, select the second feature. It must be either a line, cone, cylinder, or slot.

5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,MID,feat_1,feat_2,length
```



A - Constructed line (a mid line) equally spaced between two lines (FRONT and BACK)
B - FRONT
C - BACK
Constructing a Mid Line from Two Lines

Constructing a Parallel Line

You can construct a line parallel to any two features. PC-DMIS creates a line parallel to the first input feature and passing through the center of the second input feature.

To construct a parallel line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).

2. From the **Method** list, select the **Parallel** option.

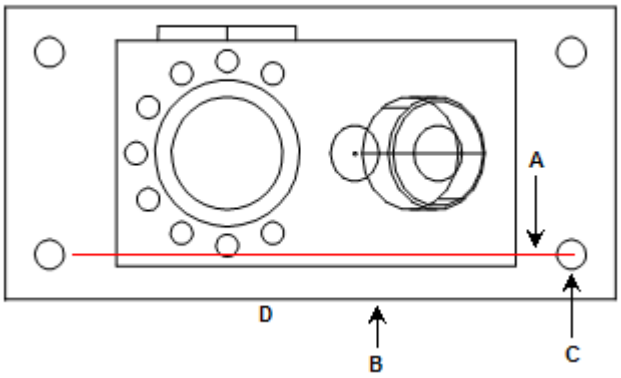
3. From the **Feature** list, select two features of any type.

4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,PLTO,feat_1,feat_2,length
```



Constructing a Parallel Line

***A** - Line constructed parallel to the first feature and through the second feature*
***B** - First feature*
***C** - Second feature*
***D** - FRONT*

Constructing a Perpendicular Line

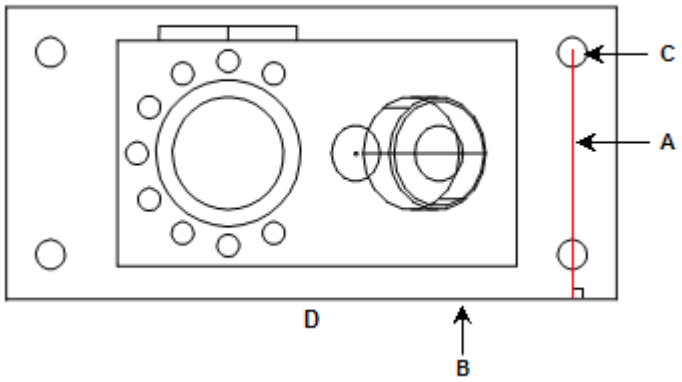
You can construct a line perpendicular to the first input feature and passing through the centroid of the second input feature.

To construct a perpendicular line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Perpendicular** option.
3. From the **Feature** list, select two features of any type.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,PRTO,feat_1,feat_2,length
```



Constructing a Perpendicular Line

***A** - Line constructed perpendicular to the first feature and through the center of the second feature*
***B** - First feature*
***C** - Second feature*
***D** - FRONT*

Constructing a Projected Line

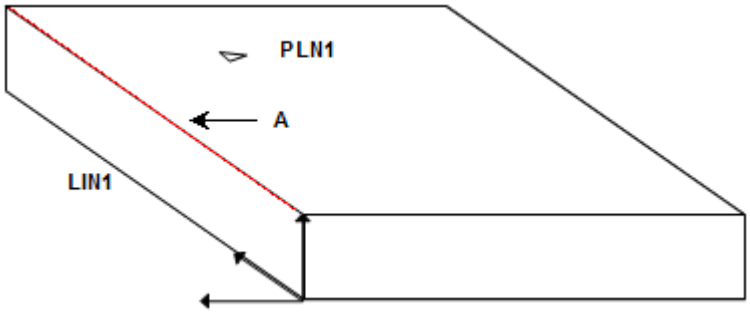
You can construct a line from any feature and a plane. PC-DMIS projects the line where the plane intersects the line. If there is only one input feature, the projection is to the workplane.

To construct a projected line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select either one or two features. The first feature may be of any type. If you select two features, the second feature *must* be a plane.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/LINE,PROJ,feat_1,(feat_2),length
```



Example of a Line feature constructed from a Line and a Plane.
***A** - Constructed line projected from a line feature (LIN1) onto a plane feature (PLN1)*

Changing the Direction of a Line

You can construct a line with a reversed vector.

To construct a reverse line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select one feature. It *must* be a line or an axial feature.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button. PC-DMIS reverses the vector of the line and also flips the start and end points.

The Edit window command line for this option would read:

```
CONSTR/LINE,REV,feat_1,length
```

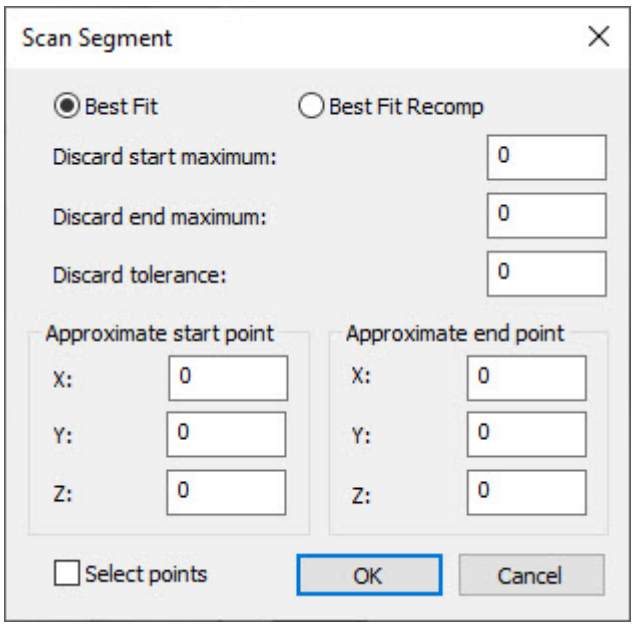
Constructing a Line from Part of a Scan

You can construct a line from a segment of a Linear Open or Linear Close scan. PC-DMIS creates a line from part of the scan. The details of the construction are included in this topic.

To construct a scan segment line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. Select the **Scan Segment** option.
3. Select a previously created Linear Open or Linear Close scan.

4. Click the **Segment Data** button to open the **Scan Segment** dialog box.



Scan Segment Dialog Box

5. Select either the **Best Fit** or **Best Fit Recomp** option.

6. Select the portion of the scan used in the construction from this dialog box.

7. Type the number of points to potentially discard by entering values in the **Discard start maximum** and **Discard end maximum** boxes.

8. Type a distance from the best fit line into the **Discard tolerance** box. This tolerance is a form tolerance; it allows you to control what end points are accepted as part of the line. If the distance from the scan point to the best fit line is greater than this tolerance value, then the end point is discarded.

9. Enter the X, Y, and Z values in the **Approximate start point** and **Approximate end point** areas. To do this, select the **Select points** check box and then click in the Graphic Display window to complete the **X**, **Y**, and **Z** boxes. You can click anywhere in the Graphic Display window; however, PC-DMIS places the point on the scan nearest to where you clicked. You can also type in the values.

10. Click **OK** to accept the data and close the **Scan Segment** dialog box.

11. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

12. Click **Create** to construct the line from the scan.

The Edit window command line for this option reads:

```
 CONSTR/
LINE,SCAN_SEGMENT,fit_type,feat_1,start_x,start_y,start_z,end_x,end_y,end_z,
```

If you want more than one arc or line from a given scan, then you need to add another command for a different section of the scan.

Determining the Data Used to Construct the Line

The data used to construct the line is determined as follows:

- First, a segment of the scan is determined using a starting point and an ending point within the scan. The starting and ending points are chosen as the point in the scan nearest `[start_x, start_y, start_z]` and the point nearest `[end_x, end_y, end_z]` , respectively.
- Points are then discarded from the start and end points of the scan. The number of points discarded is `discard_start` and `discard_end` from the start and end, respectively. A line is then fitted to this set of points.
- Finally, the starting and ending points are added back in if they are within the tolerance defined. The line is then refitted once more to the new set of points.

The value of `fit_type` may be `BF` (Best Fit) or `BFRE` (Best Fit Recompensate). This determines whether a best fit or best fit with recompensation is done when calculating the line. For a description of Best Fit and Best Fit Recompensate, see "Constructing a Best Fit or Best Fit Recompensate Line".

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

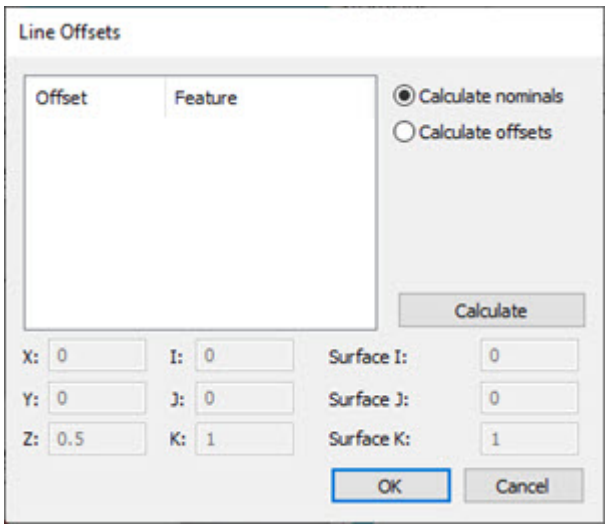
Constructing an Offset Line

You can construct an offset line between the input features and then offset by specified distances from the input features.

To construct an offset line:

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select the **Offset** option.
3. From the **Feature** list, select *at least* two features from which to create the offset line. They can be of any type.

4. Click the **Offsets** button to open the **Line Offsets** dialog box.



Line Offsets Dialog Box

5. Choose either to calculate the nominals from the offsets or calculate the offset value from the specified nominal values.

   - To calculate the nominals from the offset values, select the **Calculate nominals** option. In the **Offset** field, type the offset values for the desired features and then click **Calculate**.
   - To calculate the offsets from specified values, select the **Calculate offsets** option, modify the nominal values, and then click **Calculate**. For more information, see the procedures below.

6. Click the **OK** button to close the **Line Offsets** dialog box.
7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click the **Create** button to create the constructed offset line based on your values.

If the offset values exceed the distance between the input features, PC-DMIS cannot solve the offset line. Instead, it constructs a line without any offsets between the first and last feature.

The Edit window command line for this option would read:

```
CONSTR/LINE,OFFSET     SURFACE
NORMAL = i_vec, j_vec, k_vec, TOG1     ID = id1, id2, …
   OFFSET = val1, val2, …
```

TOG1 = changes between MULTI POINT and TWO POINT. For new constructions, you should use MULTI POINT.

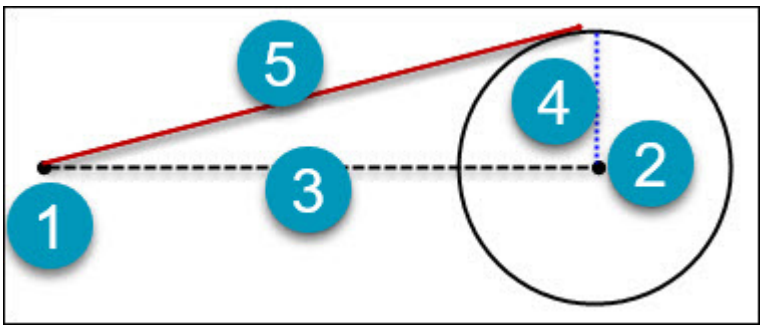
- **MULTI POINT** - This newer algorithm is the default algorithm used whenever you construct a new offset line. It constructs the offset line between the selected input features. You can specify offset values from any of the features.
- **TWO POINT** - This older algorithm is retained to support measurement routines from old PC-DMIS versions. It constructs the offset line between two input features. The offset value for the first feature must always be zero. Also, the way the positive or negative signs function is opposite how it works for MULTI POINT.

How It Works

Consider these statements along with the pictures below:

- PC-DMIS makes all offsets in a direction perpendicular to the given surface normal direction.
- PC-DMIS attempts to construct the line starting at the first input feature and ending at the last input feature.
- An offset value basically results in an imaginary circle drawn around each input feature's centroid by that value.
- PC-DMIS attempts to construct a line tangent to the imaginary circle around each input feature's offset radius. A tangent line touches the circle at only one point.
- In cases where more than two input features are used and they have varying offset values, PC-DMIS cannot construct a line tangent to all the offsets; instead, it tries to accommodate the offsets as best it can with a best fit line.
- PC-DMIS constructs a line so that the shortest distance from each input feature to the line is the corresponding offset amount.
- The offsets signs are determined by the direction of the line.

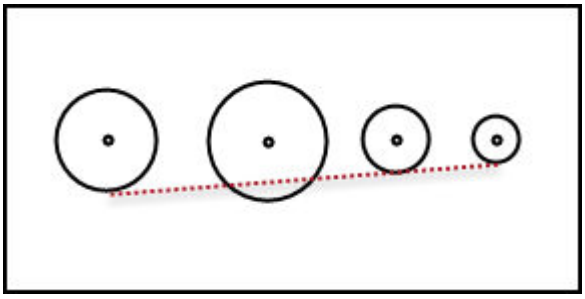
Suppose you have two points as input features, and the second point has an offset value:



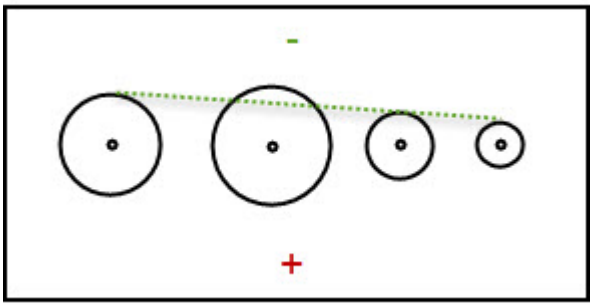
1. First point
2. Second point

3. Resulting line if no offset is defined
4. Offset distance (radius) from the feature's center that results in an imaginary circle around the feature
5. Constructed line from starting feature tangent to the imaginary circle

For example, suppose you have four points as your input features, and each point has a different offset value, as indicated by their circles. PC-DMIS constructs a best fit line that is tangent to those circles, as indicated in this diagram:



A positive or negative value on the offset determines on what side of the circle the line is constructed. For example, suppose you reverse the signs on all the offsets, from positive to negative, you might end up with something similar to this:

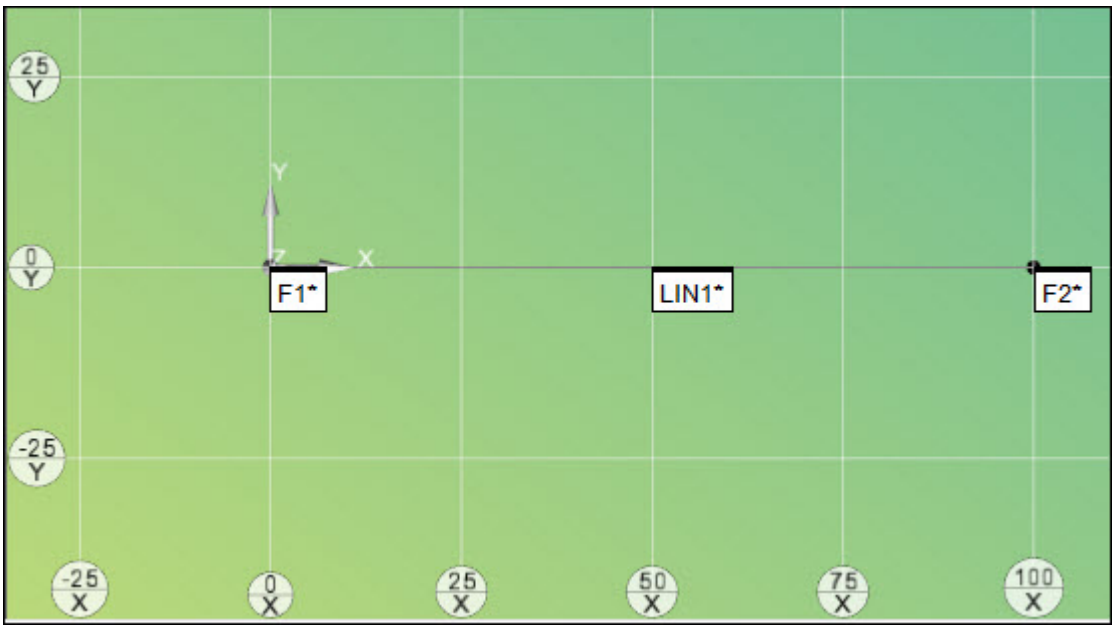


If you get a line opposite from what you expected, cancel the feature and reconstruct it, and try reversing the sign of the offsets values.

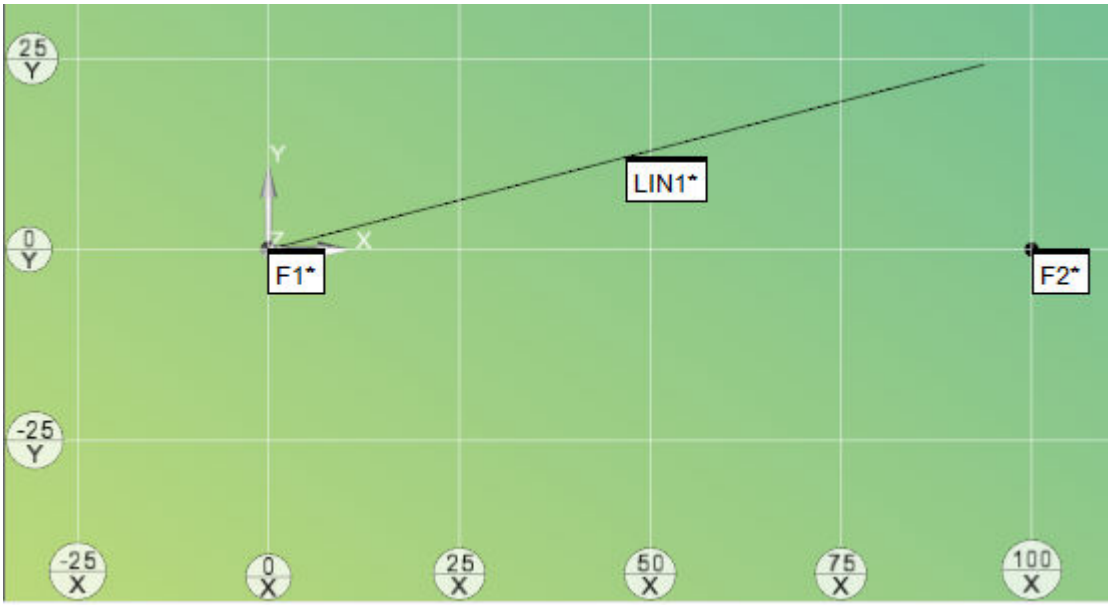
Examples

Here are some examples:

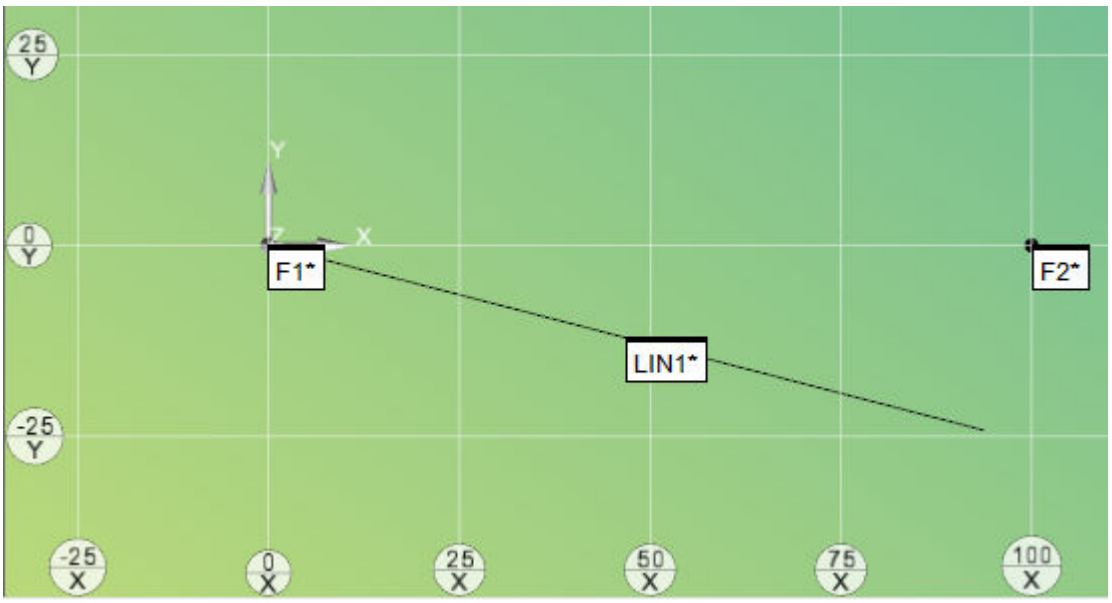
Example 1 - F1 and F2 are the inputs. Both have offsets that are set to zero:



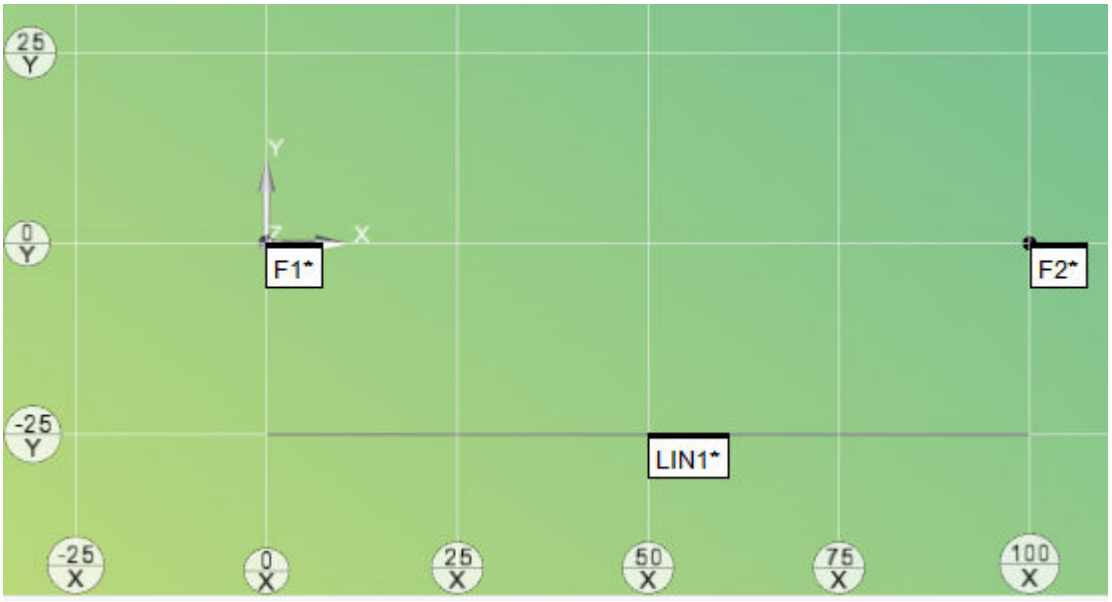
Example 2 - F1 and F2 are the inputs. The F1 offset is set to 0. The F2 offset is set to 25:



Example 3 - F1 and F2 are the inputs. The F1 offset is set to 0. The F2 offset is set to -25:



Example 4 - F1 and F2 are the inputs. Both offsets are set to -25:



Changing Offsets Directly to Calculate Nominals

To enter new offset values:

1. Click the **Offset** button from the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**) to open the **Line Offsets** dialog box.
2. From the **Line Offsets** dialog box, select the **Calculate nominals** option. The offsets portion of the dialog box becomes editable.
3. In the **Offset** column, click on a 0.000000 value to highlight it.
4. Click again on the offset so you can edit it.
5. Type a new value and press the Enter key to accept the value.

6. If desired, modify the other offset.
7. Click the **Calculate** button to update the nominal values based on your offset values.
8. Click **OK** to save the offset.

Example of Calculating Nominals

The **Calculate** button calculates the X, Y, and Z nominal values from new offsets entered.

For example, suppose you construct an offset line between two circles (CIR1 and CIR2). After you select the features and click the **Offsets** button, you have X, Y, and Z nominals of:

X = 4.5040

Y = 3

Z = 0.1582

If you change the offset values by two for each circle and click the **Calculate Noms** button, the X, Y, and Z update to:

X = 4.5040

Y = 5

Z = 0.1582

If you then click **OK** and construct the offset line, you'll notice that the newly constructed line lies two units higher in the Y axis.

Only the Y axis is offset for line offsets.

Changing Nominals Directly to Calculate Offsets

To enter new offset values:

1. Click the **Offset** button from the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**) to open the **Line Offsets** dialog box.
2. Select the **Calculate offsets** option. The nominals portion of the dialog box becomes available to edit.
3. Change the **XYZ**, **IJK**, or **Surface IJK** values.
4. Click the **Calculate** button to update the offset values based on any nominals you changed.
5. Click **OK** to save the offset.

Example of Calculating Offsets

The **Calculate Offsets** button calculates the offset values that appear in the **Offset** column when you change the nominal value.

For example, suppose you construct an offset line between two circles (CIR1 and CIR2). After you select the features and click the **Offsets** button, you have X, Y, and Z nominals of:

X = 4.5040

Y = 3

Z = 0.1582

If you change the X, Y, and Z nominals to:

X = 4.5040

Y = 4.5

Z = 0.1582

and click the **Calculate Offsets** button, the offsets for the two circles update to:

1.500000 CIR1

1.500000 CIR2

If you then click **OK** and construct the offset line, you'll notice that the line is constructed at 1.5 units higher in the Y axis.

Constructing a Secondary Datum Line

You can construct a Secondary Datum Line from a plane, line, or set of points. The set of points can be multiple selected points, a constructed set of points, or a scan feature that contains multiple points.

The secondary datum construction uses the points within the input feature (or features) as well as the plane from the primary datum input to create an "orientation constrained" and "external to material" 2D line. You can use this line in alignments to simulate a secondary datum feature.

- "Orientation constrained" means that the input points are forced perpendicular to the primary datum.
- "External to material" means that the resulting line contacts the highest point or points from the input feature (or features), subject to the orientation constraint and the selected math type.

PC-DMIS constructs the secondary datum line in the plane that you select as the primary datum.

Examples

The examples below show the points as if viewed from the selected plane.

- Constrained MinMax is the default math type specified by the ISO 1101 standard.
- Constrained L2 is the default math type specified by the ASME Y14.5 standard.

L∞

The red line is the L∞ plane targeting the external envelope (constrained MinMax type in PC-DMIS)

L2

The green line is the constrained L2 plane targeting the external envelope (constrained L2 math type in PC-DMIS)

How to Construct a Secondary Datum Line

1. Open the **Construct Line** dialog box (**Insert | Feature | Constructed | Line**).
2. From the **Method** list, select **Secondary Datum**.
3. From the **Feature** list, select the input feature (or features). For valid input feature types, see the table in the "Constructing a Line Feature" topic.
4. From the **Primary datum** list, select the primary datum. This datum can be either a plane from the measurement routine, or a workplane from the active alignment.
5. From the **Math Type** area, select the math type (**CONSTRAINED_L2** or **CONSTRAINED_MINMAX**) from the first list.
6. From the **Math Type** area, select a probe compensation option (**BF** for best fit or **BFRE** for best fit recompensate) from the second list.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click the **Create** button to insert the constructed line feature into your measurement routine.

The Edit window command line for this option is:

```
CONSTR/LINE,SECONDARY_DATUM, feat_1,feat_2, …
MATH_TYPE/TOG1,TOG2
PRIMARY DATUM PLANE/TOG3,TOG4
```

TOG1 = CONSTRAINED_L2 or CONSTRAINED_MINMAX

TOG2 = BF or BFRE

TOG3 = FEATURE or WORKPLANE

TOG4 = This is the list of workplanes if TOG3 is a WORKPLANE, or it's the Feature ID if TOG3 is a plane FEATURE.

Variations

The example below shows the Edit window when you switch TOG4 to BND.

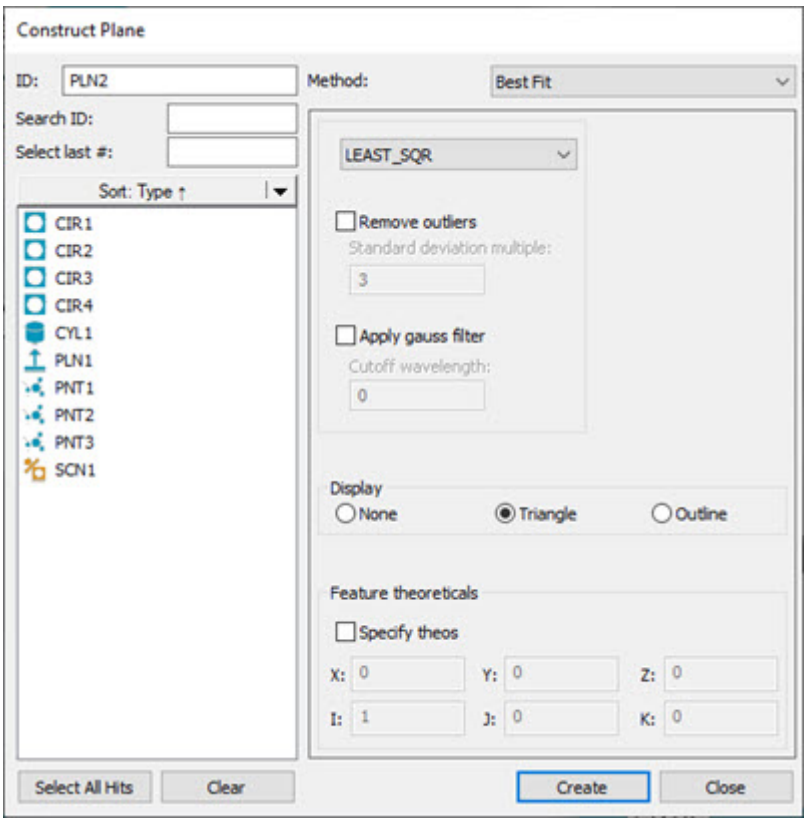
- The first x, y, z indicates the first point on the line.
- The second x, y, z indicates the ending point on the same line.
- The last value represents the theoretical or actual length of the line.

```
feature_name=FEAT/LINE,TOG1,BND
THEO/x_cord,y_cord,z_cord,x_cord,y_cord,z_cord,length
ACTL/x_cord,y_cord,z_cord,x_cord,y_cord,z_cord,length
CONSTR/TOG2,TOG3,...,length
```

The example below shows the unbounded line format:

```
feature_name=FEAT/LINE,TOG1,UNBND
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
CONSTR/TOG2,TOG3,...
```

Constructing a Plane Feature



Construct Plane Dialog Box

There are several methods to use PC-DMIS to construct a plane. The following table lists the various types of constructed planes along with their necessary inputs. Some features may require no inputs, while others may require three or more inputs. In the following table, the term "Any" indicates that the construction can take any type of feature as input for construction. PC-DMIS allows you to select the features in any order.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Tertiary Feature	Comments
Auto Plane	-	-	-	-	-	See "Auto Plane Construction".
Alignment Plane	ALIGN	0	-	-	-	Constructs a plane at alignment origin.
Best Fit Plane	BF	At least three inputs are needed.	-	-	-	Constructs best fit plane using the given inputs. See the Note below for recommended inputs.
Best Fit with Recomp Plane	BFRE	At least three inputs are needed (one must be a point).	-	-	-	Constructs a best fit plane using the given inputs. See the Note below for recommended inputs.
Cast Plane	CAST	1	Any	-	-	Constructs a plane at the centroid of the input feature.
Primary Datum Plane	PRIMARY DATUM	3 or 1	Any (one of three) Or if only one input: one feature set, one plane, or one scan	Any (2 of 3)	(Any 3 of 3)	Constructs a plane tangent to the highest available points.
Mid Plane	MID	2	Any	Any	-	Constructs a mid plane between the centroids of the inputs.

Offset Plane	OFFSET	3 or 1	Any (one of three) Or plane feature if only one input	Any (2 of 3)	Any (3 of 3)	Constructs a plane offset from the input features (or from single plane feature).
Parallel Plane	PLTO	2	Any	Any	-	Constructs a plane parallel to the first feature and passing through the second feature.
Perpendicular Plane	PRTO	2	Line or Axis Element	Any	-	Constructs a plane perpendicular to the first feature and passing through the second feature.
Reverse Plane	REV	1	Plane	-	-	Constructs a plane passing through the input with a reversed vector.
Translated Plane	TRANSLATED	1	Plane with point data	-	-	Constructs a plane at an offset from the input plane.
Extracted Plane	EXTRACTED_PLANE	1	COP or Mesh	-	-	Constructs an extracted plane from the COP or Mesh object at the specified width and length properties.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

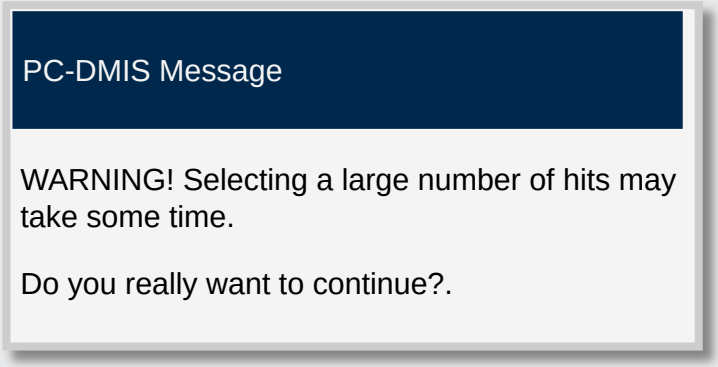
To construct a plane, do the following:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. Use the **Method** list to select the method type for the constructed plane. The available options are:
    - Auto Plane
    - Alignment Plane
    - Best Fit Plane
    - Best Fit Recompensate Plane
    - Cast Plane
    - Mid Plane
    - Perpendicular Plane
    - Parallel Plane
    - Reverse Plane
    - Primary Datum Plane
    - Offset Plane

- Translated Plane
- Extracted Plane

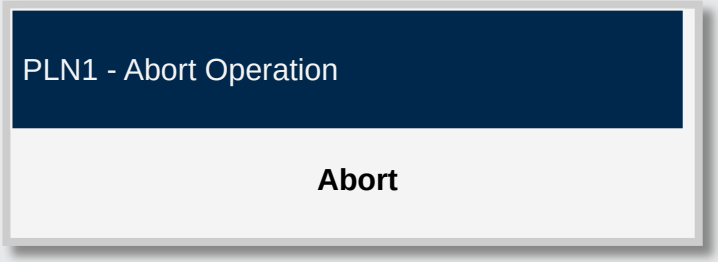
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

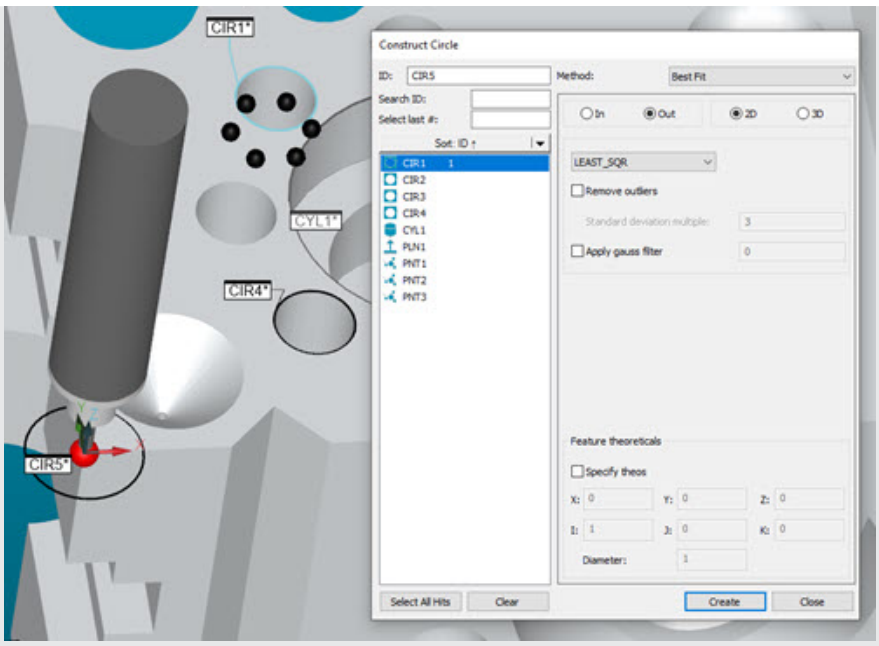


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

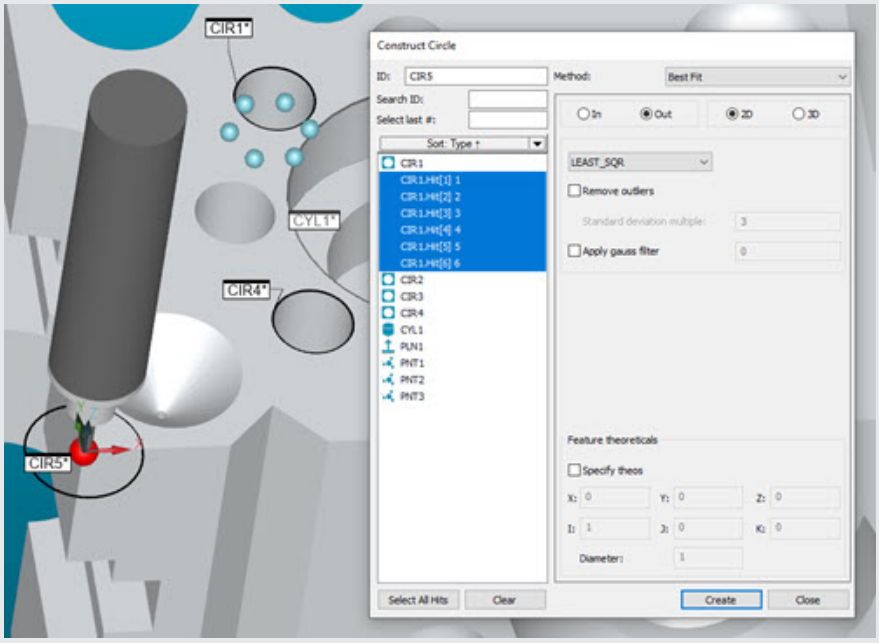
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

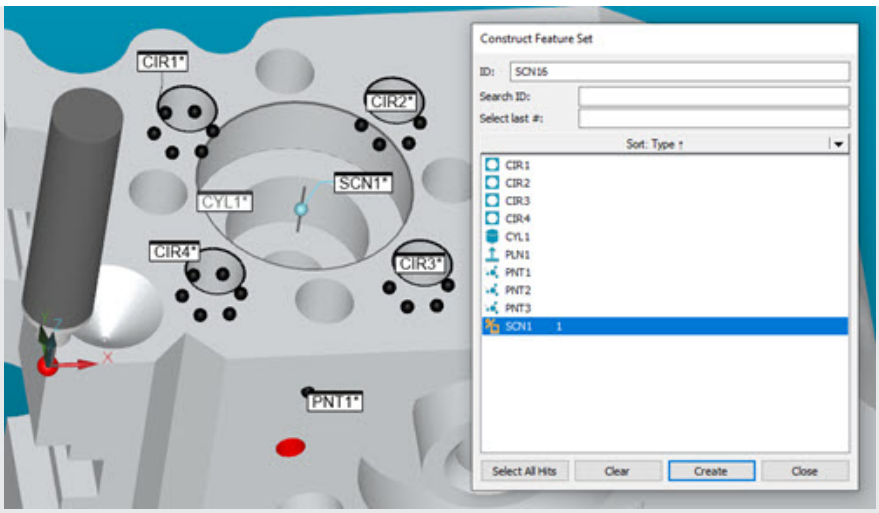
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. Use the table above to select the appropriate features from the **Feature** list for the constructed plane based on the selected method.
4. Make the necessary changes to other options that become available based on the selected method for the constructed plane. These are described in the topic specific to the method you selected.
5. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for a sample plane construction would read:

```
feature_name=FEAT/PLANE,TOG
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
CONSTR/TOG2,TOG3,...
```

If TOG2 = PLANE and TOG3 = BF or BFRE, the command has the following format:

```
feature_name=FEAT/PLANE,TOG1,TOG6
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec
CONSTR/PLANE,TOG3
OUTLIER_REMOVAL/TOG5,stdDevMultiplier
FILTER/TOG5,WAVELENGTH= cutoffWavelength
```

The actual Edit report displays in all capital letters.

AUTO is the default method of construction. This option automatically determines the best way to construct a plane using the input feature(s). See "Auto Plane Construction".

TOG1= POLR or CARTESIAN

TOG2 = PLANE

TOG3 = ALIGN / BF / BFRE / CAST / TANGENT / MID / OFFSET / PLTO / PROJ / PRTO / REV / TRANSLATED

TOG5 = ON / OFF

TOG6 = LEAST_SQR / MIN_SEP

stdDevMultiplier = This option is only available for Best Fit and Best Fit Recompensate methods. It determines whether or not a measured point is an outlier. If the point from the plane is further than the standard deviation multiplied by this, then it is an outlier and will be removed if you have selected the **Remove Outlier** option.

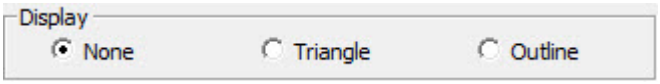
cutoffWavelength = This option is only available for Best Fit and Best Fit Recompensate methods. It controls the amount of data smoothing. The longer the wavelength, the more smoothing there is.

The first three lines that display in the Edit window are the same for constructed planes. The fourth line is slightly different, according to the type of feature you are constructing. To switch between the different types of planes, place the cursor on *TOG3* and press F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

The following subtopics describe the available options for constructing a plane:

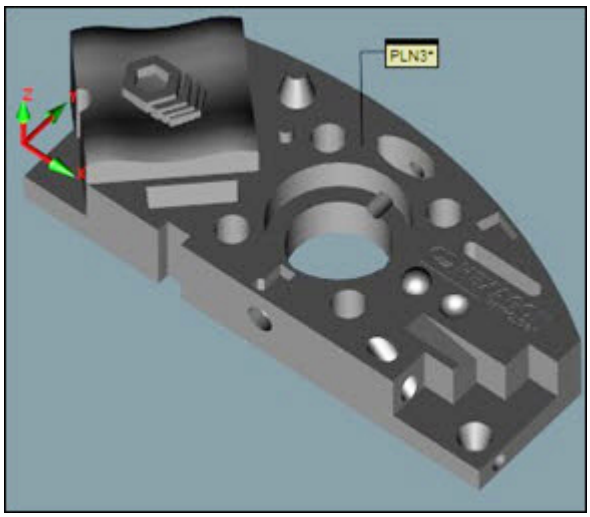
Using the Display Area



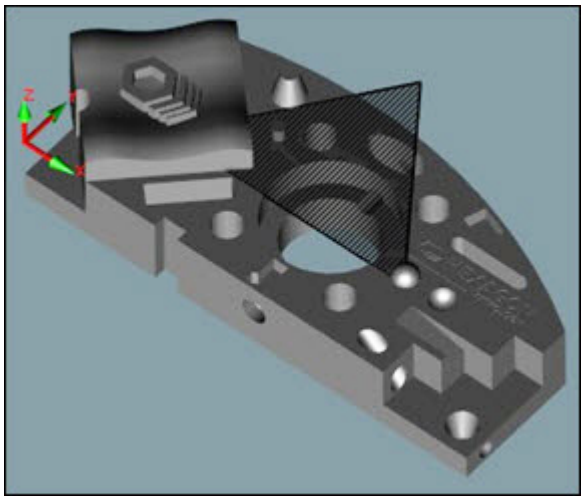
Display Area

This area defines how the plane feature draws in the Graphic Display window. It contains these options:

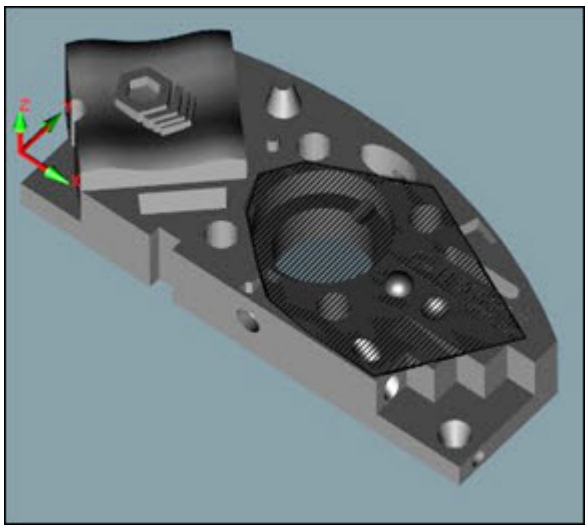
None - PC-DMIS only displays the ID of the constructed plane in the Graphic Display window. It does not display a drawing or outline of the constructed plane.



Triangle - PC-DMIS draws the constructed plane as a shaded triangle. The size of the constructed plane depends on the number of hits that make up the plane.



Outline - PC-DMIS draws the constructed plane as an outline from all the hits. The size depends on the number of hits that make up the constructed plane.



The Display Outline of Plane or Do Not Display Plane options on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**) define the default display state for future measured or constructed planes. They do not affect the display state of existing planes.

Auto Plane Construction

The "Input Feature List" table shows the type of plane you can construct when you select the specified inputs and select the **Auto** option. The order of the selected features is not important. If you select an incorrect input feature, PC-DMIS displays an error message and does not automatically construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Auto** option from the list of options.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

Input Feature List

Input Feature(s)	Constructs
Three or more features that are not all measured Points	Best Fit Plane
Any one feature (except Plane or Set)	Cast Plane
Any one Set	Best Fit Plane
All measured Points	Best Fit Recompensate Plane
One Plane	Reverse Plane
One COP or Mesh object	Extracted Plane

Constructing a Plane from an Alignment

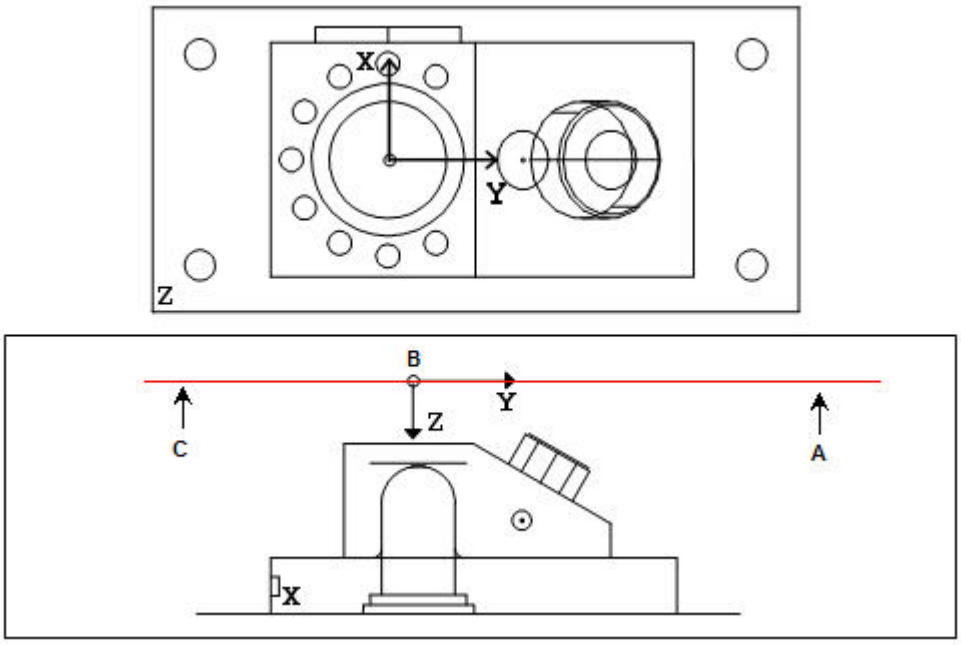
You can construct a plane through the current origin and parallel to the current workplane. (Input features do not need to be provided.)

To construct an alignment plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Alignment** option.
3. Do not select any features from the **Feature** list.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/PLANE,ALIGN
```



Constructing a Plane from an Alignment

A - Plane constructed parallel to the top (XY+Z) workplane and through the origin

B - Origin (0,0,0)

C - Current workplane

Constructing a Best Fit or Best Fit Recompensate Plane

You can construct a "best fit" plane through three or more features.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a best fit or a best fit recompensate constructed plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. From the **Feature** list, select at least three features.

4. Select the type of Best Fit construction from the **Best Fit Type** list. For information on the different types, see "Best Fit Type (for Plane)".

5. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.

6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

7. Click the **Create** button.

   If you chose **LEAST_SQR** as the best fit type, PC-DMIS computes a *least squares* plane, one for which PC-DMIS minimizes the average squared perpendicular distance from the data points to the plane.

   If you chose **MIN_SEP**, PC-DMIS creates a minimum separated plane.

For the Best Fit method, the Edit window command line for this option would read:

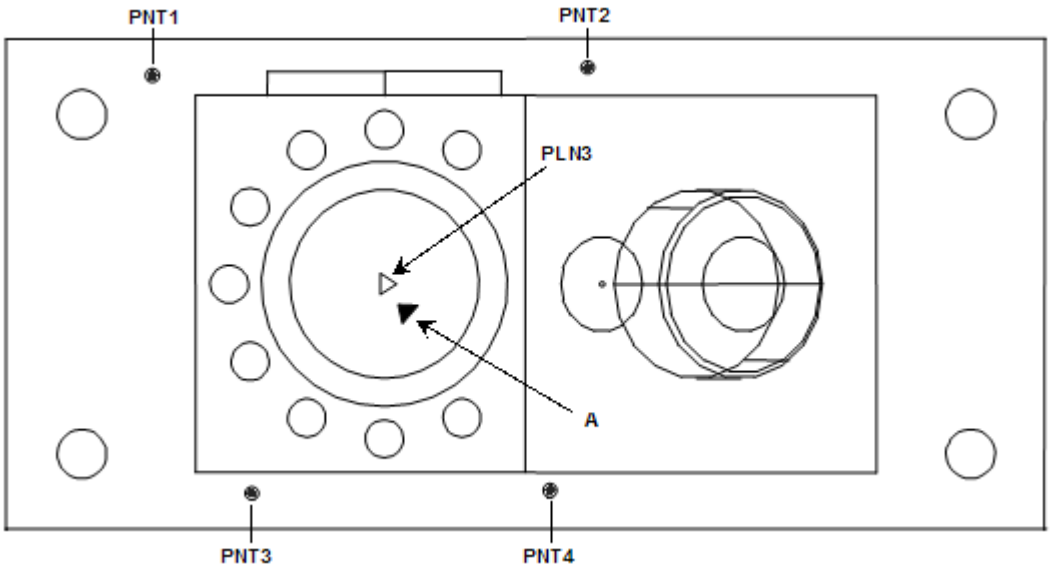
```
CONSTR/PLANE,BF, feat_1,feat_2, …

OUTLIER_REMOVAL/(ON | OFF), stdDevMultiple

FILTER/(ON | OFF), WAVELENGTH= cutoffWavelength
```

PC-DMIS uses the actual measured points for construction.

For the Best Fit Recompensate method, the Edit window command line for this option would read:

```
CONSTR/PLANE,BFRE, feat_1,feat_2, …

OUTLIER_REMOVAL/(ON | OFF), stdDevMultiple

FILTER/(ON | OFF), WAVELENGTH= cutoffWavelength
```

PC-DMIS uses the probe's center for measurement purposes.



A - Best Fit Plane Constructed from Four Points (PNT1, PNT2, PNT3, and PNT4)
Constructing a Plane from three or more Points

Remove Outliers / Standard Deviation Multiple

For a Best Fit (BF) or Best Fit Recompensate (BFRE) plane, you can choose to remove outliers based on the distance from the best fit feature. This allows the removal of anomalies that arise in the measurement process.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

PC-DMIS first fits a plane to the data and then determines which points are outliers based on the standard deviation multiple. It then does the following:

1. Recalculates the best fit plane with those outliers removed.
2. Checks for outliers again.
3. Recalculates the best fit plane.
4. Continues to repeat this process until no more outliers exist or until PC-DMIS cannot compute the plane. (PC-DMIS can't compute the plane if there are fewer than three data points.)

Best Fit Type (for Plane)

This list becomes available if you select the **Best Fit** or **BF Recomp** method for constructed planes. It allows you to specify the type of Best Fit construction used. Available types include:

- LEAST_SQR
- MIN_SEP

These types are discussed below:

LEAST_SQR
Least Squares - This calculation type provides a method of fitting in which the average squared radial distance from the data points to the plane is minimized. The square root of this quantity is the Root Mean Square (RMS) distance. Since the RMS distance is based on an average, some points may be further than the RMS distance from the computed plane.

MIN_SEP
Minimum Separation - This calculation type generates a plane from the input features containing the data points, with the difference of their radii as small as possible. The Min/Max math used by the MIN_SEP calculation minimizes the maximum error, or deviation, from the input data to the plane. The Min/Max error is one-half of the minimal separation. No input data points (or input features) lie farther than the Min/Max error from the Min/Max plane. This calculation determines whether or not all the input data (or input features) are within the given tolerances.

Apply Gauss Filter / Cutoff Wavelength

The Best Fit (BF) and Best Fit Recompensate (BFRE) constructed planes have the option of filtering the deviations of the measured data points from the best fit plane computed from the measured data.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Apply Gauss Filter - If you select this check box, PC-DMIS applies a Gaussian filter with a cutoff wavelength. Generally, a larger cutoff wavelength produces smoother filtered data.

Remove Outliers - If you select this check box and you are filtering the data, PC-DMIS removes outlier data prior to filtering. For additional information on Gaussian filtering, see the "Gaussian" topic under "Constructing a Filter Set".

Constructing a Cast Plane

A plane can be constructed by changing any feature into a plane. PC-DMIS constructs the plane at the centroid of the input feature.

To construct a cast plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/PLANE,CAST, feat_1
```

Constructing a Mid Plane

A plane can be constructed from any two features. The resultant plane (mid plane) is equally spaced from the centroids of the two specified input features.

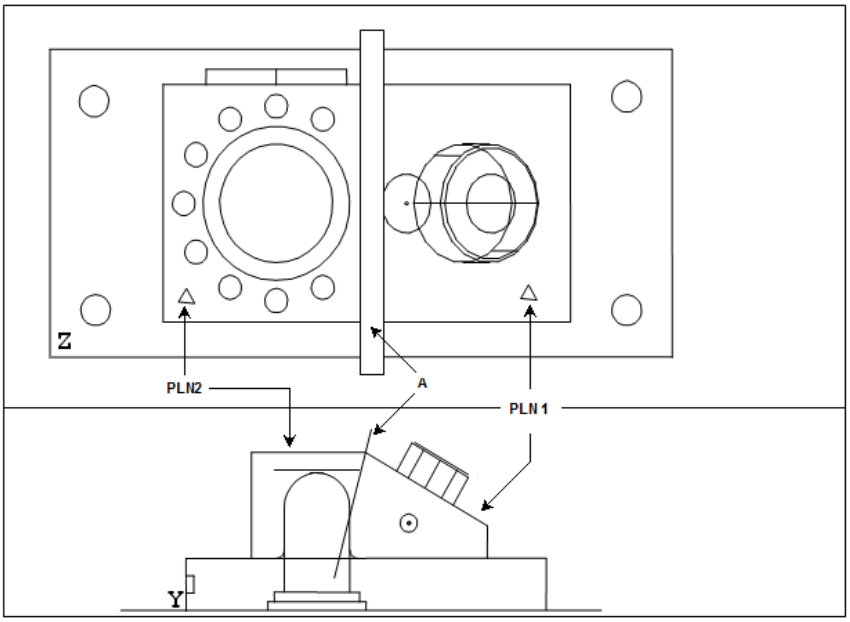
To construct a mid plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Midplane** option.
3. From the **Feature** list, select two features of any type.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/PLANE,MID, feat_1,feat_2
```



Constructing a Plane from Two Planes

A - Constructed plane equally spaced between two planes (PLN1 and PLN2).

The constructed plane and its vector depends on the input features used:

- *If you select planes for both input features*, then the mid plane is located where it bisects the smaller included angle between the two input planes.
- *If you don't select a plane for both input features*, then the constructed mid-plane passes through the point midway between the centroids of the two input features. The constructed plane's vector runs from the first input feature's centroid to the second input feature's centroid.

Constructing a Perpendicular Plane

A plane can be constructed between supported features. PC-DMIS creates a plane perpendicular to the first input feature and passing through the centroid of the second input feature.

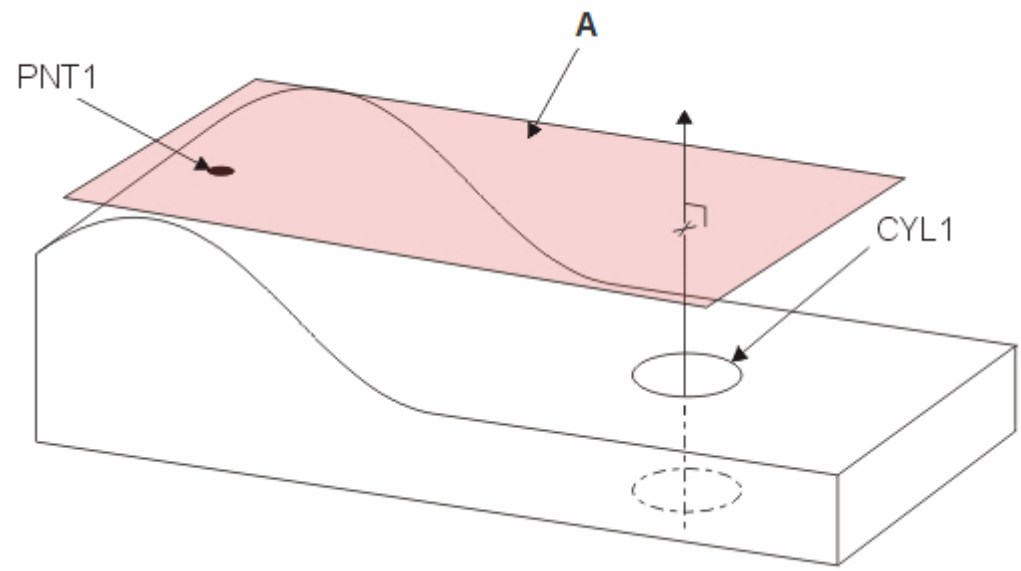
To construct a perpendicular plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Perpendicular** option.

3. From the **Feature** list, select the first feature type. This must be a plane, line, or axis element.
4. From the **Feature** list, select a second feature of any type.
5. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for this option reads:

```
CONSTR/PLANE,PRTO, feat_1,feat_2
```



Constructing a Perpendicular Plane

A - Plane constructed perpendicular to a cylinder feature (CYL1) and through a High Point feature (PNT1). Note that CYL1 contains an axis element.

By default, PC-DMIS calculates the vector of the constructed plane based on the vector from the first feature to the second feature. You may not always want this. The **Coplaner with line** check box lets you specify how PC-DMIS calculates the vector of the resulting plane. PC-DMIS enables this check box only when these conditions are met:

- The first feature is a plane-type feature (Plane, Circle, or Slot).
- The second feature is a line-type feature (Line, Cylinder, or Cone).
- The vectors of the two features are not parallel.

If you mark this check box, PC-DMIS constructs the resulting plane to be coplaner with the second feature.

Constructing a Parallel Plane

A plane can be constructed parallel to any two features. PC-DMIS creates a plane parallel to the first input feature and passing through the centroid of the second input feature.

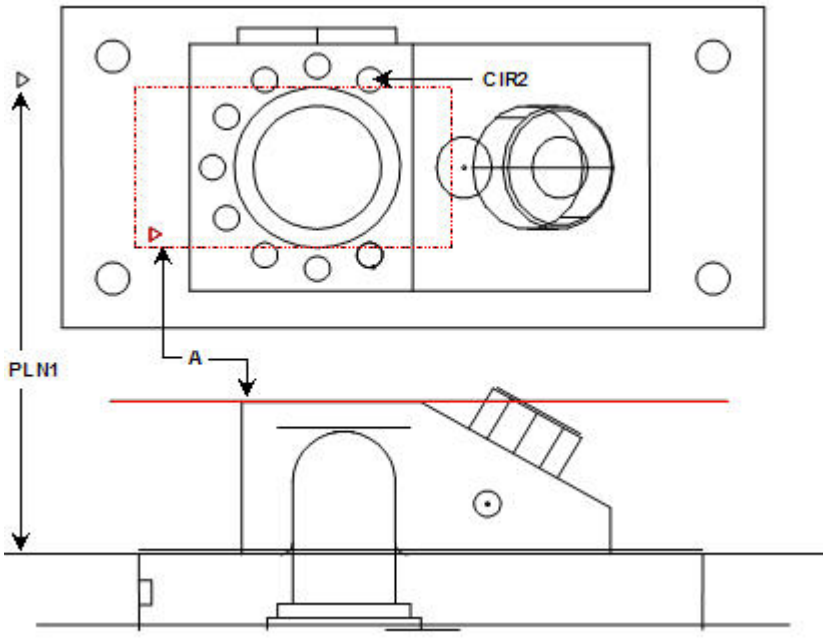
To construct a parallel plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Parallel** option.
3. From the **Feature** list, select two features of any type.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/PLANE,PLTO, feat_1,feat_2,
```

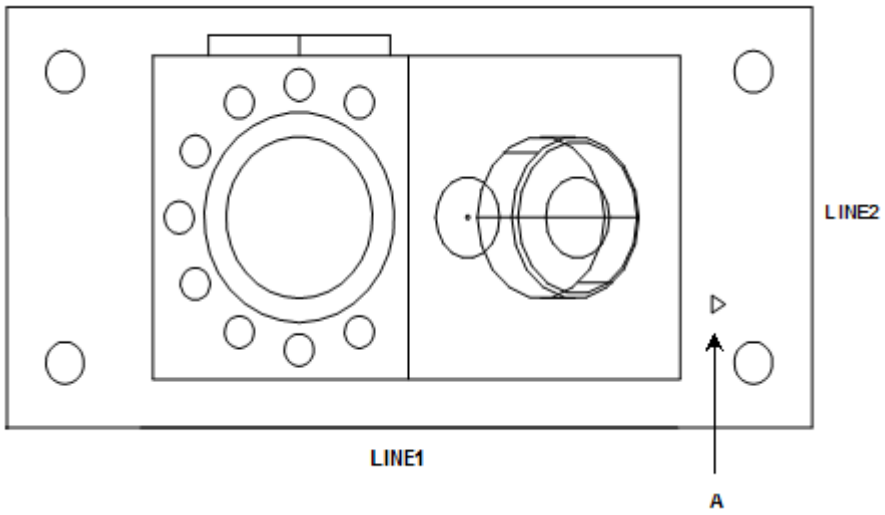
Example 1:



A - *Plane constructed parallel to the first feature, a plane (PLN1) and through the second feature, a circle (CIR2)*

Constructing a Parallel Plane Using Two Planes

Example 2:



A - *Plane constructed through the first feature, a line (LINE1) and parallel to a second line (LINE2)*
Constructing a Parallel Plane Using Two Lines

Changing the Direction of a Plane

A plane can be constructed with a reversed vector.

To construct a reverse plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select a Plane feature. It must be a plane.
4. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/PLANE,REV, feat_1
```

Constructing a Primary Datum Plane

Prior to version 2019 R1, the Primary Datum Plane was referred to as the "Tangent Plane" or "High Point Plane".

You can construct a plane that is external to the material (tangent to the peaks or the high points) on a surface. You can then use this feature as a datum in alignments or dimensions. Do not dimension a primary datum plane. That is not its purpose. You should use it only as a datum feature.

To construct a Primary Datum plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Primary Datum** option.
3. From the **Feature** list, select the input features from one of these options:
    - Three or more features (you may use any feature type).
    - Any single feature Set. For information on feature sets, see "Constructing a Set of Features".
    - Any Plane feature.
    - Any scan.
4. Select a math type from the first list in the **Math Type** area. The available options are:
    - **CONSTRAINED_L1**
    - **CONSTRAINED_L2**
    - **CONSTRAINED_MINMAX**

    For information on these math types, see the "Primary Datum Plane Math Types" topic in this documentation.
5. Select either **BF** (Best Fit) or **BFRE** (Best Fit Recompensate) from the second list in the **Math type** area.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

For information on Best Fit planes, see "Constructing a Best Fit or Best Fit Recompensate Plane".

6. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click the **Create** button.

After you click the **Create** button, PC-DMIS performs these steps to construct the Primary Datum Plane:

1. First, PC-DMIS constructs a Best Fit plane from the input features according to the math type.
2. Next, PC-DMIS rotates to the Best Fit plane.
3. Then, PC-DMIS finds the optimal primary datum plane from the highest points.

The Edit window command line for this option reads:

```
CONSTR/PLANE,PRIMARY DATUM, feat_1, feat_2, feat_3,…
```

Primary Datum Plane Math Types

The **Math Types** area of the constructed Primary Datum Plane provides three different math types. These math types support the ASME and ISO datum plane standards.

The three math types in the first drop-down list of the **Math Types** are:

- **CONSTRAINED_L1**
- **CONSTRAINED_L2**

- **CONSTRAINED_MINMAX**

Definitions

These terms correspond to the following definitions in the standards:

PC-DMIS	ASME Y14.5	ISO 5459
Constrained L1	Alternative Math Type	Alternative Math Type
Constrained L2 (default)	Default Math Type	Alternative Math Type
Constrained minmax	Alternative Math Type	Default Math Type

Before you choose a math type, you need to understand the concepts of void filtering (external envelope), norms, and outside-material constraint (constrained fit). These are covered below:

Void Filtering

Void filtering is also known as the external envelope.

In reality, surfaces of planar features are not planar. Surfaces have both convex and concave regions (peaks and valleys). The concave regions are guaranteed to never contact a perfectly flat surface plate. These are known as "voids". The software can filter out these voids by interpolating over them. Since the voids do not affect how the part interacts with an ideal surface plate, you may want to filter out the voids:

Illustration of a void-filtered surface. The dotted gray line is the real surface feature. The solid black line is the void filtered surface. The space between the real feature surface and the void filtered surface are the voids.

Norms

In general, ideal geometric shapes are fitted to non-ideal surfaces by minimizing the distances between the ideal shape and the non-ideal surface. In reality, there are an infinite number of distances between the two surfaces. Norms are mathematical concepts that transform these distances into a single distance while satisfying certain mathematical properties. The constructed PC-DMIS Primary Datum Plane supports three norms:

- **L1** - This is equal to the sum of the distances.
- **L2** - This is equal to the square-root of the sum of the squares of the distances. Minimizing the L2 norm is the same as a least-squares fit.

- **L∞** - This is equal to the maximum distances between the ideal surface and the non-ideal surface. Minimizing the L∞ norm is the same as minimizing the maximum deviation, so we use the term "minmax" for this norm.

The fitting process minimizes the distances according to the selected norm.

Outside-Material Constraint

Outside-material constraint is also known as a constrained fit. When fitting ideal geometric shapes to non-ideal surfaces, it is possible to add constraints to the fitting process. An outside material constraint means the fitting process is constrained while minimizing the norm. The constraint is that the ideal surface must lie outside the real surface. This is similar to a surface plate. A surface plate is always outside the part. For example, a constrained L∞ plane minimizes the maximum deviation out of all planes that lie external to the part.

Putting It All Together

The PC-DMIS Primary Datum Plane uses the concepts of void filtering, norms, and outside-material constraint together. From the provided measurements of the actual surface, the Primary Datum Plane first filters out the voids. It then finds the plane that minimizes the chosen norm of the distances to the void-filtered surface, subject to an outside material constraint.

Examples

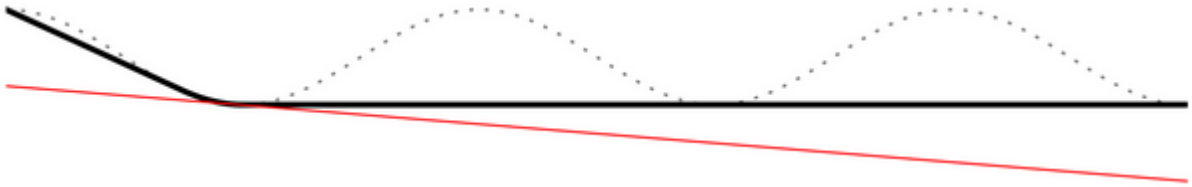
The three math types that the Primary Datum Plane exposes have different behaviors in different circumstances. These examples illustrate those circumstances. They also demonstrate how the L2 algorithm gives a repeatable approximation of a physical datum (such as a surface plate).

- L∞ usually gives a result controlled by the edges of the part. This makes it undesirable in most cases.
- L1 behaves somewhat like how a part would rest on a surface plate under the influence of gravity, but it does not handle rocker conditions gracefully. A rocker condition is when a part might rock on a surface plate instead of having obvious stable contact points. A convex datum plane results in a rocker condition.
- L2 performs much like L1 in many cases, but in rocker conditions, it always gives an equalized solution. This is the recommended norm for most applications.

Sinusoidal Surface Profile Examples

In each illustration, the dotted gray line is the real surface feature. The solid black line is the void-filtered surface (the external envelope).

L∞



The red line is the constrained L∞ plane targeting the external envelope (constrained minmax type in PC-DMIS). Observe how this plane is tilted with respect to how a surface plate would work.

L1

The blue line is the constrained L1 plane targeting the external envelope (constrained L1 math type in PC-DMIS).

L2

The green line is the constrained L2 plane targeting the external envelope (constrained L2 math type in PC-DMIS)

V-Like Surface Profile Examples

In each illustration, the solid black line is the void-filtered surface (the external envelope).

L∞

The red line is the L∞ plane targeting the external envelope (constrained minmax type in PC-DMIS). Observe how this plane equalizes the rocking condition.

L1

The blue line is the constrained L1 plane targeting the external envelope (constrained L1 math type in PC-DMIS).

L2

The green line is the constrained L2 plane targeting the external envelope (constrained L2 math type in PC-DMIS). Observe how this plane equalizes the rocking condition.

Wiggly Surface Profile Examples

In each illustration, the dotted gray line is the real surface feature. The solid black line is the void-filtered surface (the external envelope).

L∞

The red line is the L∞ plane targeting the external envelope (constrained minmax type in PC-DMIS). Observe how this plane is tilted with respect to how a surface plate would work.

L1

The blue line is the constrained L1 plane targeting the external envelope (constrained L1 math type in PC-DMIS).

L2

The green line is the constrained L2 plane targeting the external envelope (constrained L2 math type in PC-DMIS).

Migration

From earlier PC-DMIS versions to PC-DMIS 2019 R1:

A High Point Plane feature becomes a Primary Datum Plane feature *with a constrained L1 math type*.

A Tangent Plane becomes a Primary Datum Plane feature and the math type is preserved.

From PC-DMIS 2019 R1 to PC-DMIS 2017 R1 through PC-DMIS 2018 R2:

The Primary Datum Plane feature becomes a Tangent Plane feature.

From PC-DMIS 2019 R1 to PC-DMIS version 2016.0 (or earlier):

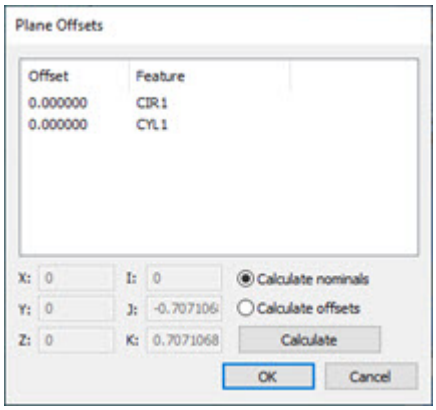
If the PC-DMIS version supports the High Point Plane, the Tangent Plane feature becomes a High Point Plane feature.

Constructing an Offset Plane

A plane can be constructed a specified distance from the input feature or features.

To construct an offset plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Offset** option.
3. From the **Feature** list, select a single plane feature or at least three features of any type.
4. Click the **Offsets** button to open the **Plane Offsets** dialog box.



Plane Offsets Dialog Box

5. Click the **Calculate Noms** option and enter the offset values for the input feature or features in the offset field, or click the **Calculate Offsets** option and change the nominal values (see the procedures below).
6. Click **Calculate** to calculate either the nominal values or the offset values.
7. Click the **OK** button to close the **Plane Offsets** dialog box.
8. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.

9. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

10. Click the **Create** button. PC-DMIS creates the offset plane based on the input feature or features.

Using a Single Plane Input Feature

PC-DMIS creates a parallel plane at the offset distance.

The Edit window command line for this option would read:

```
CONSTR/PLANE,OFFSET
ID=id1,,, …
OFFSET=val1…
```

Using Three Input Features

PC-DMIS iterates and constructs a plane such that the shortest distance from each input feature to the plane is the corresponding offset amount. PC-DMIS applies negative offsets in the same general direction that the points are measured. Positive offset values are applied opposite the probing direction. If there is no probing direction (for example, the input points were constructed), PC-DMIS uses the current workplane to determine the general direction for applying the offsets. Positive offsets are applied in the plus direction of the third axis of the current workplane. Negative offsets are applied in the minus direction of the third axis of the current workplane.

The sign (positive or negative) of the offset value controls on which side of the input features the plane is constructed. If you get the opposite plane than was expected, cancel the feature and reconstruct it, reversing the sign of the keyed in offsets. For example, if the offsets are 1.0, 2.5, 3.5, change them to -1.0, -2.5, -3.5.

The Edit window command line for this option would read:

```
CONSTR/PLANE,OFFSET
ID= id1, id2, id3, …
OFFSET= val1, val2, val3, …
```

Changing Offsets Directly to Calculate Nominals

To enter new offset values:

1. Click the **Offsets** button from the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**) to open the **Plane Offsets** dialog box.
2. In the **Offset** column, click on the **0.000000** value (or current value) to select it.
3. Type a new value.
4. Press the Enter key.
5. Click **Calculate** to update the nominal values based on any offset(s) you chose.
6. Click **OK** to save the offset.

Example of Calculating Nominals

The **Calculate** button on the **Plane Offsets** dialog box (**Insert | Feature | Constructed | Plane**) allows you to calculate the X, Y, and Z nominal values from new offsets entered.

For example, suppose you construct an offset plane between three circles (CIR1, CIR2, and CIR3). After you select the features and click the **Offsets** button, you have X, Y, and Z nominals of:

X = 6

Y = 2

Z = 0.95

If you change the offset values by 3.0 for each circle and click the **Calculate** button, the X, Y, and Z are updated to:

X = 6

Y = 2

Z = 3.95

If you then click **OK** and construct the offset plane, you'll notice that the newly constructed plane lies 3.0 units higher in the Z axis.

Plane offsets only apply the offset in the Z axis.

Changing Nominals Directly to Calculate Offsets

To enter new offset values:

1. From the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**), click **Offsets** to open the **Plane Offsets** dialog box.
2. Select the **Calculate offsets** option. The nominals portion of the dialog box becomes editable.

3. Change the **XYZ** or **IJK** values.
4. Click **Calculate** to update the offset values based on any nominals you've changed.
5. Click **OK** to save the offset.

Example of Calculating Offsets

The **Calculate** button on the **Plane Offsets** dialog box (**Insert | Feature | Constructed | Plane**) allows you to calculate the offset values appearing in the **Offset** column when you change the nominal value.

For example, suppose you construct an offset plane between three circles (CIR1, CIR2, and CIR3). After you select the features and click the **Offsets** button, you have X, Y, and Z nominals of:

X = 6

Y = 2

Z = 0.95

If you change the X, Y, and Z nominals to:

X = 6

Y = 2

Z = 3.95

and click the **Calculate** button, the offsets for the three circles are updated to:

3.000000 CIR1

3.000000 CIR2

3.000000 CIR3

If you then click **OK** and construct the offset plane, you'll notice that the plane is constructed at 3.0 units higher in the Z axis.

Constructing a Translated Plane

You can construct a Translated Plane feature from another input plane feature. The input plane feature must contain point data. Valid plane features with point data are Constructed Best Fit (BF), Constructed Best Fit Recompensate (BFRE), Measured, or Auto plane features. Invalid plane features do not contain point data (such as Cast, Mid, or Perpendicular).

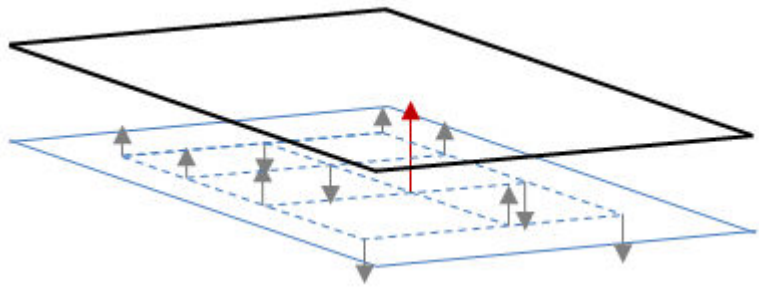
For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

PC-DMIS moves the translated constructed plane from the input plane in the direction of the input plane vector based on the calculation method:

- If you choose **Maximum Deviation**, PC-DMIS translates the plane to the maximal deviation point.
- If you choose **Standard Deviation**, PC-DMIS calculates the translation by multiplying the **Standard deviation multiple** value by the standard deviation.

The Translated Plane is offset from the input plane by the maximum deviation. The Standard Deviation plane is similarly offset, but the distance is a multiple of the standard deviation, as described above.



In this figure, the light blue plane represents the input feature, the black plane represents the maximum deviation plane, and the red arrow represents the maximum deviation.

To construct a translated plane:

1. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**).
2. From the **Method** list, select the **Translated Plane** option.
3. From the **Feature** list, select the input feature. This must be a Plane feature with point data.
4. Select the calculation method. The available options are **Maximum Deviation** or **Standard Deviation**.

5. If the calculation method is **Standard Deviation**, type the **Standard deviation multiple** value.
6. From the **Display** area, select how you want PC-DMIS to display the constructed plane. For details, see the "Using the Display Area" topic in this documentation.
7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click **Create** to insert the constructed Translated Plane command into the Edit window.

The Edit window command line for this option reads:

```
CONSTR/PLANE,TRANSLATED,feat_1,

METHOD/( MAXIMUM_DEVIATION | STANDARD_DEVIATION),
standardDeviationMultiple
```

Constructing an Extracted Plane

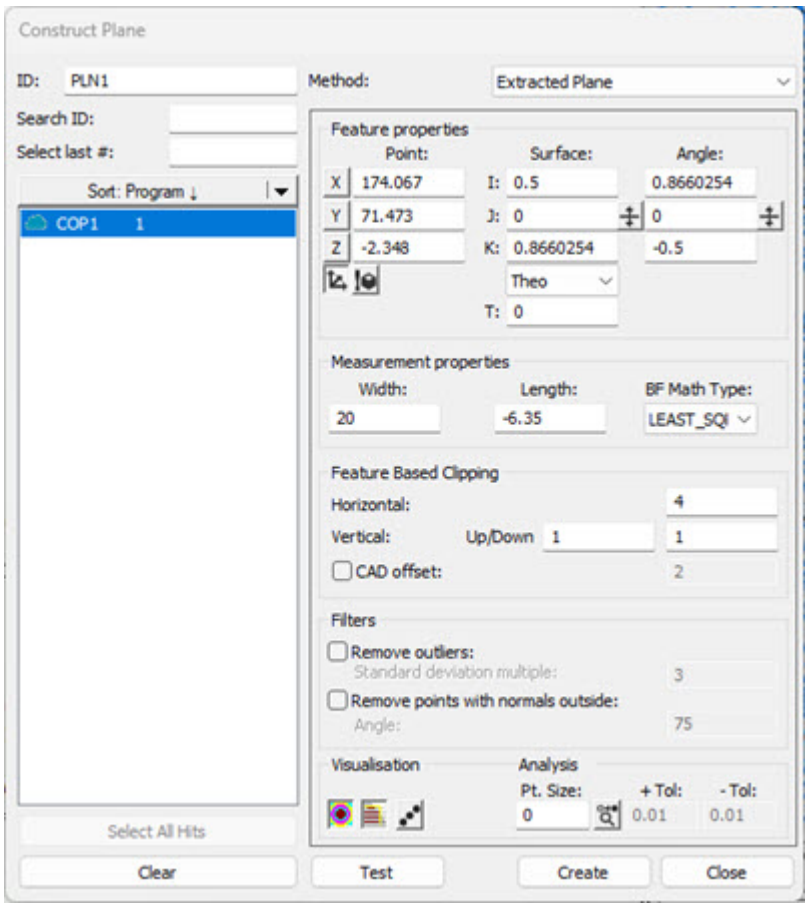
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a plane that PC-DMIS extracts from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Plane** dialog box (**Insert | Feature | Constructed | Plane**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Plane Dialog Box - Extracted Plane Option

3. From the **Method** list, select the **Extracted Plane** option.
4. From the **Reference** area, select the COP or Mesh that you want to use to extract the plane from.
5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.
6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. You can use the **Material Thickness Type** list and the **T** box below it to define the material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

   Flip Vectors

   Polar/Cartesian

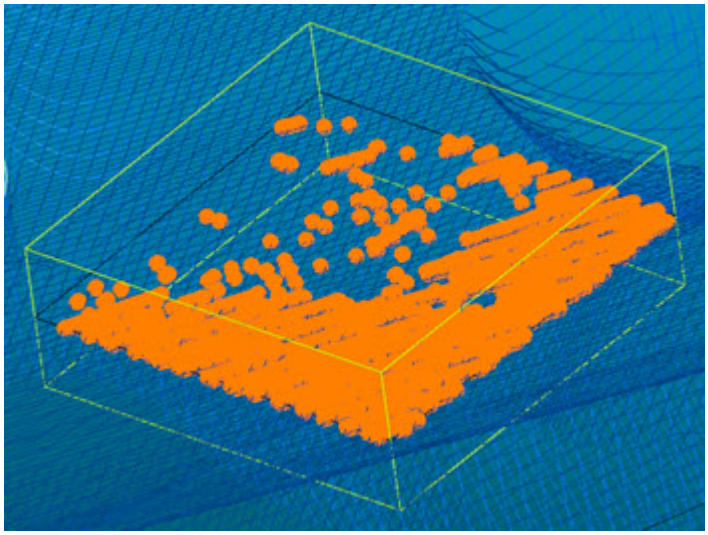
   Find Nearest CAD element

   For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation."

7. In the **Measurement properties** area, type the properties for the plane in the **Width** and **Length** boxes. Also, select the type of Best Fit algorithm to use to construct the plane from the **BF Math Type** list. For details on the algorithm types, see the "Best Fit Type (for Plane)" topic in this documentation.

   PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the zone that PC-DMIS uses for the extracted plane.

   - The black outline is the surface.
   - The green boundary represents the horizontal and vertical boundary zones.
   - The orange points are the candidate points that the extraction considers.

   Example of an extracted plane showing the candidate points.

8. From the **Feature Based Clipping** area, define the **Horizontal**, **Vertical**, and **Vertical Up/Down** values. This sets the dimensions for the extraction zone region. Consider part variability when you define the extraction zone.

   Alternatively, you can clip data within an offset boundary around all the CAD elements on a surface with the **CAD offset** option. This is also called *CAD segregation*. For details, see the "CAD offset" section of the "Feature Based Clipping Parameters" topic in the PC-DMIS Laser documentation.

   If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

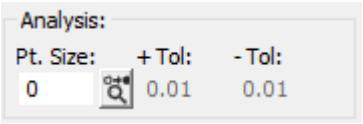
9. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

10. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

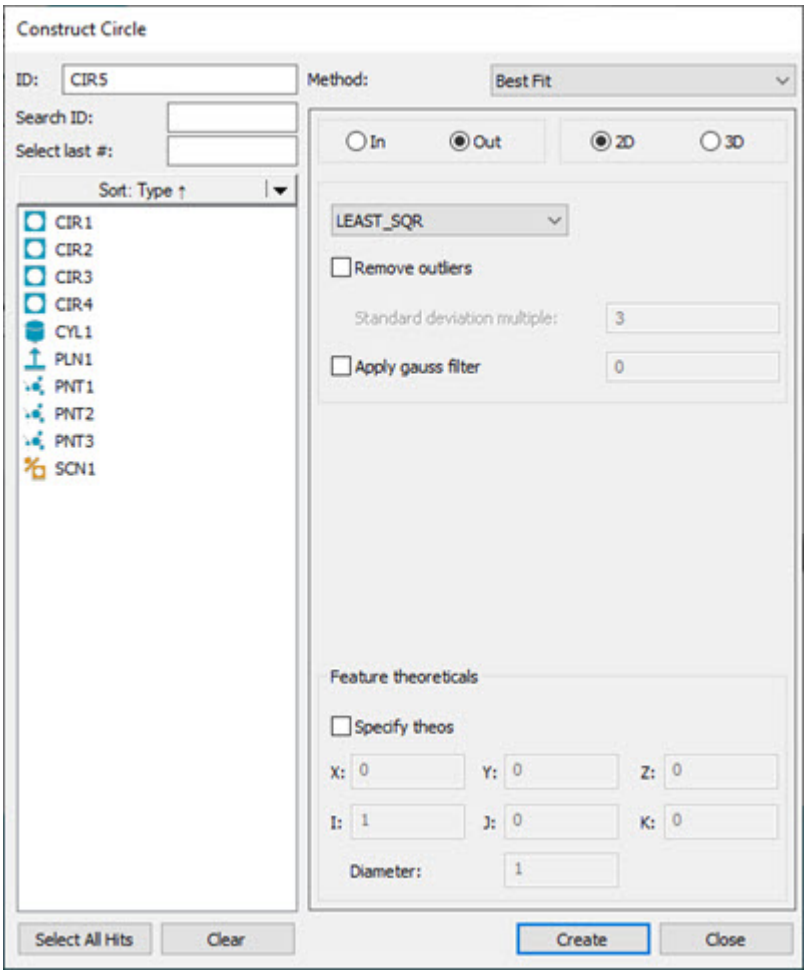
11. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

12. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
PLN3=FEAT/POINT,CARTESIAN,LEAST_SQR
   THEO/<73.598,33.658,0>,<0,0,1>,<1,0,0>
   ACTL/
<74.116,36.299,0.001>,<-0.0004985,0.0007789,0.9999996>
   WIDTH=5,LENGTH=5,
   THEO_THICKNESS,0,
   HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=2,
   USE OUTLIER REMOVAL=ON,0.125,
   REMOVE POINTS WITH NORMALS OUTSIDE=ON,25,
   CONSTR/PLANE,EXTRACTED_PLANE,REF=COP1
```

Constructing a Circle Feature



Construct Circle Dialog Box

There are several ways to use PC-DMIS to construct a circle. The following table shows the types of constructed circles found in the **Method** list and their required inputs. Some constructed circles may require no inputs while others may require three or more. The term "Any" in the following table indicates that the construction can take any type of feature as input for construction. You can select the input features in any order.

Method	SYMBOL IN EDIT WINDOW	# OF INPUT FEATS	FEAT #1:	FEAT #2:	FEAT #3:	COMMENTS
Auto Circle	-	-	-	-	-	See "Auto Circle Construction".
Best Fit Circle	BF	At least 3 inputs are needed.	-	-	-	Constructs a best fit circle using the given inputs. See the Note below for recommended inputs.
Best Fit with Recompensate Circle	BFRE	At least 3 inputs are needed. (1 must be a point)	-	-	-	Constructs a best fit circle using the given inputs. See the Note below for recommended inputs.
Intersect Circle	INTOF	2	Circle, Sphere, Cone, or Cylinder	Plane	-	Constructs a circle at the intersection of a circular feature with a plane, cone, or cylinder.
			Plane	Circle, Sphere, Cone, or Cylinder		
			Cone	Cone or Cylinder		
			Cylinder	Cone		
Cast Circle	CAST	1	Any	-	-	Constructs a circle at the centroid of the input feature.
Project Circle	PROJ	1 or 2	Any	Plane	-	One input feature will project the circle to the workplane.
Reverse Circle	REV	1	Circle	-	-	Constructs a circle with a reversed vector.

Tangent 2 Lines	TANLINES	2	Line	Line	-	Constructs a circle tangent to the two lines at a given diameter.
Tangent 3 Lines	TANLINES	3	Line	Line	Line	Constructs a circle tangent to the three lines.
Tangent 3 Circles	TANCIRCLES	3	Circle	Circle	Circle	Constructs a circle tangent to the three circles.
Scan Segment Circle	SCAN_SEGMENT	1	Scan	-	-	Constructs an arc from a part of a Linear Open or Linear Close scan.
Scan Minimum	SCAN_MINIMUM	1	Scan Minimum	-	-	Constructs a 2D circle of a given radius at a minimum point along a linear scan.
Cone Circle - also known as Gage Diameter	CONE	1	Cone	-	-	Constructs a circle at the specified diameter or height of the cone.
Sphere Circle	SPHERE	1	Sphere	-	-	Constructs a circle at the specified diameter or height of the sphere.
Cylinder Circle	CYLINDER	1	Cylinder	-	-	Constructs a circle at the specified diameter or height of the cylinder.

Extracted Circle	EXTRACTED_CIRCLE	1	COP or Mesh	-	-	Constructs an extracted circle from the COP or Mesh object at the specified diameter or height of the cylinder.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

To construct a circle:

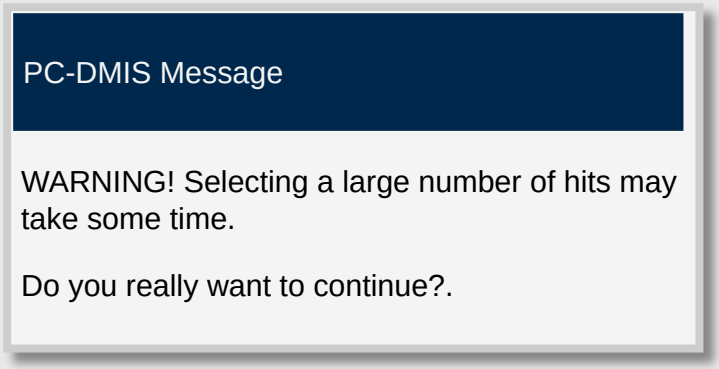
1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. Input the desired features.
3. Select the **In** or **Out** option.
4. Select the method of construction. Available options include:
    - * Auto Circle
    - Best Fit or Best Fit Recompensate Circle
    - Intersect Circle
    - Cast Circle
    - Projected Circle

- Reverse Direction Circle
- Tangent 2 Lines
- Tangent 3 Lines
- Tangent 3 Circles
- Scan Segment
- Scan Minimum Point
- Circle from a Cone
- Circle from Sphere
- Circle from Cylinder
- Extracted Circle

** Auto is the default method of construction. See "Auto Circle Construction".*

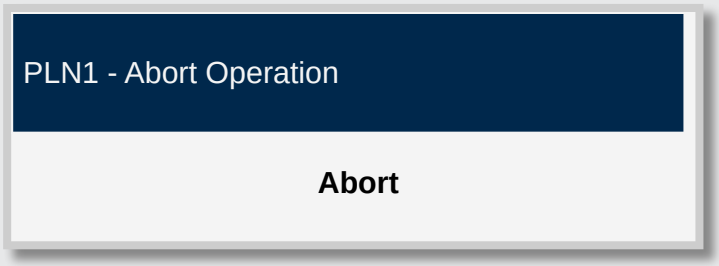
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

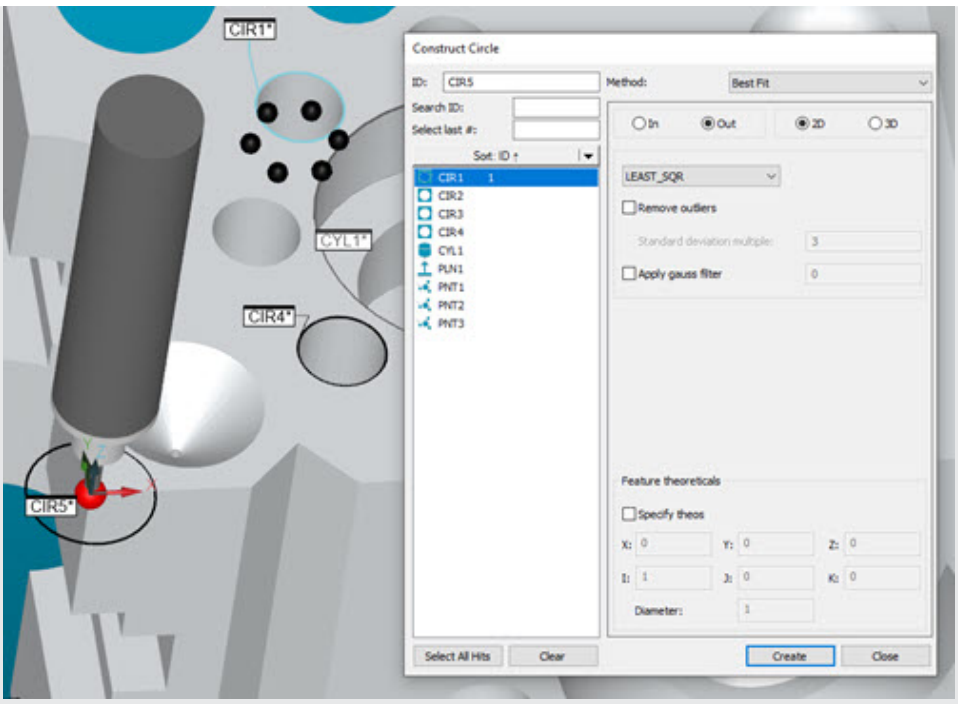


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

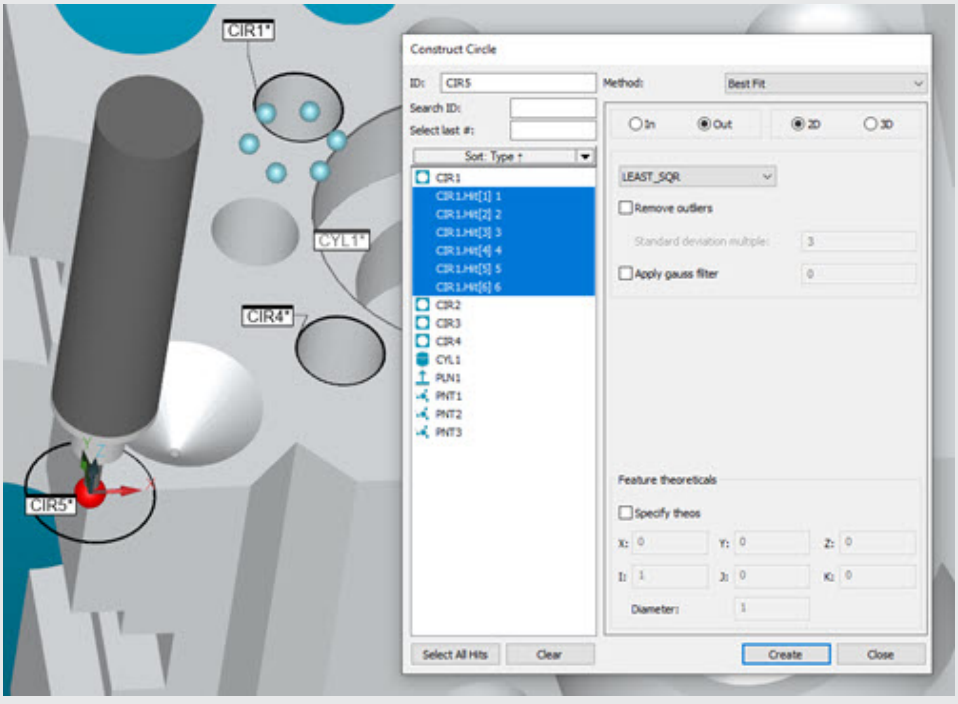
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

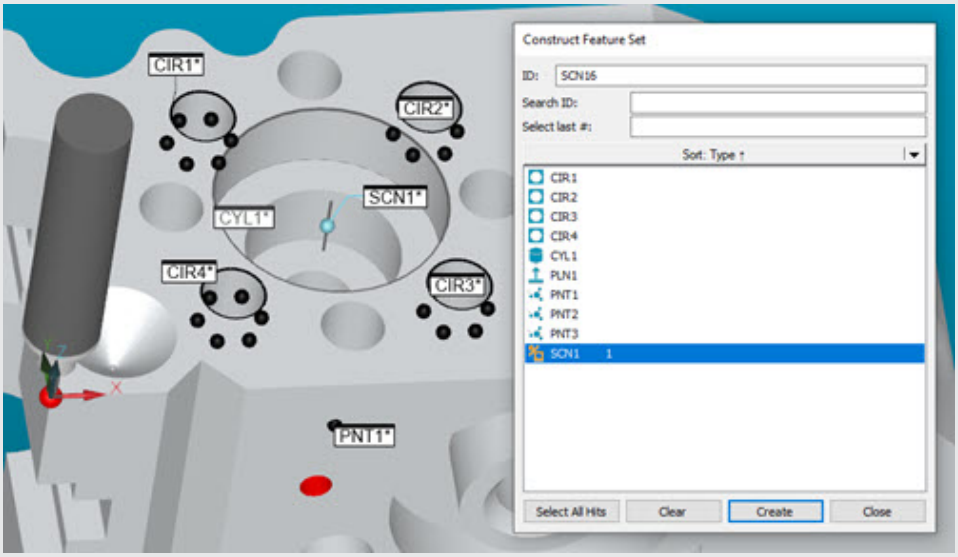
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

5. Some circle types have additional options or items that appear in the dialog box when selected. Select or use these options as needed.

6. Click the **Create** button.

The Edit window command line for a sample Circle Construction would read:

```
feature_name=FEAT/CIRCLE,TOG1,TOG4,TOG5
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam
CONSTR/TOG2,TOG3
```

If TOG2 = CIRCLE and TOG3 = BF or BFRE then the command has the following format:

```
feature_name=FEAT/CIRCLE,TOG1,TOG4,TOG5
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam
CONSTR/CIRCLE,TOG3,TOG7,feat_1,feat_2, …
OUTLIER_REMOVAL/TOG6, stdDevMultiplier
FILTER/TOG6, UPR =cutoffFrequency
```

The actual Edit report displays in all capital letters.

TOG1= POLR or RECT

TOG2 = CIRCLE

TOG3 = BF / BFRE / CAST / CONE / INTOF / PROJ / REV / TANLINES / TANCIRCLES / SCAN_SEGMENT

TOG4 = IN / OUT

TOG5 = LEAST_SQR / MAX_INSC / MIN_CIRCSC / MIN_SEP / FIXED_RAD (only for measured, BF and BFRE circles)

TOG6 = ON or OFF

TOG7 = 2D or 3D (This appears only if TOG3 reads BF or BFRE)

stdDevMultiplier = This multiplier value determines whether or not a measured point is an outlier. If the point from the circle is further than the standard deviation multiplied by this, then it is an outlier and will be removed if you have selected the **Remove Outlier** option.

cutoffWavelength = This value controls the amount of data smoothing. The longer the wavelength, the more the smoothing.

The first three lines displayed in the Edit window will be the same for constructed circles. The fourth line will be slightly different, according to the type of feature you are constructing. You can switch between the different types of circles by placing the cursor on *TOG3* and pressing F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

The following describe the available options for constructing a circle:

In / Out Circle

The **In** and **Out** options tell PC-DMIS whether to construct the circle as an internal or an external circle.

- If you select **In**, PC-DMIS constructs the circle as an internal circle.
- If you select **Out**, PC-DMIS constructs the circle as an external circle, or pin.

2D / 3D Circle

The **2D** and **3D** options tell PC-DMIS whether to construct the feature as a 2D or 3D circle. These options become available if you select the **Best Fit** or **BF Recomp** option from the **Method** list.

- If you select **2D**, PC-DMIS constructs the circle, projecting it onto the workplane.
- If you select **3D**, PC-DMIS constructs a best fit plane from the inputs. Those inputs are then projected to the plane, and a constructed circle is created from the projected points.

Auto Circle Construction

The list below indicates the type of circle that the software constructs when you select certain input features along with the **Auto** option. The order of the selected features is not important. If you select an incorrect input feature or features, PC-DMIS display an error message and does not automatically construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Access the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Auto** option.
3. Select the desired feature or features based on the table below.
4. Click the **Create** button.

Input Feature List

	Circle	Cone	Cylinder	Ellipse	Line	Plane	Point	Set	Slot	Sphere
Circle						Project Circle				
Cone		Intersect Circle	Intersect Circle			Intersect Circle				Intersect Circle
Cylinder		Intersect Circle								Intersect Circle
Ellipse						Project Circle				
Line						Project Circle				
Plane	Project Circle	Intersect Circle		Project Circle	Project Circle	Project Circle	Project Circle	Project Circle	Project Circle	Intersect Circle
Point						Project Circle				
Set						Project Circle				
Slot						Project Circle				
Sphere		Intersect Circle	Intersect Circle			Intersect Circle				

Any 1 Circle	Reverse Circle
Any 1 Cone (with Diameter)	Cone Circle
Any 1 Feature (Except Circle, Cone and Set)	Cast Circle
Any 1 Set	Best Fit Circle
One COP or Mesh object	Extracted Circle

Constructing a Best Fit or Best Fit Recompensate Circle

You can construct a "best fit" circle from three or more features. The constructed circle's vector is normal to the current workplane.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

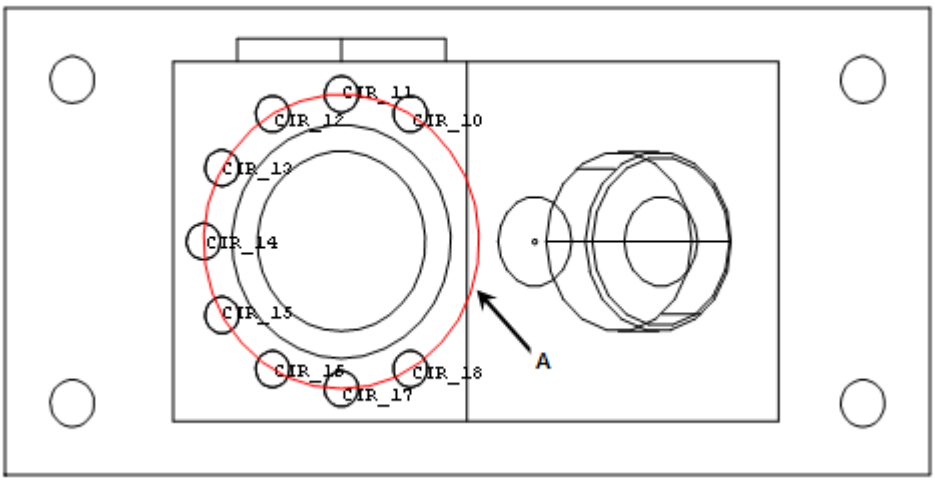
For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a Best Fit Circle or a Best Fit Recompensate Circle:

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. Select the type of Best Fit construction from the **Best Fit Type** list. For information on the different types, see "Best Fit Type (for Circle)".
4. Select at least three features.
5. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CIRCLE, BF,  feat_1, feat_2, …

OUTLIER_REMOVAL/ (OFF | ON),  stdDevMultiple

FILTER/(OFF | ON), UPR = cutoffFrequency
```



A - Best fit circle constructed from three or more features (in this example from three or more circle features)

Outlier removal and Filtering are described in the topics below:

Best Fit Type (for Circle)

This list becomes available if you select **Best Fit** or **BF Recomp** options when constructing the circle. It allows you to specify the type of Best Fit construction used. Available types include:

- LEAST_SQR
- MIN_SEP

- MAX_INSC
- MIN_CIRCSC
- FIXED_RAD

These types are discussed below:

LEAST_SQR
Least Squares - This calculation type provides a method of fitting in which the average squared radial distance from the data points to the circle is minimized. The square root of this quantity is the Root Mean Square (RMS) distance. Since the RMS distance is based on an average, some points may be further than the RMS distance from the computed circle.

MIN_SEP
Minimum Separation - This calculation type generates a circle that is halfway between two concentric circles containing the data points, with the difference of their radii as small as possible. The Min/Max math used by the MIN_SEP calculation minimizes the maximum error, or deviation, from the input data to the circle. The Min/Max error is one-half of the minimal separation. No input data points (or input features) lie farther than the Min/Max error from the Min/Max circle. This calculation determines whether or not all the input data (or input features) are within the given tolerances.

MAX_INSC
Maximum Inscribed - This calculation type generates an empty circle with the largest possible diameter that lies within the data. PC-DMIS first computes a Minimum Circumscribed circle and requires that the center of the Maximum Inscribed circle lies within it. This option could be used for a circular feature that requires a mating stud. For example, if the input data represents a hole, then this calculation returns a circle with the diameter of the largest stud that will fit inside the hole. Do not use this calculation type for arcs of less than 90 degrees.

MIN_CIRCSC
Minimum Circumscribed - This calculation type generates a circle with the smallest possible diameter that encloses the input data (or input features). This option could be used when measuring a stud that would fit into a mating circular feature. The resulting feature would be the smallest hole into which the stud would fit. Do not use this calculation type for arcs of less than 180 degrees.

FIXED_RAD
Fixed Radius - This calculation type creates a circle of a given diameter, positioned so that the maximal radial distance from the data points to the circle is minimized. It is similar to the Max/Min math used in the MIN_SEP calculation except that, since the diameter is known in advance, the radius cannot vary. Only the circle's position is allowed to vary.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

Remove Outliers / Standard Deviation Multiple for a Constructed Circle

For a best fit (BF) or best fit recompensate (BFRE) circle, you can choose to remove outliers based on the distance from the best fit feature. This allows the removal of anomalies that arise in the measurement process.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

PC-DMIS first fits a circle to the data, then determines which points are outliers based on the standard deviation multiple. It then does the following:

- Recalculates the best fit circle with those outliers removed.
- Checks for outliers again.
- Recalculates the best fit circle.
- Continues repeating this process until no more outliers exist or until PC-DMIS cannot compute the circle (PC-DMIS can't compute the circle if there are fewer than three data points).

Apply Gauss Filter / Cutoff Frequency

The Best Fit (BF) and Best Fit Recompensate (BFRE) constructed circles have the option of filtering the deviations of the measured data points from the best fit circle computed from the measured data.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Apply Gauss Filter - If you mark this check box, PC-DMIS applies a Gaussian filter with a cutoff frequency input as undulations per revolution (UPR). Generally, a lower cutoff frequency produces smoother filtered data.

Remove Outliers - If you mark this check box and you are filtering the data, it removes the outlier data prior to filtering.

Constructing an Intersect Circle

You can construct a circle between a cone (circle, cylinder, sphere) and a plane. It can also be created between two concentric cones or a concentric cone/cylinder combination.

PC-DMIS creates a circle at the intersection of the circular feature and the plane or between the cone/cone or cone/cylinder combinations.

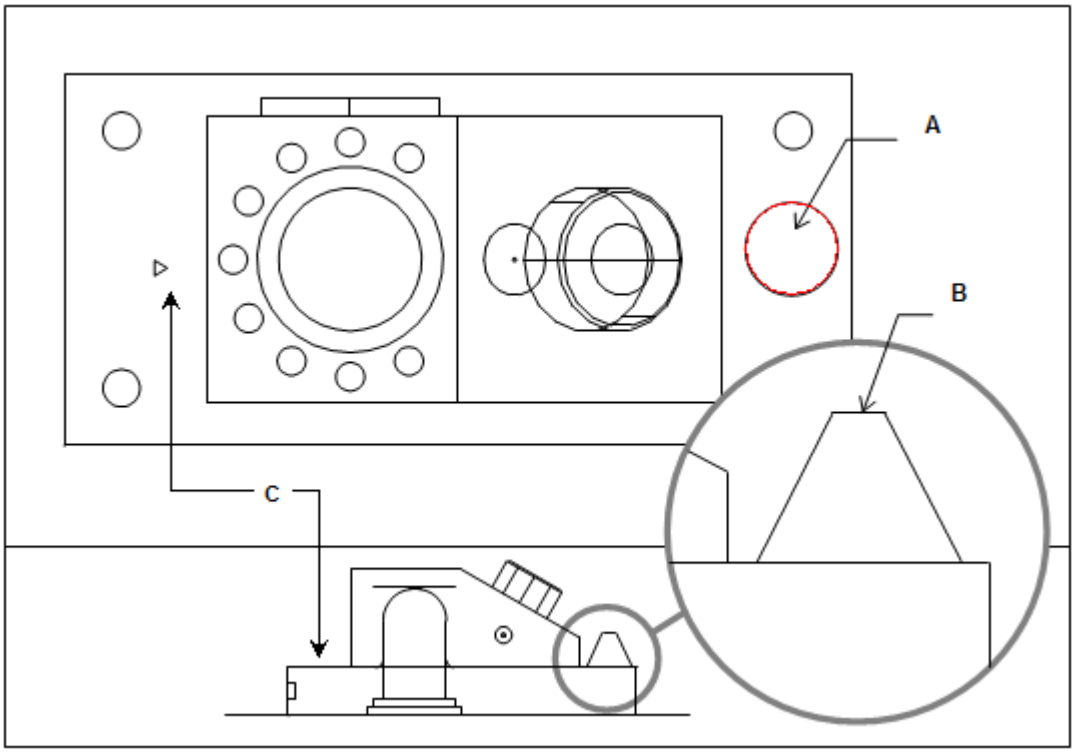
- In the case of a circular feature and a plane intersection, PC-DMIS always makes a true circle (not an ellipse) even if the circular feature is not exactly perpendicular to the plane. The center point of the new circle is at the pierce point of the circular feature's centerline and the plane. The vector of the circle is the vector of the piercing circular feature.
- In the case of a cone/cone or cone/cylinder combination, a true circle is created even if the intersecting features do not form a true circle.

To construct an intersect circle:

1. Access the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Intersect** option.
3. Select the first feature. It must be either a Circle, Cone, Cylinder, or Sphere.

4. Select the second feature. It must be a Plane.
5. Click the **Create** button.

The Edit window command line for this option would read:
```
CONSTR/CIRCLE,INTOF,feat_1,feat_2
```



A - Circle Constructed from the intersection of a cone and a plane.
B - Cone feature
C - Plane feature
Constructing a Circle from a Cone and a Plane

Constructing a Cast Circle

You can construct a circle by changing any given feature into a circle. PC-DMIS constructs the circle at the centroid of the input feature. If you use a sheet metal point, the diameter is the probe diameter. For some sheet metal features (such as slots and notches), PC-DMIS uses the width as the diameter. For features that do not have diameters (lines, points, etc.), PC-DMIS uses a value that is four times the probe diameter.

You can modify the diameter of the circle; this changes the circle from DEPENDENT to INDEPENDENT. This means that when the circle is executed, the diameter does not change based on the input feature but is independent of the input feature, while the position and vector remain dependent on the input feature. This allows you to have control over the diameter in cases where the input feature doesn't really have a diameter, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field that you can change.

PC-DMIS uses this diameter value for all calculations instead of using the default diameter value as described above.

To construct a cast circle:

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Cast** option.
3. Select at least one feature of any type.
4. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CIRCLE,CAST,feat_1,(DEPENDENT | INDEPENDENT)
```

Constructing a Projected Circle

You can construct a circle from any feature and a plane. PC-DMIS will project the centroid of the given feature onto the plane, creating a circle. If there is only one input feature, the projection will be to the current workplane. The diameter of the projected circle will be four times the probe's diameter.

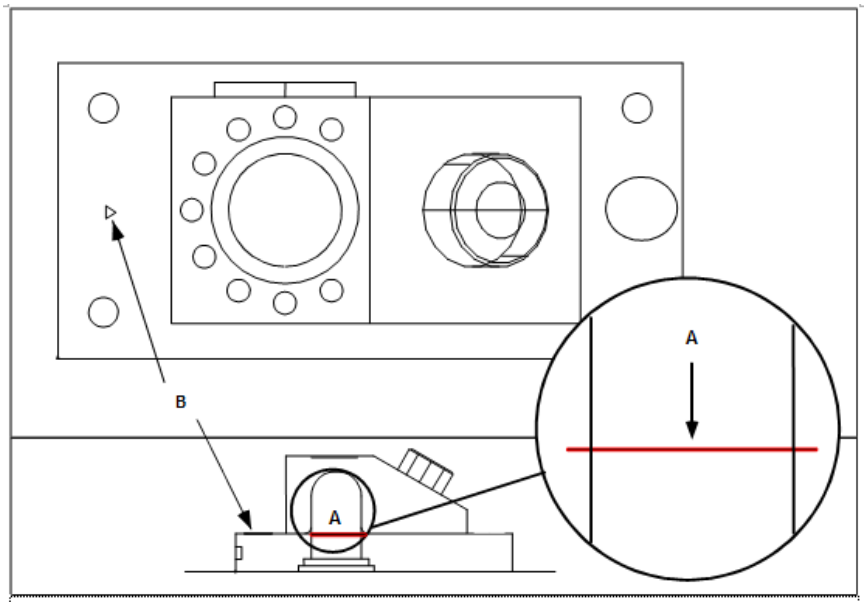
To construct a projected circle:

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Projection** option.
3. Select a feature of any type.

   You can select a second feature, but it must be a Plane.

4. Click the **Create** button.

The Edit window command line for this option would read:
`CONSTR/CIRCLE,PROJ,` *feat_1,(feat_2)*



A - Circle constructed from the projection of a circle and a plane.
B - Plane feature.
Example of a circle feature constructed from a circle and a plane.

Changing the Direction of a Circle

You can construct a circle with a reversed vector.

To change the direction of a circle:

1. Access the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Reverse** option.
3. Select one feature. It must be a circle.
4. Click the **Create** button.

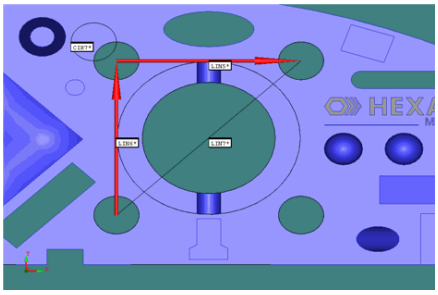
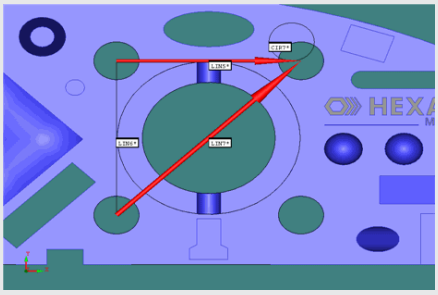
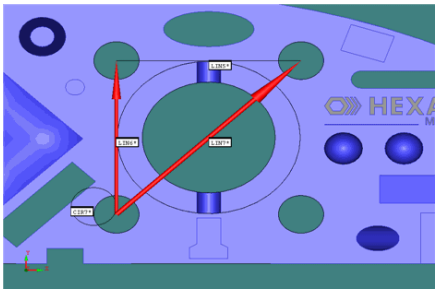
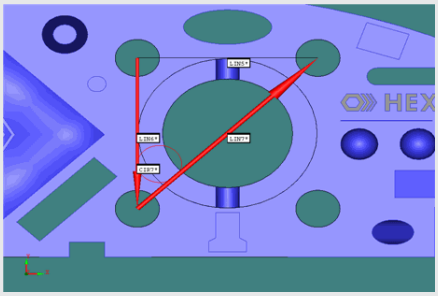
The Edit window command line for this option would read:
```
CONSTR/CIRCLE,REV,feat_1
```

Constructing a Tangent Circle

You can construct the following three types of tangent circles using the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**):

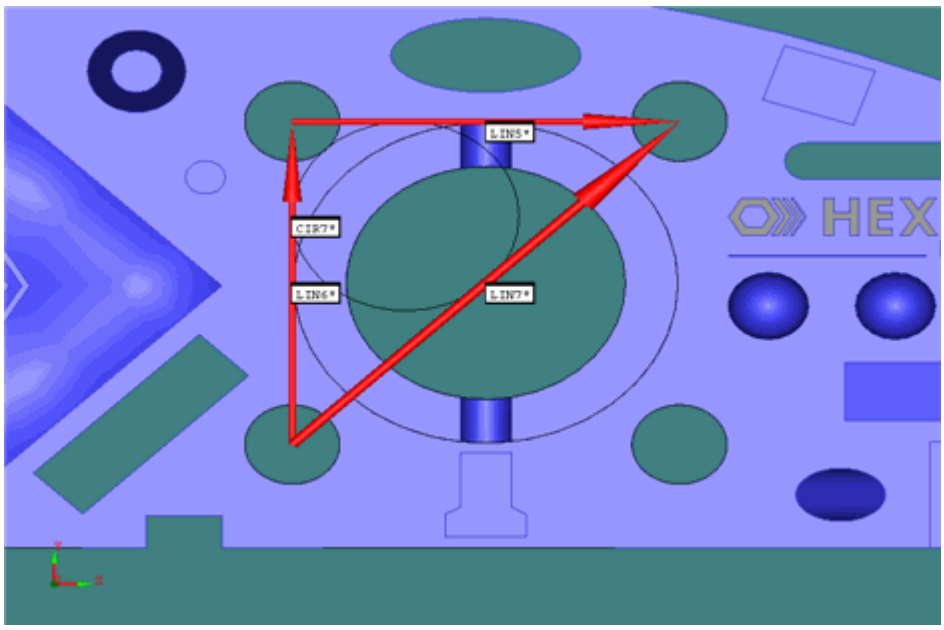
- **Tangent 2 Lines** - This option constructs a circle tangent to two lines. The exact location is determined by the size of the circle and the direction of the lines. Type a **Diameter** value for the construction after selecting the two input lines, and then click **Create**. If PC-DMIS doesn't show the constructed circle where you expect it to be, change the direction of one of the lines.

Examples of a Circle Feature Constructed Tangent to 2 Lines

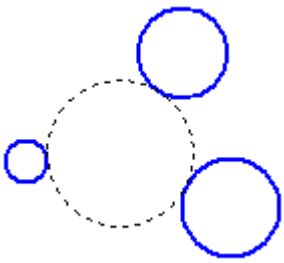
Circle 7 Constructed Tangent to Lines 5 and 6	
Circle 7 Constructed Tangent to Lines 5 and 7	
Circle 7 Constructed Tangent to Lines 6 and 7	
Circle 7 Constructed Tangent to Lines 6 and 7. Note the difference from the above example. Because the direction of Line 6 changed, the location of the constructed circle changed.	

- **Tangent 3 Lines** - This option constructs a circle tangent to three input lines that form a triangle. Select the three input lines, and then click **Create**.

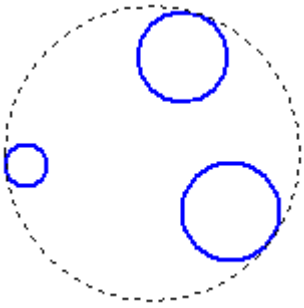
Example of a Circle Feature Constructed Tangent to 3 Lines



- **Tangent 3 Circles** - This option constructs a circle tangent to three input circles. Select the three input circles, and then click **Create**. The tangent circle can either contain all three circles (circumscribed circle) or none of the three (inscribed circle).



Example of an inscribed circle tangent to three input circles



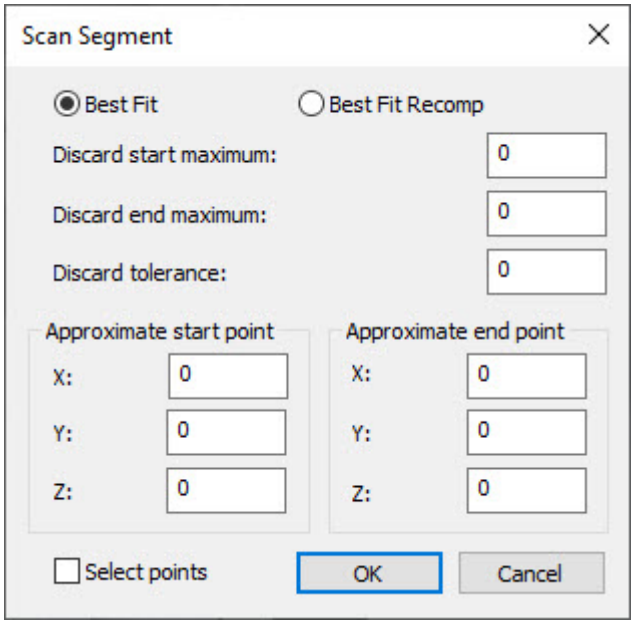
Example of a circumscribed circle tangent to three input circles

Constructing an Arc from Part of a Scan

You can construct a circle from a segment of a Linear Open, Linear Close, or Basic Circle scan. PC-DMIS creates an arc from part of the scan.

To construct a scan segment circle:

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Scan Segment** option.
3. Select a previously created Linear Open, Linear Close, or Basic Circle scan.
4. Click the **Segment Data** button to open the **Scan Segment** dialog box.



Scan Segment Dialog Box

5. Select either the **Best Fit** or **BF Recomp** option.
6. Select the portion of the scan used in the construction from this dialog box.
7. Type the number of points to potentially discard in the **Discard Start** and **Discard End Maximum** boxes.
8. Type a distance from the best fit circle into the **Discard Tolerance** box. This tolerance is a form tolerance; it allows you to control what end points PC-DMIS accepts as part of the arc. If the distance from the scan point to the best fit arc is greater than this tolerance value, then PC-DMIS discards the end point.
9. Select the **Select Points** check box to input the **Approximate Start** and **End Points** of the scan, and then click in the Graphic Display window to fill in the X, Y, and Z fields. You can click anywhere in the Graphic Display window; however, PC-DMIS places the point on the scan nearest to where you clicked. You can also type in the point values.
10. Click **OK** to accept the data and close the **Scan Segment** dialog box.
11. Click **Create** to construct the arc from the scan.

The Edit window command line for this option reads:

```
 CONSTR/
CIRCLE,SCAN_SEGMENT,fit_type,feat_1,start_x,start_y,start_z,end_x,end_y,en
```

If you want more than one arc or line from a given scan, you need to add another command for a different section of the scan.

Determining the Data Used to Construct the Arc

The software determines the data used to construct the arc as follows:

- First, the software determines a segment of the scan using a starting point and an ending point within the scan. The software chooses the starting and ending points as the point in the scan nearest `[start_x, start_y, start_z]` and the point nearest `[end_x, end_y, end_z]` respectively.
- The software then discards the points from the start and end points of the scan. The number of points discarded from the start is `discard_start` and from the end is `discard_end` . An arc is then fitted to this set of points.
- Finally, the software adds the starting and ending points back in if they are within the defined tolerance. The arc is then re-fitted once more to the new set of points.

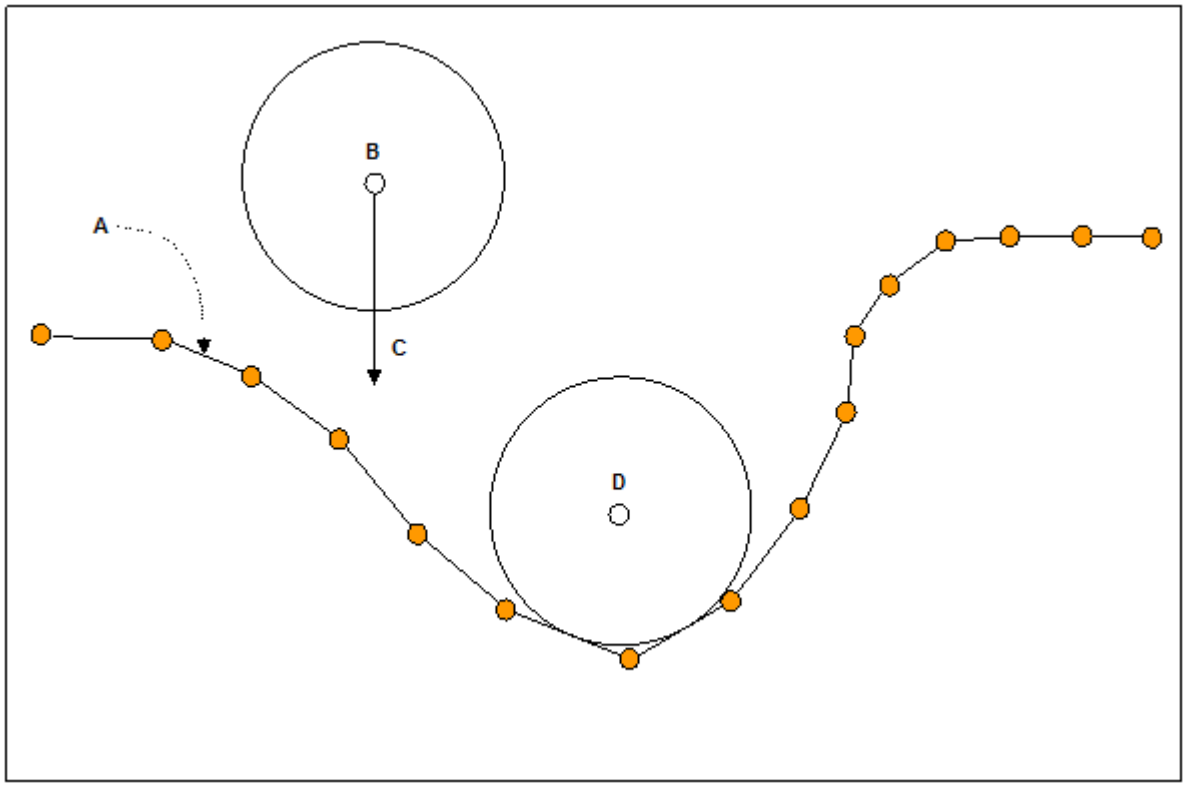
The value of `fit_type` may be `BF` (Best Fit) or `BFRE` (Best Fit Recompensate). This determines whether the software performs a best fit or best fit with recompensation when calculating the arc. For a description of Best Fit and Best Fit Recompensate, see "Constructing a Best Fit or Best Fit Recompensate Circle".

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Constructing a Circle at a Scan's Minimum Point

This feature lets you construct a 2D circle of a given radius at a minimum point along a linear scan. PC-DMIS finds the minimum point using a starting point and a downward direction vector (as shown below). Think of the downward vector as a gravity vector. In a sense, the circle will be "pulled" in that direction.



A - Scan
B - Start Point
C - Down Vector
D - Final Position
Minimum point of a circle along a scan with a defined downward vector and starting point.

PC-DMIS projects the scan to the current workplane and the circle will lie in a plane parallel to the workplane. The software interprets the scan as a line between consecutive points (piecewise linear). Thus, the circle that the software places at the minimum point along the scan will not "fall" between two consecutive scan points, but instead is constrained to touch a line that connects the two points.

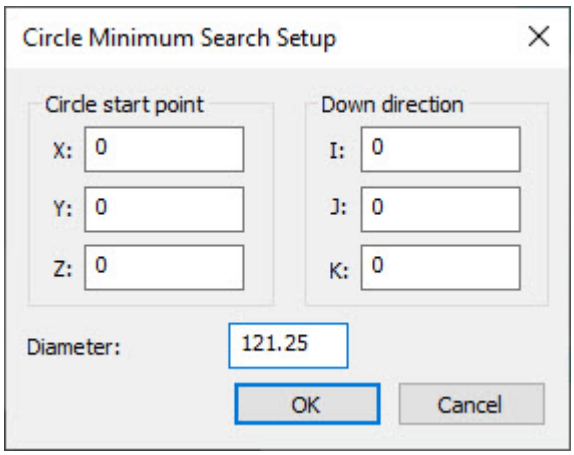
Valid Input Types

The input for this construction *must be a line type scan*. This *excludes* all scans designed to scan a surface, such as Patch, UV, Grid, Multisection, Manual laser, and Cylinder scans.

Construction Procedure

To start this construction:

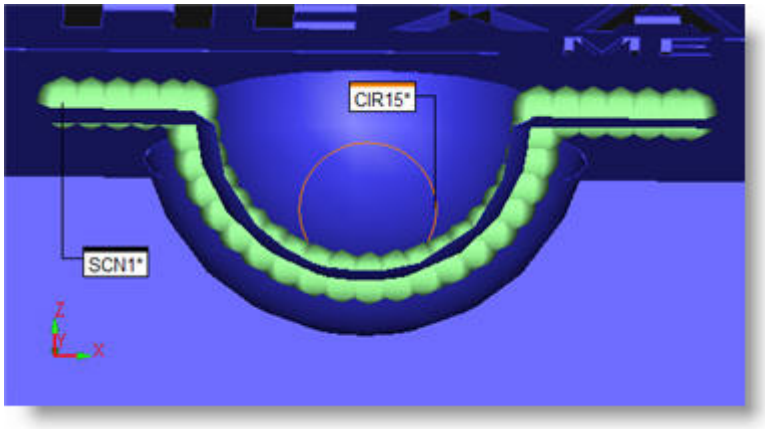
1. Access the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Scan Minimum** option.
3. Select a linear scan from the feature list. You cannot use a surface scan.
4. Click the **Search Setup** button.
5. The **Circle Minimum Search Setup** dialog box appears:



Circle Minimum Search Setup dialog box

6. Define the circle's start point, its down direction, and its diameter.

7. Click **OK** to construct the circle. PC-DMIS constructs the circle and inserts the construction command into the Edit window:

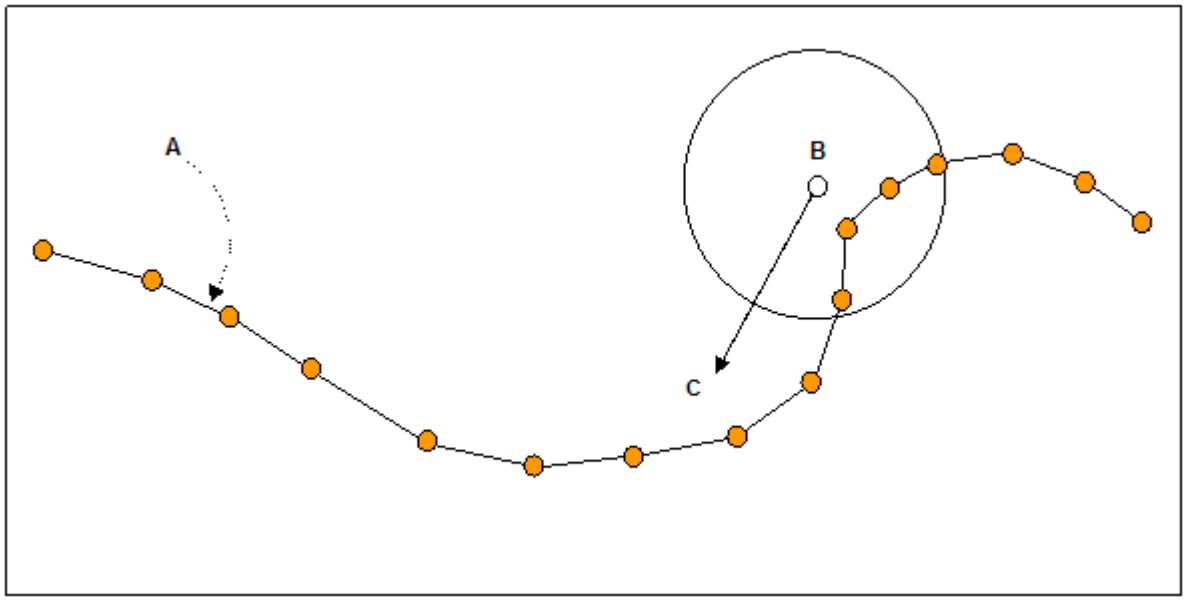


Cutaway view showing a circle (CIR15) constructed at the minimum point on the scan line (SCN1)

Rules for Construction

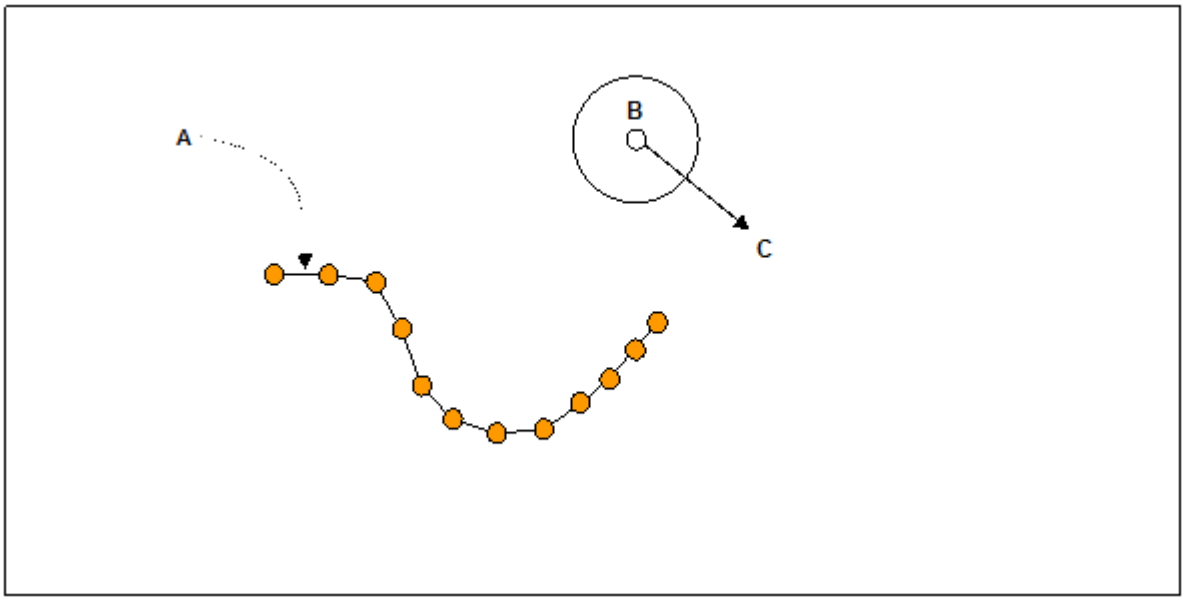
A valid starting point and vector follow two rules:

First, a circle of the given diameter and starting point should not intersect the scan. The image below shows a violation of this rule.



A - Scan
B - Start Point
C - Down Vector
Invalid starting point due to intersection with the scan

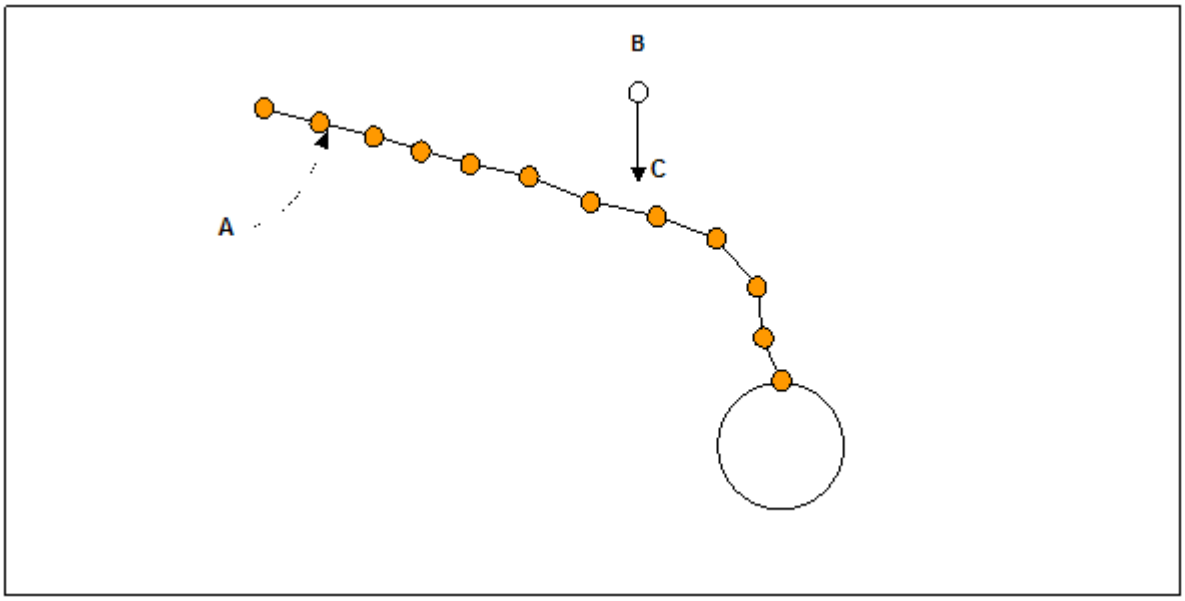
Second, the circle projected from the start point along the down vector must intersect the scan. The image below shows a violation of this rule.



A - Scan
B - Start Point
C - Down Vector
Invalid starting point due to the circle missing the scan

No Local Minimum

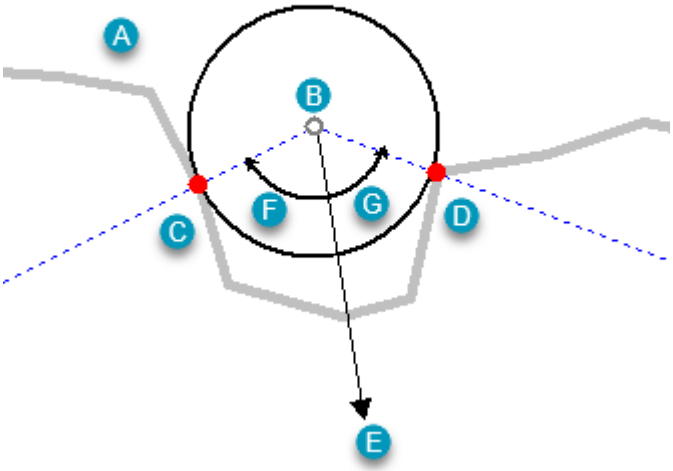
If the scan has no local minimum, or a natural resting place for the circle, then the circle follows the scan to its lowest point while still in contact with the scan.



A - Scan
B - Start Point
C - Down Vector
Example of a scan without a local minimum.

Edit Window Command for a Constructed Scan Minimum Circle

```
CIR1        =FEAT/CIRCLE,RECT,OUT
             THEO/<tx,ty,tz>,<ti,tj,tk>,td,ta1,ta2
             ACTL/<mx,my,mz>,<mi,mj,mk>,md,ma1,ma2
             CONSTR/CIRCLE,SCAN_MINIMUM,Scan ID
             CONTACT POINT/
<tcp1x,tcp1y,tcp1z>,<mcp1x,mcp1y,mcp1z>
             START ANGLE/tca1,mca1
             CONTACT POINT/
<tcp2x,tcp2y,tcp2z>,<mcp2x,mcp2y,mcp2z>
             END ANGLE/tca2,mca2
             TOLERANCE/tol
             START/xSP, xSP, xSP
             DOWN/iDV, iDV, iDV
```



A - Scan line
B - Circle's final position
C - Contact Point 1
D - Contact Point 2
E - Down Vector
F - Contact Angle 1
G - Contact Angle 2

tx,ty,tz
These represent the theoretical circle position.

ti,tj,tk
These represent the theoretical circle vector.

td
This represents the theoretical circle diameter.

ta1
This represents the theoretical angle 1 value.

ta2
This represents the theoretical angle 2 value.

mx,my,mz
These represent the measured circle position.

mi,mj,mk
These represent the measured circle vector.

md
This represents the measured circle diameter.

ma1
This represents the measured angle 1 value.

ma2
This represents the measured angle 2 value.

Scan ID
This represents the id of the scan to be used.

tcp1x,tcp1y,tcp1z
These represent the theoretical XYZ position for contact point 1.

mcp1x,mcp1y,mcp1z
These represent the measured XYZ position for contact point 1.

tca1
This represents the theoretical contact angle 1 value.

mca1
This represents the measured contact angle 1 value.

tcp2x,tcp2y,tcp2z
These represent the measured XYZ position for contact point 2.

mcp2x,mcp2y,mcp2z
These represent the measured XYZ position for contact point 2.

tca2
This represents the theoretical contact angle 2 value.

mca2
This represents the measured contact angle 2 value.

tol
This represents the tolerance value to use when locating the two contact points. PC-DMIS recomputes the contact points by averaging all the points that fall within the tolerance given.

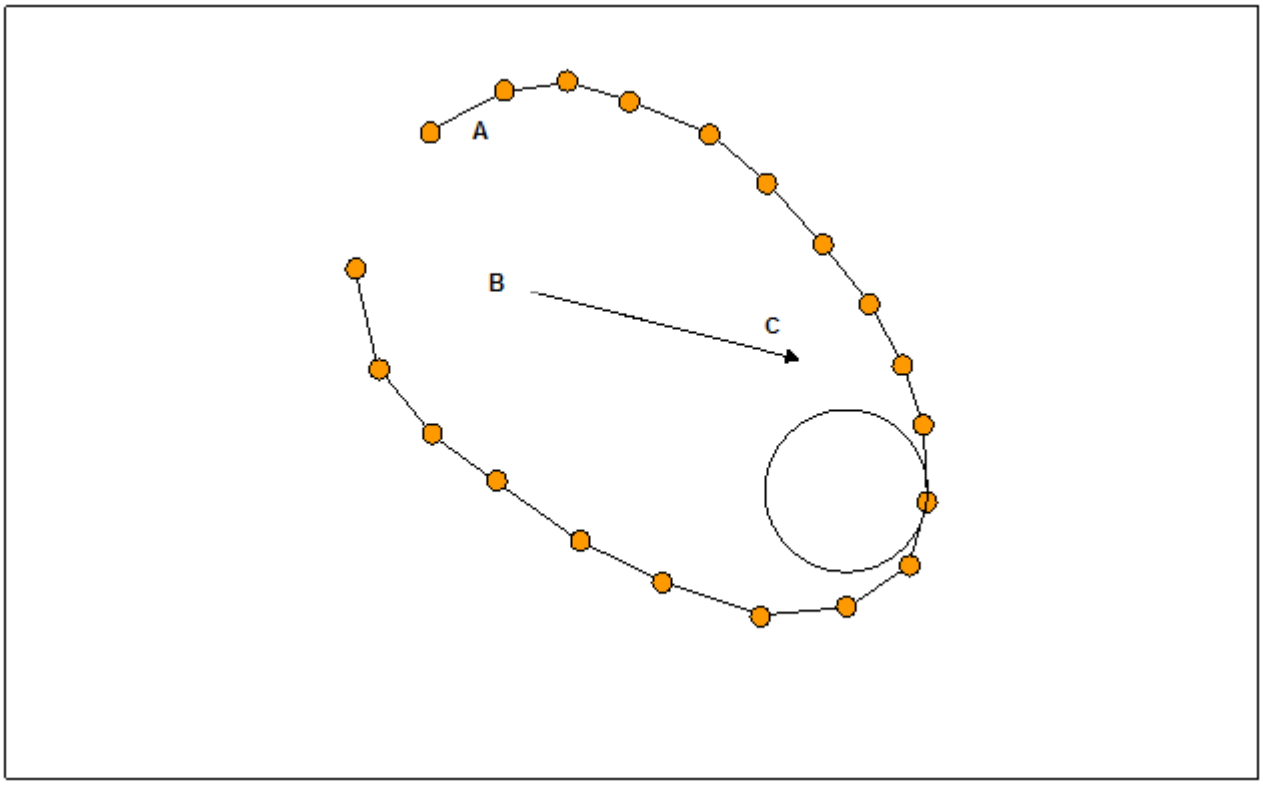
xSP, xSP, xSP
These represent the starting point for finding the minimum.

iDV, iDV, iDV
This represent the downward direction vector.

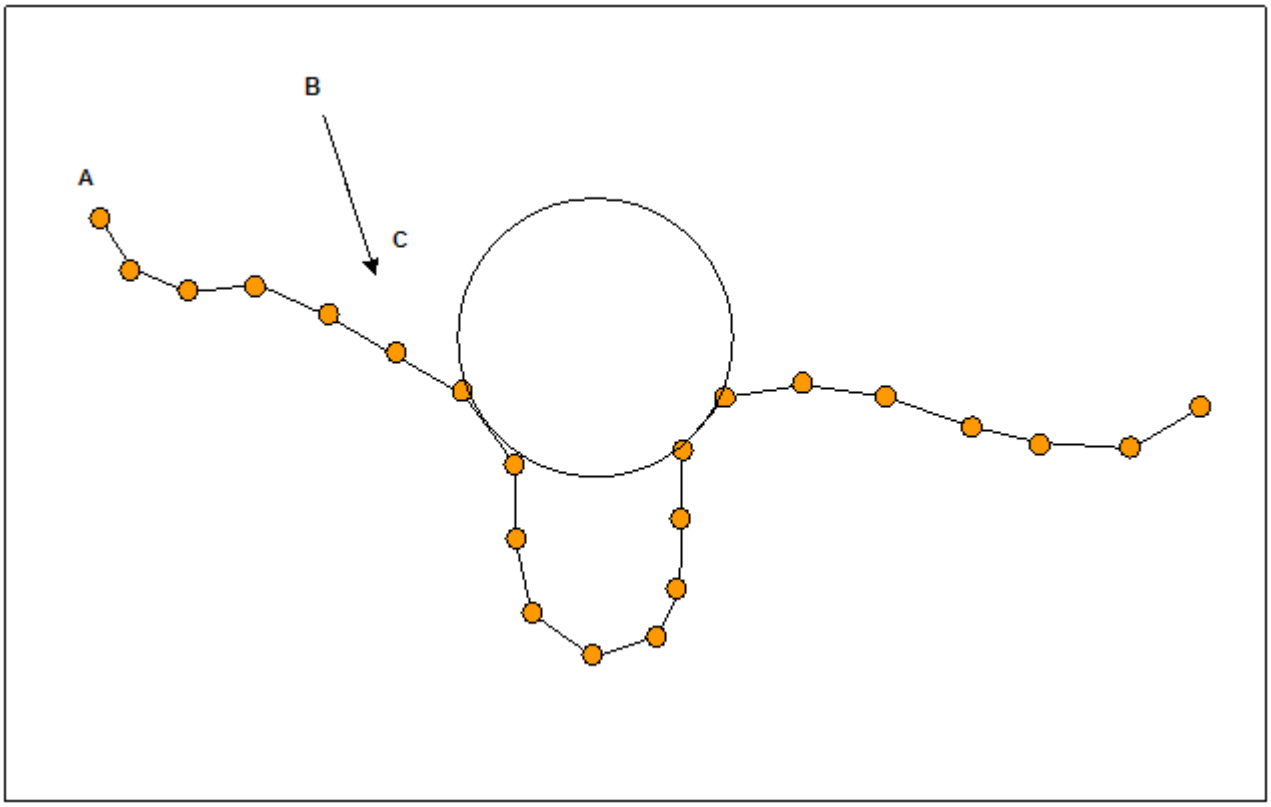
Using Expressions

You can also use expressions within the Edit window to pull information from a Constructed Scan Minimum Circle. See "Accessing Information from a Constructed Scan Minimum Circle" in "Using Expressions and Variables".

Additional Examples



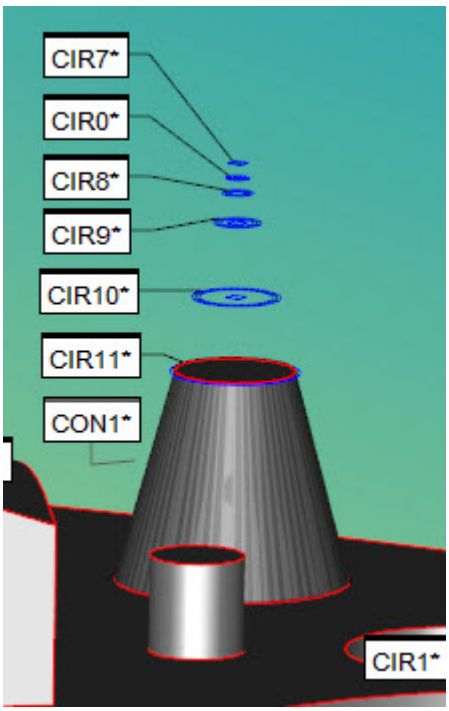
A - Scan
B - Starting Point
C - Down Vector
Starting point inside of a scan



A - Scan
B - Starting Point
C - Down Vector
Scan where not all points are reachable given the circle size

Constructing a Circle from a Cone

You can construct a circle from a cone at a specified diameter of the cone, or you can construct a circle at a given height along that cone from the current alignment plane. A Cone Circle feature that you construct at a given height is sometimes known as a "gage diameter" or "gage point".



Example showing multiple circles created at different heights from the cone input feature (CON1).

Understanding the Height Value

If you construct a circle at a given height, PC-DMIS calculates the circle in this way:

- It creates a plane from the reference point and the reference vector.
- It then creates a parallel offset plane from this plane by the height value. This parallel plane intersects with the cone axis, and the intersection creates the location of the resulting circle feature. The diameter of the circle is the diameter of the cone at that intersection point.

Available reference point (REF_POINT) options:

- CONE_VERTEX
- CONE_START
- CONE_END
- ORIGIN

Available reference vector (REF_VECTOR) options:

- CONE_VECTOR
- WORKPLANE
- ZPLUS
- ZMINUS

- XPLUS
- XMINUS
- YPLUS
- YMINUS

For example, if you use the origin as a reference point and ZPLUS as your reference vector, PC-DMIS creates a plane from the origin point and the ZPLUS vector. Then it creates a parallel plane at the given height value, and where it intersects with the cone, it creates the circle feature. Your Edit window's code might look something like this:

```
CIR2=FEAT/CIRCLE,RECT,OUT

THEO/-67.295,2.595,-7.152,0.0310723,-0.0214397,-0.9992872,29
.411

ACTL/-67.295,2.595,-7.152,0.0310723,-0.0214397,-0.9992872,29
.411
   CONSTR/CIRCLE,CONE,CON2,HEIGHT,5,REF_POINT =
ORIGIN,REF_VECTOR = ZPLUS
```

To construct a Cone Circle:

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Cone** option.
3. Select one feature. The type must be a cone.
4. Select either **DIAMETER** or **HEIGHT** from the **Type** list.
5. Type a value for the diameter or height into the **Value** box.
6. If you selected **Height**:
    - Select a reference point from the **Point** list.
    - Select a reference vector from the **Vector** list.
7. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CIRCLE,CONE,DIAMETER,feat_1
```

or

```
CONSTR/
CIRCLE,CONE,HEIGHT,value,REF_POINT=point,REF_VECTOR=vector,f
eat_1
```

Constructing a Circle from a Sphere

You can construct a circle from a sphere at a specified diameter of the sphere, or you can construct it at a given height from the sphere centroid.

Valid input feature type

Sphere

Valid construct type

Diameter or Height

Available reference point (REF_POINT) options:

- SPHERE_CENTROID

Available reference vector (REF_VECTOR) options:

- SPHERE_VECTOR
- WORKPLANE
- ZPLUS
- XPLUS
- YPLUS
- ZMINUS
- XMINUS
- YMINUS

Construct type input value overview

When you use a Diameter construct type, the input value is the diameter of the resultant sphere circle.

PC-DMIS uses the input value as the diameter of the resolved circle. It then calculates the centroid along the reference vector from the reference point that fits the diameter so that it is equal to the input value.

If you use a Height construct type, the input value represents the distance along the reference vector from the reference point, which is the sphere centroid.

PC-DMIS uses the value points as the offset from the reference point along the reference vector. The resultant centroid and diameter are identical after the offset is applied.

To Construct a Sphere Circle

1. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Sphere** option.
3. Select one feature. The type must be a sphere.
4. Select either **DIAMETER** or **HEIGHT** from the **Type** drop-down list.
5. Type a value for the diameter or height into the **Value** box.
6. If you selected **Height**:
    - Select a reference point from the **Point** list.
    - Select a reference vector from the **Vector** list.
7. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CIRCLE,SPHERE,feat_1,DIAMETER,REF_VECTOR = ZPLUS
```

or

```
CONSTR/CIRCLE,SPHERE,feat_1,HEIGHT,0,REF_POINT =
SPHERE_CENTROID,REF_VECTOR = vector
```

Constructing a Circle from a Cylinder

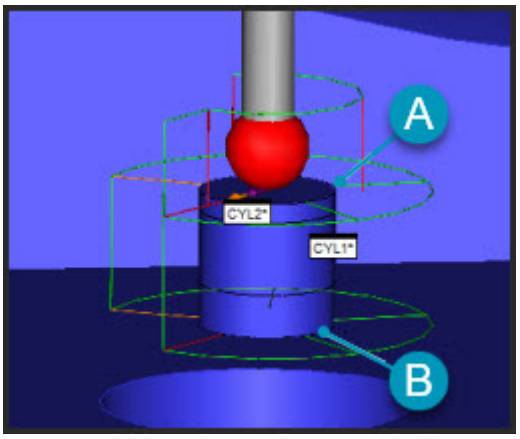
Similar to the cone and sphere circle constructions, this type of construction creates a circle from a cylinder at a height (or distance) along the defined vector. The resulting circle feature will have the same diameter as the reference cylinder. This type of construction takes three inputs: a height **Value**, a reference **Point**, and a **Vector**.

Value - This box lets you type the height value. PC-DMIS constructs the circle at this distance from the selected reference point and along the selected vector. A positive value uses the same direction as the vector. A negative value will use the opposite direction along that vector.

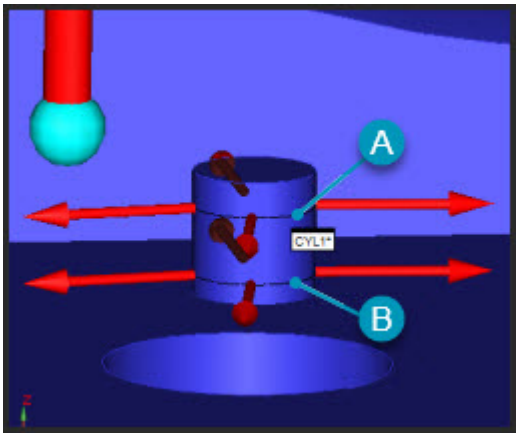
Point - This list lets you define a reference point from which PC-DMIS constructs the circle. It has these options:

- CYLINDER_START - The start location of the cylinder. This point is located at the centroid of the circle defined from the first level of hits.
- CYLINDER_END - The end location of the cylinder. This point is located at the centroid of the circle defined from the last level of hits.
- ORIGIN - The origin of your coordinate system.

These images show some sample start and end locations for different cylinder types:



***A** - A sample CYLINDER_START location for an Auto Cylinder*
***B** - A sample CYLINDER_END location for an Auto Cylinder*



***A** - A sample CYLINDER_START location for a Measured Cylinder*
***B** - A sample CYLINDER_END location for a Measured Cylinder*

Vector - This defines the vector of the constructed circle and the vector along which the height value is applied. Eight reference vectors are available: CYLINDER_VECTOR, WORKPLANE, ZPLUS, ZMINUS, XPLUS, XMINUS, YPLUS, and YMINUS.

To construct a cylinder circle:

1. Access the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**).
2. From the **Method** list, select the **Cylinder** option.
3. Select a single cylinder feature.

4. Select the reference point from the **Point** list.
5. Select a reference vector from the **Vector** list. The direction on which the cross section is taken is chosen from the **Vector** list.
6. Type a distance into the **Value** box.
7. Click the **Create** button.

The Edit window command line for this option would read:
`CONSTR/CIRCLE,CYLINDER,feat1,HEIGHT,value,REF_POINT = point,REF_VECTOR = vector`

Constructing an Extracted Circle

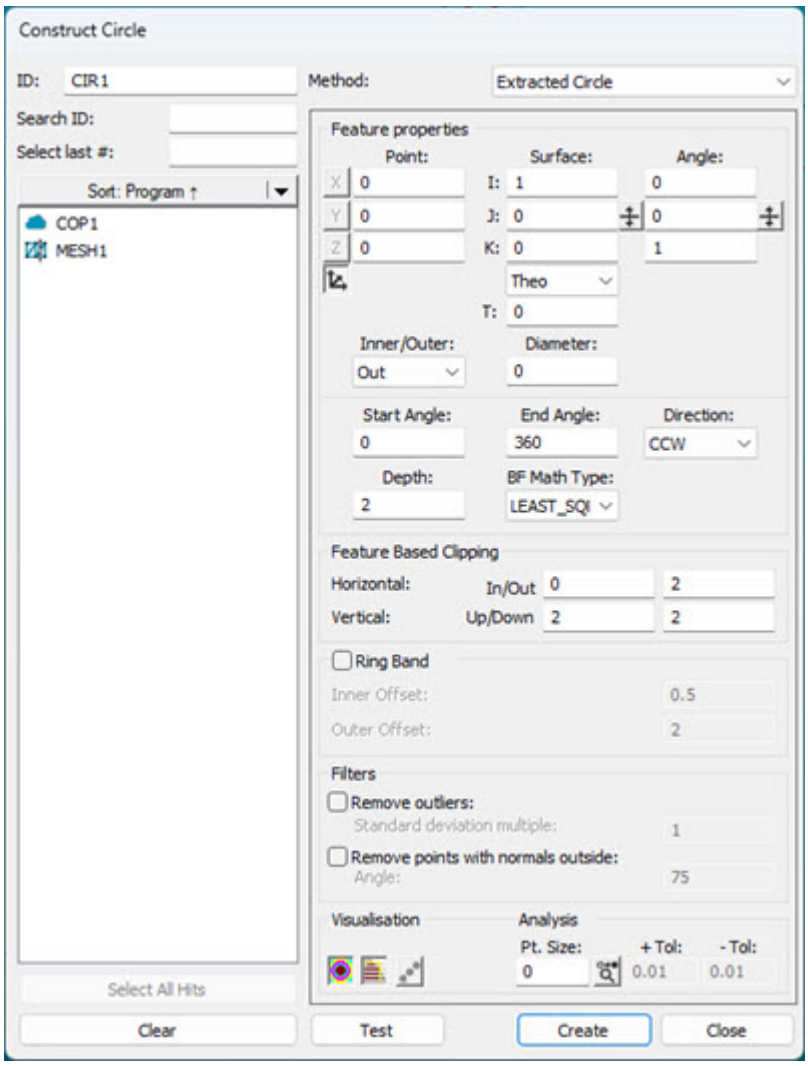
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a circle that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Circle** dialog box (**Insert | Feature | Constructed | Circle**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Circle Dialog Box - Extracted Circle Option

3. From the **Method** list, select the **Extracted Circle** option.

4. From the **Reference** area, select the COP or Mesh you want to use to extract the circle from.

5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.

6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. You can use the **Material Thickness Type** list and the **T** box below it to type in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

   Flip Vectors

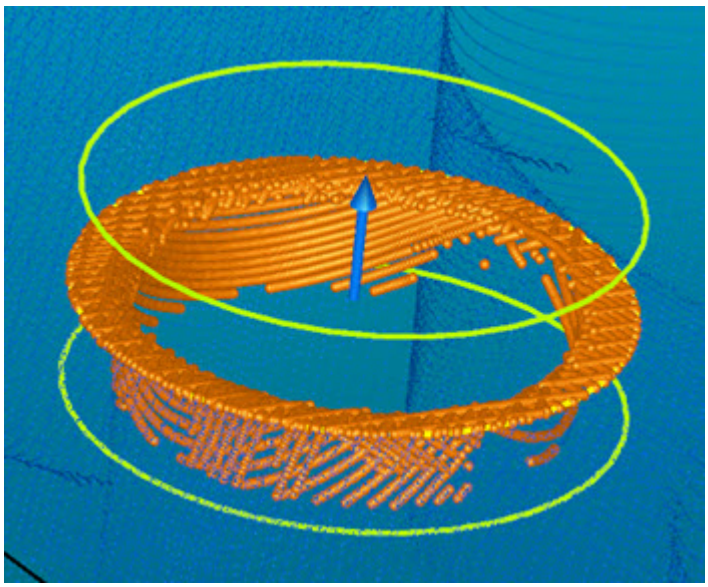
   Polar/Cartesian

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted circle is an inner or outer circle type from the **Inner/ Outer** list.

8. Enter the diameter of the circle in the **Diameter** box.

   PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the cylinder zone that PC-DMIS uses for the extracted circle. The yellow cylinder is the surface. The yellow cylinder is also the horizontal zone and the green cylinder is the vertical zone. The orange points are the candidate points that the extraction considers.



Example of an extracted circle showing the candidate points.

9. From the **Feature Based Clipping** area, define the **Horizontal In/Out** and **Horizontal** values, and the **Vertical Up/Down** and **Vertical** values. This sets the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

10. If you want to define the Ring Band offsets, click the **Ring Band** check box and type the **Inner Offset** and **Outer Offset** values. For details on how the **Ring Band** works, see the "Ring Band Parameters" topic in the PC-DMIS Laser documentation.

11. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

12. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

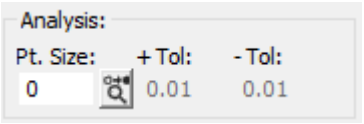
For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

13. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

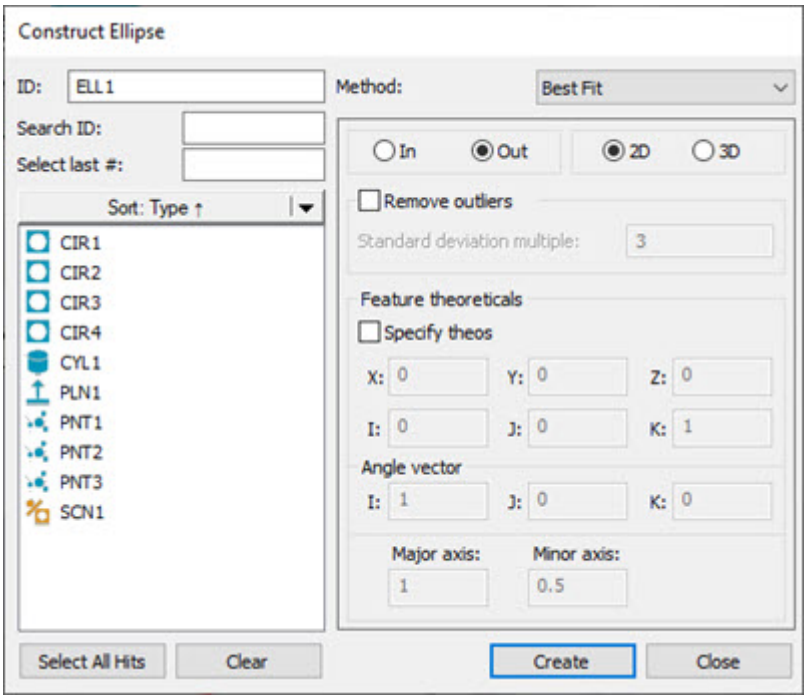
14. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

15. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
CIR1=FEAT/CIRCLE,CARTESIAN,INNER,LEAST_SQR
   THEO/<25.504,15.504,0>,<0,0,1>,8.2
   ACTL/<25.504,15.504,0>,<0,0,1>,1.768
   ANGLE VEC=<1,0,0>
   DEPTH=0,START ANG=0,END ANG=360
   DIRECTION=CCW
   THEO_THICKNESS,0,
   HORIZONTAL CLIPPING=1,VERTICAL CLIPPING=2,INNER
HORIZONTAL CLIPPING=0
   USE OUTLIER REMOVAL=YES,0,
   REMOVE POINTS WITH NORMALS OUTSIDE=YES,0,
   RINGBAND=ON,INNER OFFSET=0,OUTER OFFSET=0
   CONSTR/CIRCLE,EXTRACTED_CIRCLE,REF=COP1
```

Constructing an Ellipse Feature



Construct Ellipse Dialog Box

There are several ways to construct an ellipse using PC-DMIS. The following table lists the various types of constructed ellipses, along with their necessary inputs. Some features may require no inputs while others may require three inputs or more. The term "Any" in the following table indicates that the construction can take any type of feature as input for construction. PC-DMIS allows you to select the features in any order.

Construct Feature Type	Symbol in the Edit Window	Number of Input Features	Feature #1:	Feature #2:	Comments
Auto Ellipse	-	-	-	-	See "Auto Ellipse Construction".
Best fit Ellipse	BF	At least 4 inputs are needed or a scan or set with at least 4 points.	-	-	Constructs a best fit ellipse using the given inputs. See the Note below for recommended inputs.
Best fit with Recompensate Ellipse	BFRE	At least 4 inputs are needed, with one input being a point or a scan or set with at least 4 points.	-	-	Constructs a best fit ellipse using the given inputs. See the Note below for recommended inputs.
Intersection	INT	2	Cylinder	Plane	Constructs an ellipse at the intersection of the input features.
Cast Ellipse	CAST	1	Any	-	Constructs an ellipse at the centroid of the input feature.
Project Ellipse	PROJ	1 or 2	Any	Plane	One input feature will project the ellipse to the workplane.
Reverse Ellipse	REV	1	Ellipse	-	Constructs an ellipse with a reversed vector.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

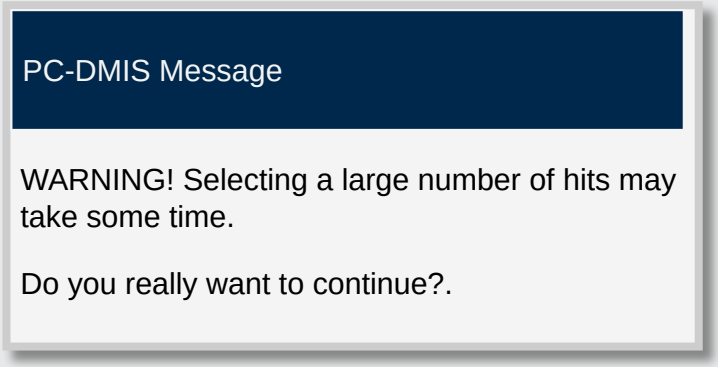
"Cannot construct [feature]. Combination of input features not accepted."

To construct an ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select the method of the constructed ellipse. The available options are:
    - Auto
    - Best Fit
    - BF Recomp
    - Intersection
    - Cast
    - Projection
    - Reverse

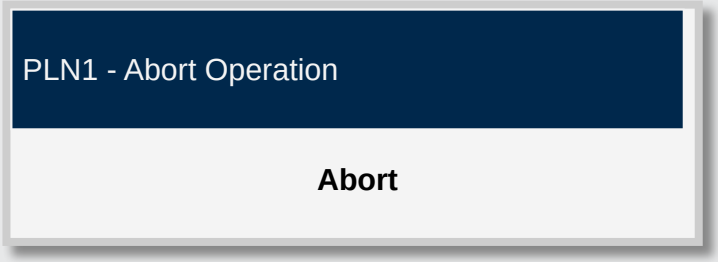
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

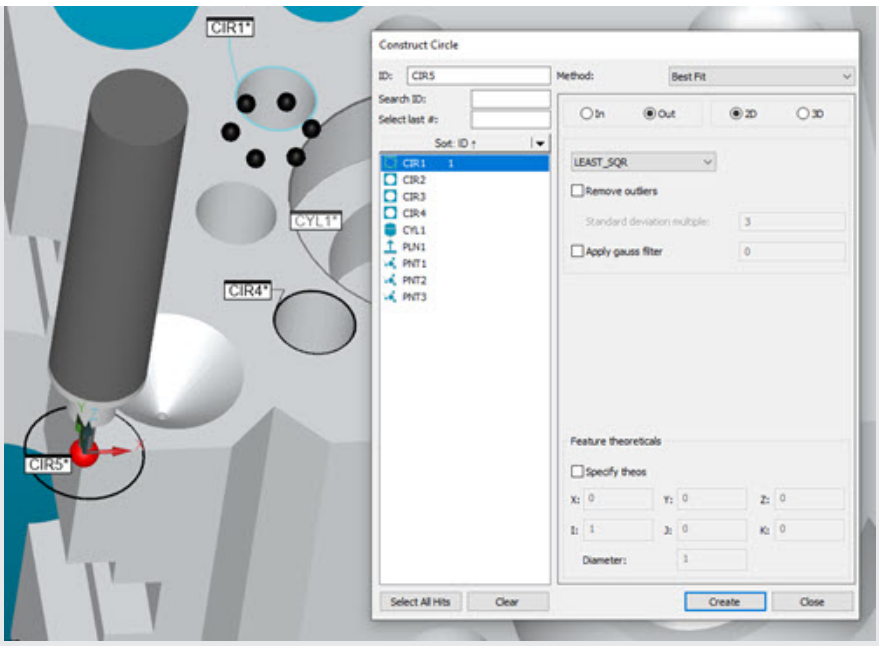


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

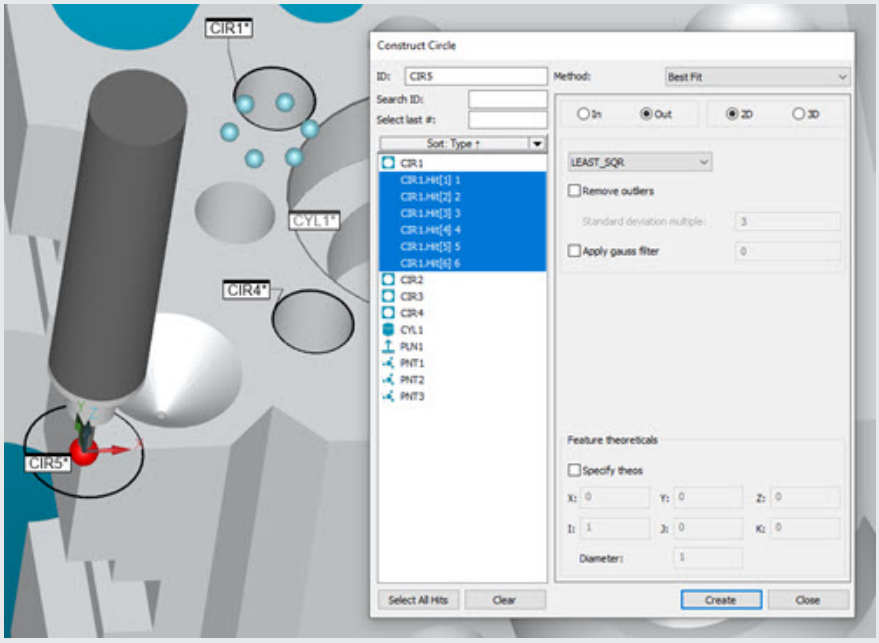
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

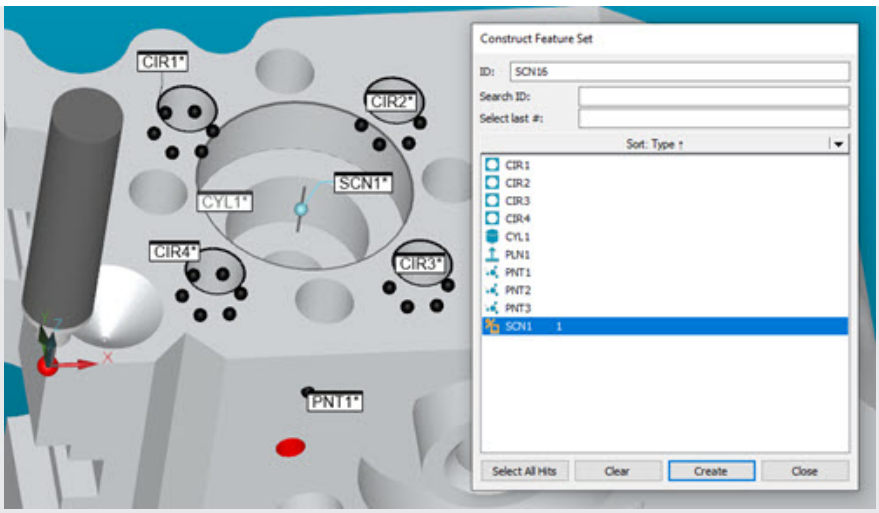
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. From the **Feature** list, use the table above to select the features based on the selected method.
4. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.
5. Some ellipse types have additional options or items that appear in the dialog box. Select or use these options as needed.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for a sample Ellipse Construction would read:

```
 feature_name=FEAT/Ellipse,TOG1,TOG4
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,major diam,minor
diam,i angle vec, j angle vec, k angle vec
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,major diam,minor
diam,i angle vec,j angle vec,k angle vec
CONSTR/TOG2,TOG3,TOG5,feat_1,feat_2, ...
```

The actual Edit report displays in all capital letters.

Auto is the default method of construction. For information on the Auto method, see "Auto Ellipse Construction".

TOG1= POLR or RECT

TOG2 = ELLIPSE

TOG3 = BF / BFRE / CAST / INT / PROJ / REV

TOG4 = IN / OUT

TOG5 = 2D / 3D (This appears only if TOG3 reads BF or BFRE)

The first three lines in the Edit window will be the same for constructed ellipses. The fourth line will be slightly different according to the type of feature you are constructing. You can switch between the different types of ellipses by placing the cursor on *TOG3* and pressing F7 and F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

The following topics describe the available options for constructing an ellipse:

In / Out Ellipse

The **In** and **Out** options tell PC-DMIS whether to construct the ellipse as an internal or an external ellipse.

- If you select **In**, PC-DMIS constructs the ellipse as an internal ellipse.
- If you select **Out**, PC-DMIS constructs the ellipse as an external ellipse or pin.

2D / 3D Ellipse

The **2D** and **3D** options specify whether PC-DMIS constructs the feature as a 2D or 3D ellipse. These options become available when you select the **Best Fit** or **BF Recomp** option.

- If you select **2D**, PC-DMIS constructs the ellipse, projecting it onto the workplane.
- If you select **3D**, PC-DMIS constructs a best fit plane from the inputs. PC-DMIS then projects these inputs onto the plane and creates a constructed ellipse from the projected points.

Auto Ellipse Construction

The "Input Feature List" table indicates the type of ellipse that you can construct based on the selected features when you choose the **Auto** option from the **Method** list. The order of the selected features is not important. If you select an incorrect input feature or features, PC-DMIS shows an error message and will not construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select the **Auto** option.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.
4. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
5. Click the **Create** button.

The following lists the constructed features you can create based on the input feature(s) you select:

Input Feature List

Input Feature(s)	Constructs
Any One Set =	Best Fit Ellipse
Any One Ellipse =	Reverse Ellipse
Any One Feature (except Ellipse or Set) =	Cast Ellipse
Plane + Any Feature =	Projected Ellipse
Set + Set =	Best Fit Ellipse
Three or more Features =	Best Fit Ellipse

Constructing a Best Fit or Best Fit Recompensate Ellipse

You can construct a "best fit" ellipse from three or more features. The ellipse lies in the current workplane. PC-DMIS computes a *least squares* ellipse, one for which PC-DMIS minimizes the average squared distance from the data points to the ellipse.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a Best Fit Ellipse or a Best Fit Recompensate Ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. From the **Feature** list, select at least four features, a scan, or a set comprised of at least four points.
4. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.
5. Select the **2D** or **3D** option. For details, see the "2D / 3D Ellipse" topic in this documentation.
6. Select the **Remove outliers** check box if you want to remove outliers based on the distance from the best fit feature. For details, see the "Remove Outliers / Standard Deviation Multiple for a Constructed Ellipse" topic in this documentation.
7. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
8. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/ELLIPSE,BF, feat_1,feat_2,feat_3...
```

(Uses the actual measured points for construction.)

or

```
CONSTR/ELLIPSE,BFRE, feat_1,feat_2,feat_3...
```

(Uses the center of the probe for measurement.)

Remove Outliers / Standard Deviation Multiple for a Constructed Ellipse

For a best fit (BF) or best fit recompensate (BFRE) ellipse, you can choose to remove outliers based on the distance from the best fit feature. This allows the removal of anomalies that arise in the measurement process.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

PC-DMIS first fits an ellipse to the data and then determines which points are outliers based on the standard deviation multiple. It then does the following:

- Recalculates the best fit ellipse with those outliers removed.
- Checks for outliers again.
- Recalculates the best fit ellipse.
- Continues repeating this process until no more outliers exist or until PC-DMIS cannot compute the ellipse (PC-DMIS can't compute the ellipse if there are fewer than 4 data points).

Constructing an Intersect Ellipse

You can construct an ellipse from the intersection of a non-parallel plane and cylinder.

To construct an intersect ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select the **Intersect** option.
3. From the **Feature** list, select the first feature; it can be either a cylinder or a plane.
4. From the **Feature** list, select the second feature.
    - If you selected a cylinder as the first feature, then this must be a plane.
    - If you selected a plane as the first feature, then this must be a cylinder.

5. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.

6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

7. Click the **Create** button. PC-DMIS constructs the ellipse at the intersection of the two features. The constructed ellipse will have the plane's normal vector.

The Edit window command block for a sample ellipse would look something like this:

```
ID      =FEAT/ELLIPSE,CARTESIAN,OUT,NO
         THEO/X,Y,Z,I,J,K
         ACTL/X,Y,Z,I,J,K
         CONSTR/ELLIPSE,INT,feat1,feat2
```

Constructing a Cast Ellipse

You can construct an ellipse by changing any given feature into an ellipse. PC-DMIS constructs the ellipse at the centroid of the input feature. If you use a sheet metal point, the major diameter is the probe diameter. For some sheet metal features (such as slots and notches), PC-DMIS uses the width as the major diameter. For features that do not have a width (lines, points, etc.), PC-DMIS uses a value that is four times the probe diameter. The minor diameter is the length of the input feature. For features that do not have a length (points, circles, etc.) PC-DMIS uses a default length of 1.

You can modify the major and minor diameters of the ellipse; this changes the ellipse from DEPENDENT to INDEPENDENT. This means that when you execute the ellipse, the major and minor diameters do not change based on the input feature but are independent of the input feature, while the position and vector remain dependent on the input feature. This allows you to have control over the diameters in cases where the input feature doesn’t really have a diameter, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field you may change.

PC-DMIS uses these diameter values for all calculations instead of the default diameter values described above.

To construct a cast ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).

2. From the **Method** list, select the **Cast** option.

3. From the **Feature** list, select at least one feature of any type.

4. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/ELLIPSE,CAST, feat_1 ,(DEPENDENT | INDEPENDENT)
```

Constructing a Projected Ellipse

You can project an ellipse onto a plane. PC-DMIS projects the centroid of the given feature onto the plane, creating an ellipse. If there is only one input feature, the projection is to the current workplane. The major diameter of the projected ellipse is the width of the projected feature, or the probe's diameter (for features without a defined width). The minor diameter is the length of the input feature, or unit 1 (for features without a length defined).

To construct a projected ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select a feature of any type.

   You can also select a second feature. It must be a plane.

4. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/ELLIPSE,PROJ, feat_1,(feat_2)
```

Changing the Direction of an Ellipse

You can construct an ellipse with a reversed vector.

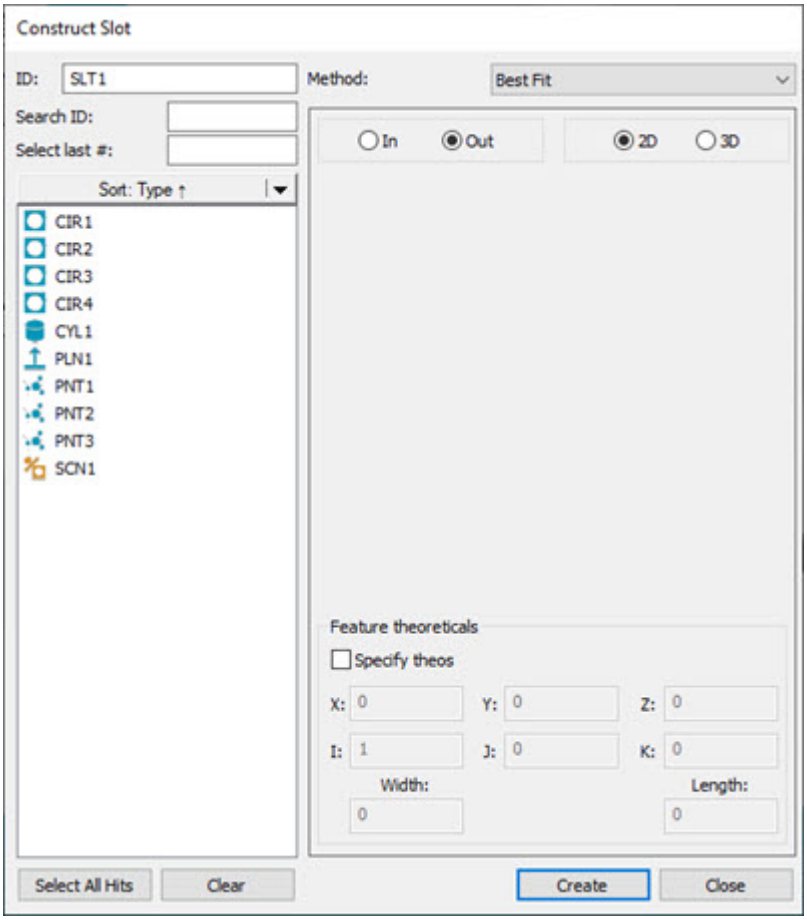
To change the direction of a ellipse:

1. Open the **Construct Ellipse** dialog box (**Insert | Feature | Constructed | Ellipse**).
2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select one feature. It must be an ellipse.
4. Select the **In** or **Out** option. For details, see the "In / Out Ellipse" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/ELLIPSE,REV,feat_1
```

Constructing a Round Slot Feature



Construct Slot Dialog Box

There are two round slot types in PC-DMIS:

- A round slot created from two circles (**Circles** option)
- A round slot created from four or more inputs (**Best Fit** or **BF Recomp** option).

The table below shows the inputs for the slot and the editor definitions.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Comments
Circles	CIRCLES	2	Circle	Circle	Constructs a slot in the plane of the first circle from center to center.
Best Fit Round Slot	BF	4 or more	-	-	Constructs a Best Fit slot using the given inputs. See the Note below for recommended inputs.
Best Fit Recompensate Round Slot	BFRE	4 or more	-	-	Constructs a Best Fit Recompensate slot using the given inputs. See the Note below for recommended inputs.
Projected Round Slot	PROJ	2	Slot	Plane	Constructs a Projected Round Slot onto the plane.
Cast Round Slot	CAST	1	Any	-	Constructs a round slot at the centroid of the input feature.
Extracted Round Slot	EXTRACTED_ROUND_SLOT	1	COP or Mesh	-	Constructs an extracted round slot from the COP or Mesh object at the specified diameter or height.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

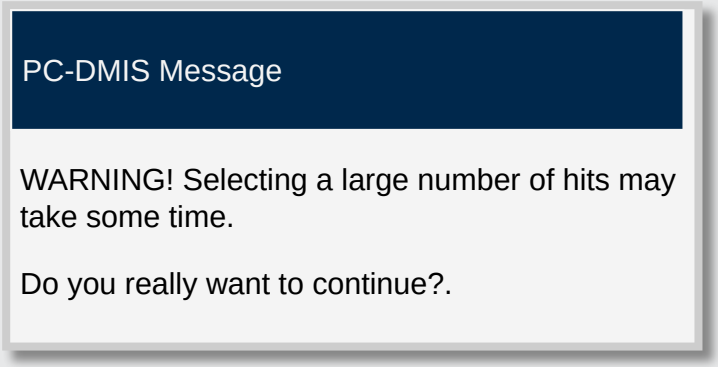
"Cannot construct [feature]. Combination of input features not accepted."

To construct a round slot:

1. Open the **Construct Round Slot** dialog box (**Insert | Feature | Constructed | Round Slot**).
2. From the **Method** list, select the method of the constructed round slot. The available options are:
    - Circles
    - Best Fit
    - BF Recomp
    - Projection
    - Cast
    - Extracted Round Slot

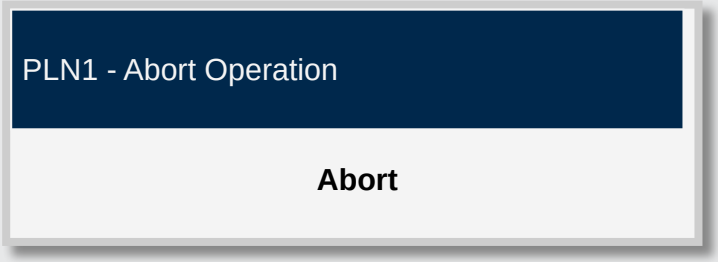
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

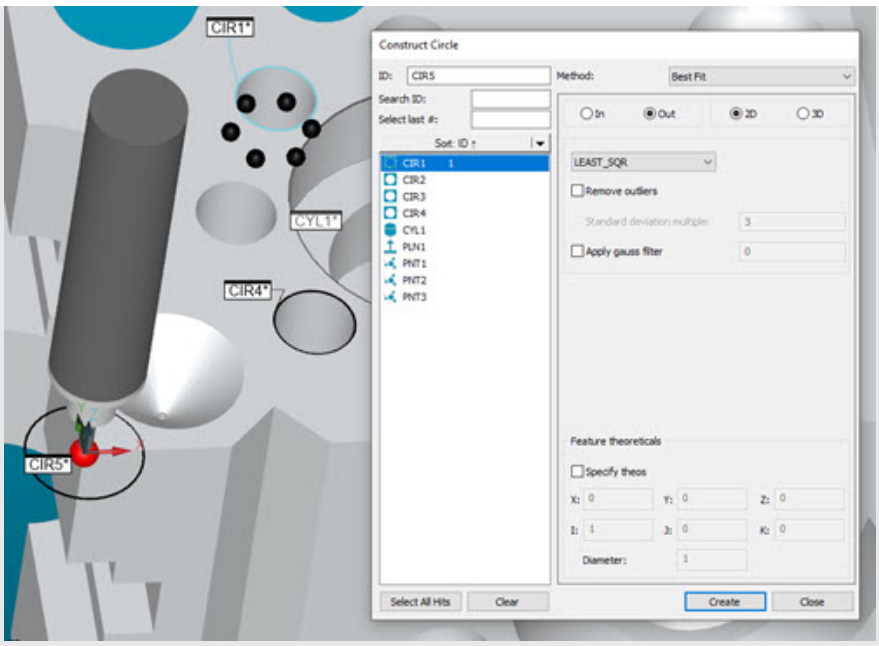


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

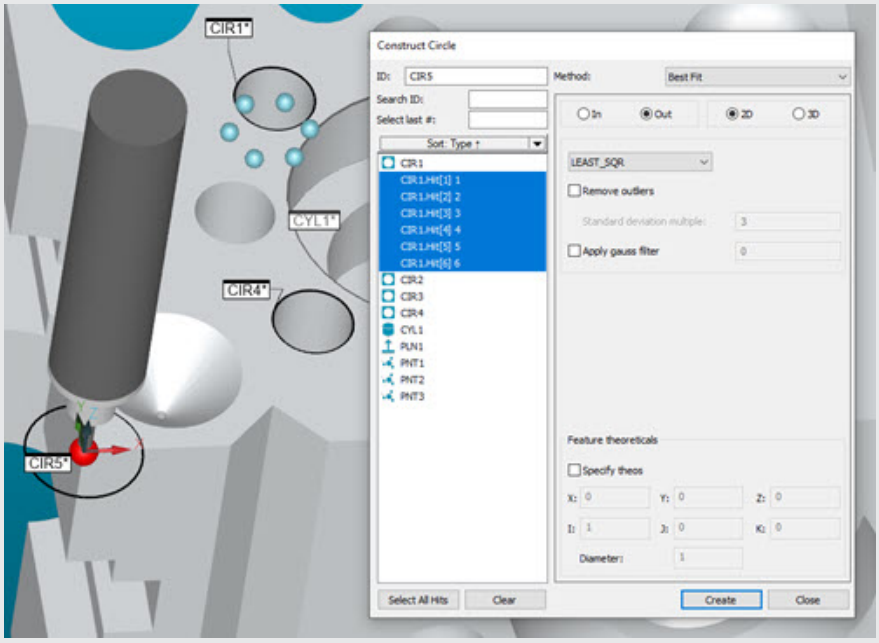
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

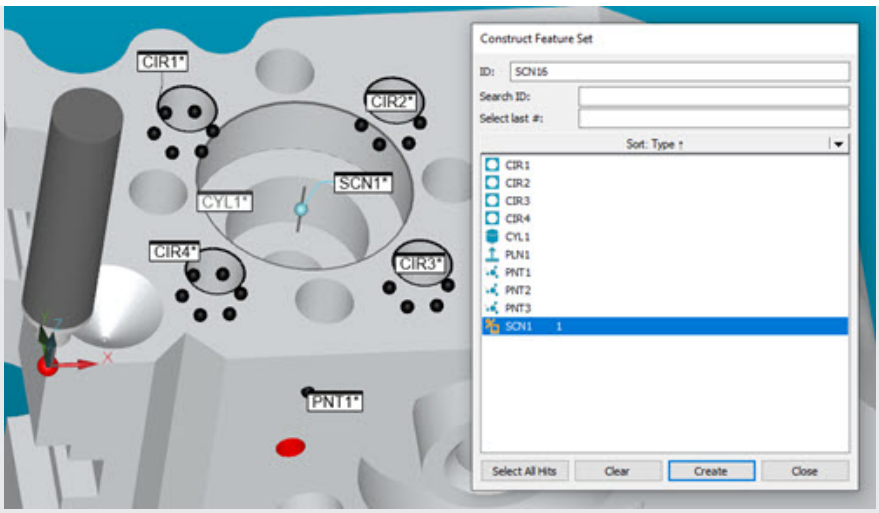
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. Using the table above, select the features from the **Feature** list based on the selected method.
4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. If you chose one of the best fit methods, select either the **2D** or **3D** option. For details, see the "2D / 3D Slot" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for a sample Slot Construction would read:

```
feature_name=FEAT/SLOT,TOG1,TOG2,TOG3
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,width,length
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,width,length
CONSTR/TOG4,TOG5,TOG6,TOG7,feat_1,feat_2, …
```

The actual report displays in all capital letters.

TOG1 = CARTESIAN or POLAR

TOG2 = OUT or IN

TOG3 = YES or NO

TOG4 = SLOT (or other construction type)

TOG5 = ROUND or SQUARE

TOG6 = CIRCLES or BF or BFRE or PROJ

TOG7 = 2D or 3D (This appears only if TOG6 reads BF or BFRE)

In / Out Slot

The **In** and **Out** options tell PC-DMIS whether to construct the slot as an internal or an external slot.

- If you select **In**, PC-DMIS constructs the slot as an internal slot.
- If you select **Out**, PC-DMIS constructs the slot as an external slot.

2D / 3D Slot

The **2D** and **3D** options tell PC-DMIS whether to construct the feature as a 2D or 3D slot. These options become available if you select the **Best Fit** or **BF Recomp** option.

- If you select **2D**, PC-DMIS constructs the slot, projecting it onto the workplane.
- If you select **3D**, PC-DMIS constructs a best fit plane from the inputs. Those inputs are then projected to the plane, and a constructed slot is created from the projected points.

Constructing a Circle Slot

PC-DMIS defines the round slot constructed from two circles by the first circle selected. PC-DMIS constructs the slot in the same plane as the first circle. PC-DMIS also determines the width of the slot by the diameter of the first circle. The software only uses the second circle to determine the length of the slot. The length is the distance from the center of the first circle to center of the second plus the diameter of the first circle.

If the two input circles are not coplanar, PC-DMIS projects the center of the second circle perpendicularly onto the plane of the first circle. The software then calculates the distance from the center of the first circle to the projected center of the second circle.

To construct a round slot from circles:

1. Open the **Construct Slot** dialog box (**Insert | Feature | Constructed | Slot**).
2. From the **Method** list, select the **Circles** option.
3. From the **Feature** list, select two circle features for inputs.
4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for the Round Slot would read:
```
CONSTR/SLOT,CIRCLES,feat_1,feat_2
```

Constructing a Best Fit or Best Fit Recompensate Slot

The Best Fit (BF) and the Best Fit Recompensate (BFRE) are constructed from five or more features. The constructed slots vector is normal (perpendicular) to the workplane. The BFRE construction uses the ball center combined with the probe radius to compute the slot. The compensation is an integral part of the fitting. The BF construction compensates the measured points prior to fitting.

The height of the slot from the workplane is the average of all the input features.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a Best Fit or Best Fit Recompensate slot:

1. Open the **Construct Slot** dialog box (**Insert | Feature | Constructed | Slot**).
2. From the **Method** list, select either the **Best Fit** or **Best Fit Recompensate** option (shown as **Best Fit** and **BF Recomp**).
3. From the **Feature** list, select at least four features. They can be any feature type.

4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. Select either the **2D** or **3D** option. For details, see the "2D / 3D Slot" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for the Best Fit or Best Fit Recompensate Slot would read:
```
CONSTR/SLOT,BF (or BFRE),feat_1,feat_2, ...
```

Constructing a Projected Slot

Similar to a Projected Circle, PC-DMIS can create a slot feature projected onto a specified plane.

To construct a projected slot:

1. Open the **Construct Round Slot** dialog box (**Insert | Feature | Constructed | Round Slot**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select two features. The first feature should be a slot. The second feature should be a plane.
4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button. The slot is constructed projected onto the specified plane.

The Edit window command line for the projected slot would read:
```
CONSTR/SLOT,ROUND,PROJ,feat_1,feat_2, ...
```

Constructing a Cast Round Slot

Similar to a Cast Circle, you can construct a Cast Round Slot by changing any given feature into a slot. PC-DMIS uses the Cast option from the **Construct Round Slot** dialog box to construct the slot at the centroid of the input feature.

Guidelines when you construct a cast slot:

- The software uses the diameter or width of the input feature as the width of the slot.
- The software uses the length of the input feature as the length of the slot.
- If the width or diameter of the input feature is 0 (zero), the width of the slot is the probe's diameter.
- If the length of the input feature is 0 (zero), the length of the slot is set to three time the probe's diameter.
- The software uses the vector of a line feature as the slot's angle vector; the software uses the vector of any other feature types as the slot's surface vector.

To construct a cast round slot:

1. Open the **Construct Round Slot** dialog box (**Insert | Feature | Constructed | Round Slot**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.
4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for the Cast Round Slot would read:

```
CONSTR/SLOT,ROUND,CAST,feat_1,DEPENDENT
```

Constructing an Extracted Round Slot

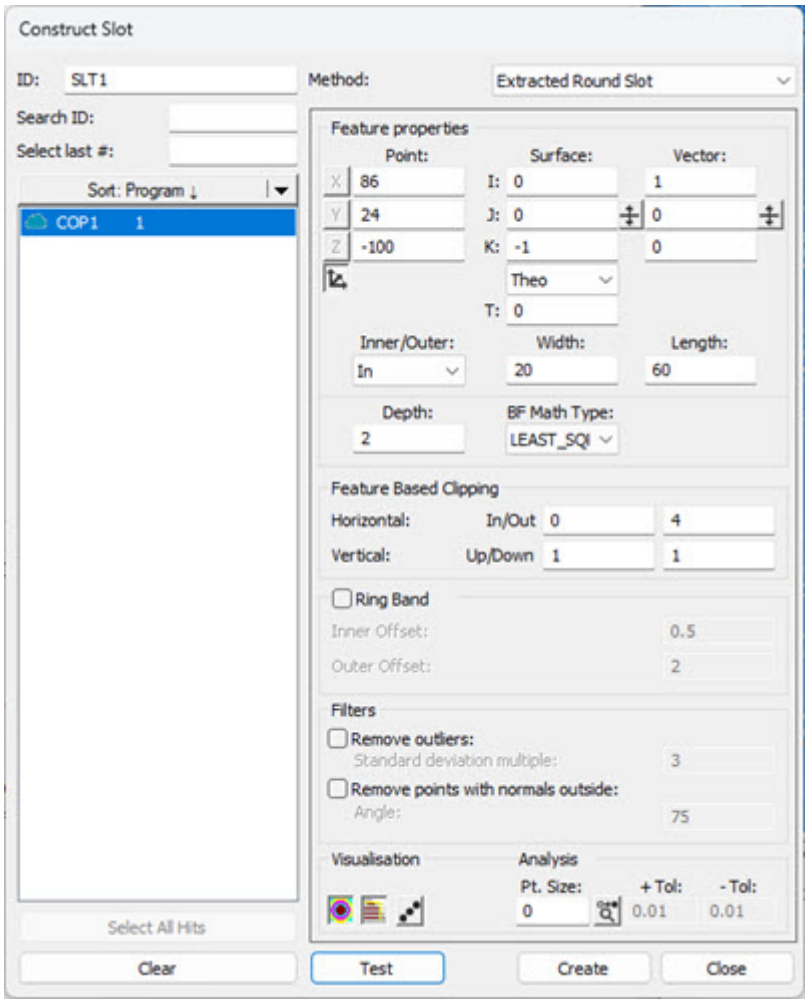
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a round slot that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Slot** dialog box (**Insert | Feature | Constructed | Round Slot**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Slot Dialog Box - Extracted Round Slot Option

3. From the **Method** list, select the **Extracted Round Slot** option.
4. From the **Reference** area, select the COP or Mesh that you want to use to extract the round slot from.
5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.
6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. In the **Angle** section, enter the corresponding vector angle values. You can use the **Material Thickness Type** list and the **T** box below it to type in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

Flip Vectors

Polar/Cartesian

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted round slot is an inner or outer round slot type from the **Inner/Outer** list.

8. Enter the width and length of the round slot in the respective boxes. The **Width** value defines the width of the round slot, and the **Length** value defines the length or the longest side of the round slot.

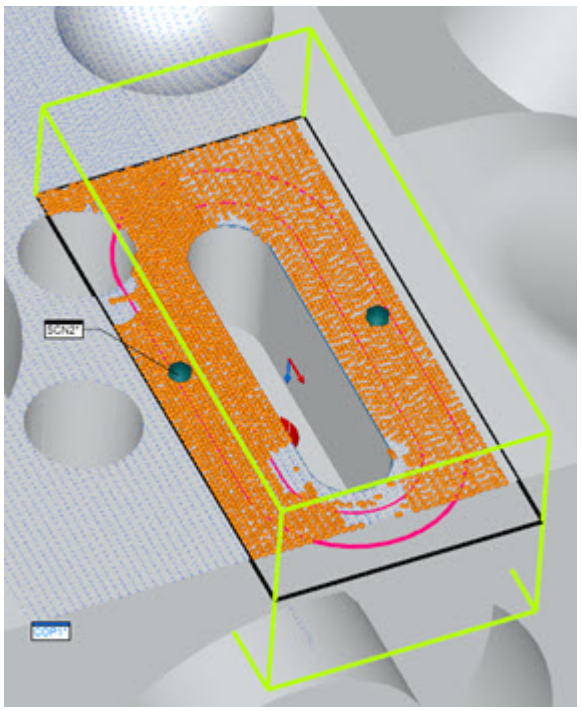
9. Enter the **Depth** value. This parameter controls the location of the laser focal point in relation to the round slot's outside diameter (outer round slots) or the round slot's center axis (inner round slots). This allows you to control how the laser stripes fall on the round slot's surface by specifying how far or close the laser is to the round slot's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height using data found at the lowest possible depth from the surface plane. A depth of any other value causes the software to perform the calculation at that depth.

10. From the **BF Math Type** list, select the type of Best Fit algorithm to use to construct the round slot. The available options are:

    LEAST_SQR
    Least Squares - This calculation type provides a method of fitting in which the average squared radial distance from the data points to the round slot is minimized. The square root of this quantity is the Root Mean Square (RMS) distance. Since the RMS distance is based on an average, some points may be further than the RMS distance from the computed round slot.

    MAX_INSC
    Maximum Inscribed - For Inner slots, this calculation type generates a slot with the largest possible footprint that lies within the data. PC-DMIS first computes a Minimum Circumscribed slot and requires that the center of the Maximum Inscribed slot lies within it. Do not use this calculation type for arcs of less than 90 degrees.

    MIN_CIRCSC
    Minimum Circumscribed - For Outer slots, this calculation type generates a slot with the smallest possible footprint that encloses the input data (or input features). This option could be used when measuring a stud that would fit into a mating feature. The resulting feature would be the smallest slot into which the mating feature would fit. Do not use this calculation type for arcs of less than 180 degrees.



Example of a constructed extracted round slot showing the candidate points.

11. From the **Feature Based Clipping** area, define the **Horizontal In/Out** and **Horizontal** values, and the **Vertical Up/Down** and **Vertical** values. These values set the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

12. If you want to define the Ring Band offsets, click the **Ring Band** check box and type the **Inner Offset** and **Outer Offset** values. For details on how the **Ring Band** works, see the "Ring Band Parameters" topic in the PC-DMIS Laser documentation.

13. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

14. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

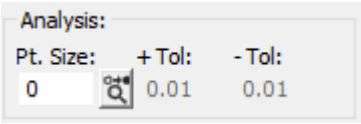
For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

15. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

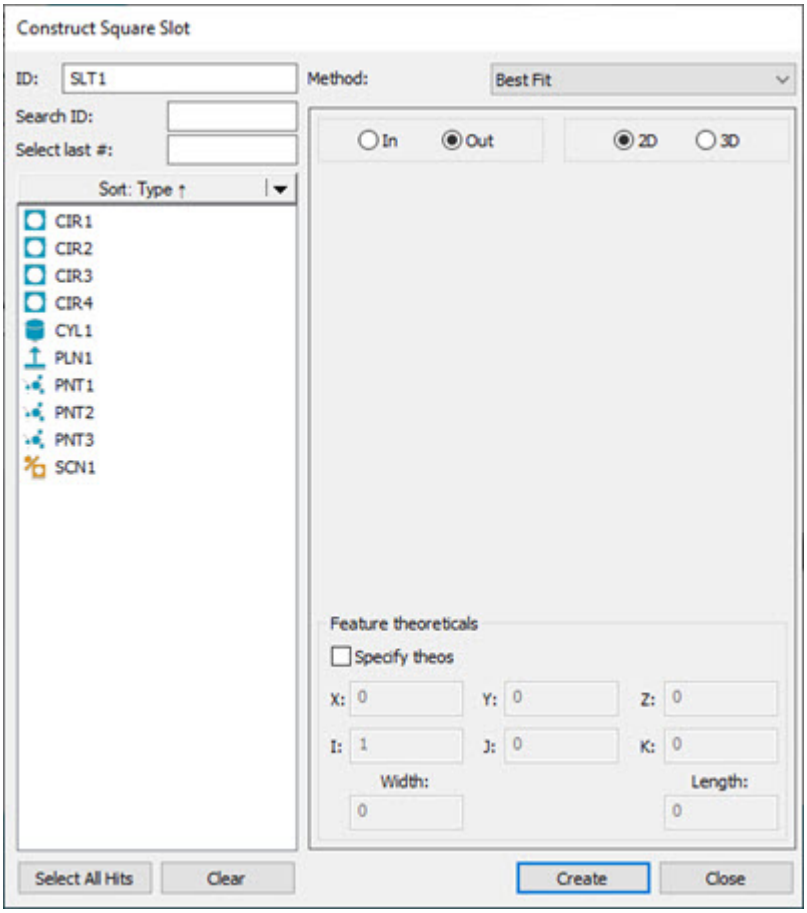
16. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

17. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
SLT1=FEAT/ROUND_SLOT,CARTESIAN,INNER,LEAST_SQR
   THEO/<70,15.9,0>,<0,0,-1>,<0,-1,0>,6,26
   MEAS/<70,15.9,0>,<0,0,-1>,<0,-1,0>,6,26
   DEPTH=0
   THEO_THICKNESS,0
   HORIZONTAL CLIPPING=7,VERTICAL CLIPPING=9,INNER
HORIZONTAL CLIPPING=0
   RINGBAND=ON,INNER OFFSET=2,OUTER OFFSET=5
   USE OUTLIER REMOVAL=ON,0
   REMOVE POINTS WITH NORMALS OUTSIDE=ON,1
   CONSTR/SLOT,ROUND,EXTRACTED,COP1
```

Constructing a Square Slot Feature



Construct Square Slot Dialog Box

The table below shows the inputs for the slot and the editor definitions.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Comments
Best Fit Square Slot	BF	5 or more	-	-	Constructs a Best Fit slot using the given inputs. See the Note below for recommended inputs.
Best Fit Recompensate Square Slot	BFRE	5 or more	-	-	Constructs a Best Fit Recompensate slot using the given inputs. See the Note below for recommended inputs.
Projected Square Slot	PROJ	2	Slot	Plane	Constructs a Projected Square Slot onto the plane.
Cast Square Slot	CAST	1	Any	-	Constructs a Square Slot at the centroid of the input feature.
Extracted Square Slot	EXTRACTED_SQUARE_SLOT	1	COP or Mesh	-	Constructs an extracted square slot from the COP or Mesh object at the specified diameter or height.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

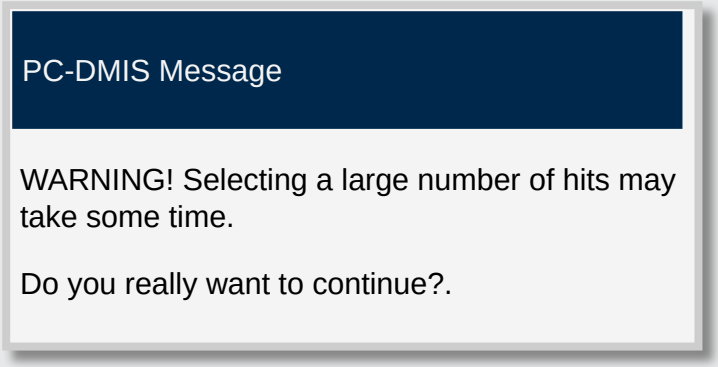
"Cannot construct [feature]. Combination of input features not accepted."

To construct a square slot:

1. Open the **Construct Square Slot** dialog box (**Insert | Feature | Constructed | Square Slot**).
2. From the **Method** list, select the method of the constructed square slot. The available options are:
    - Best Fit
    - BF Recomp
    - Projection
    - Cast
    - Extracted Square Slot

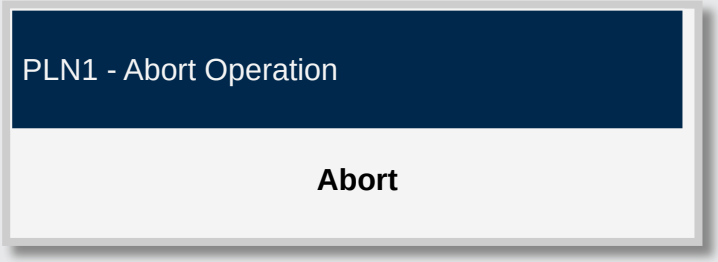
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

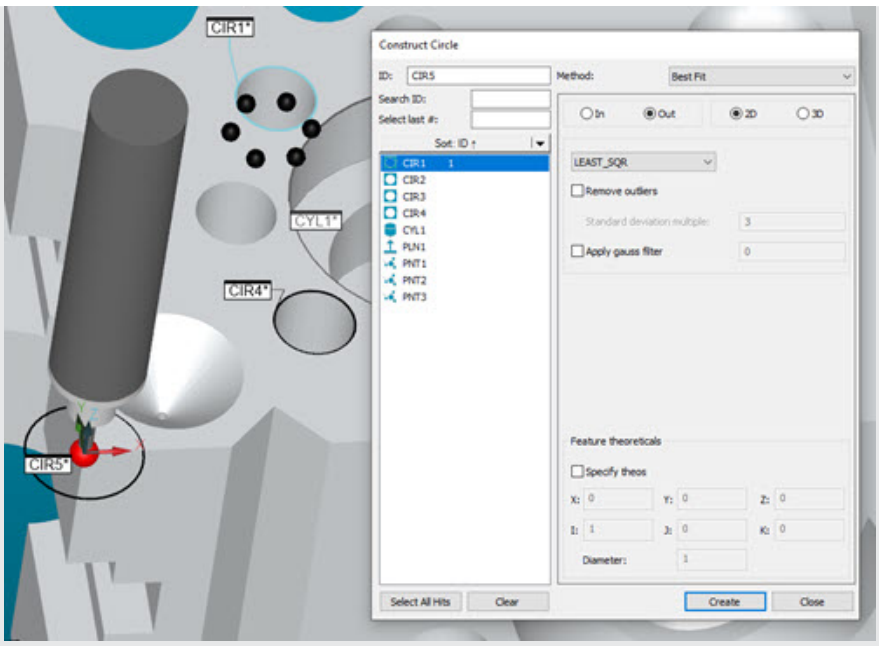


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

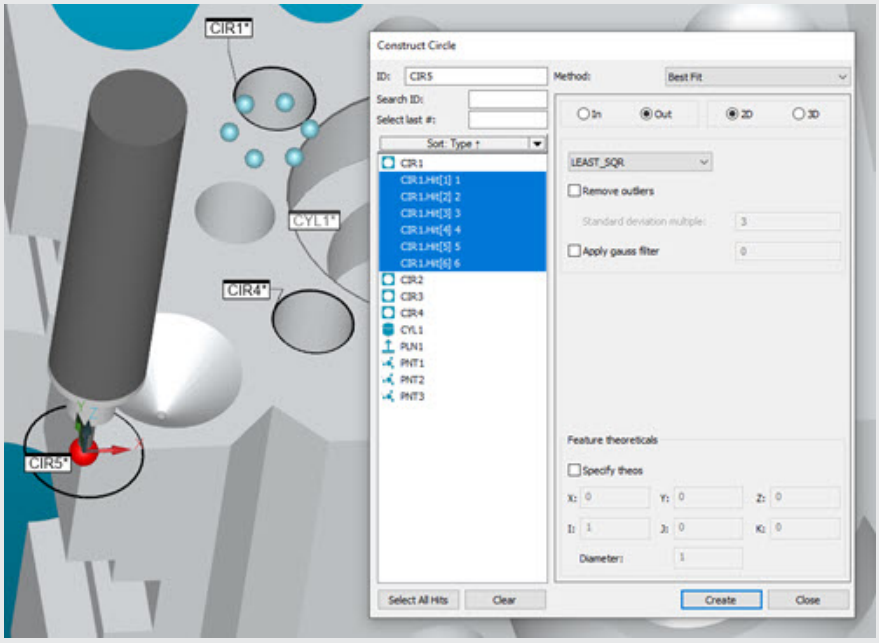
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

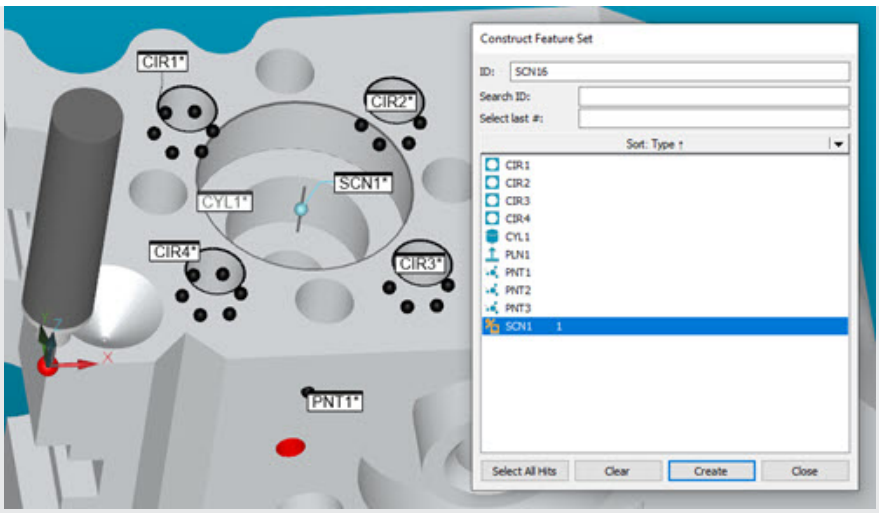
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.

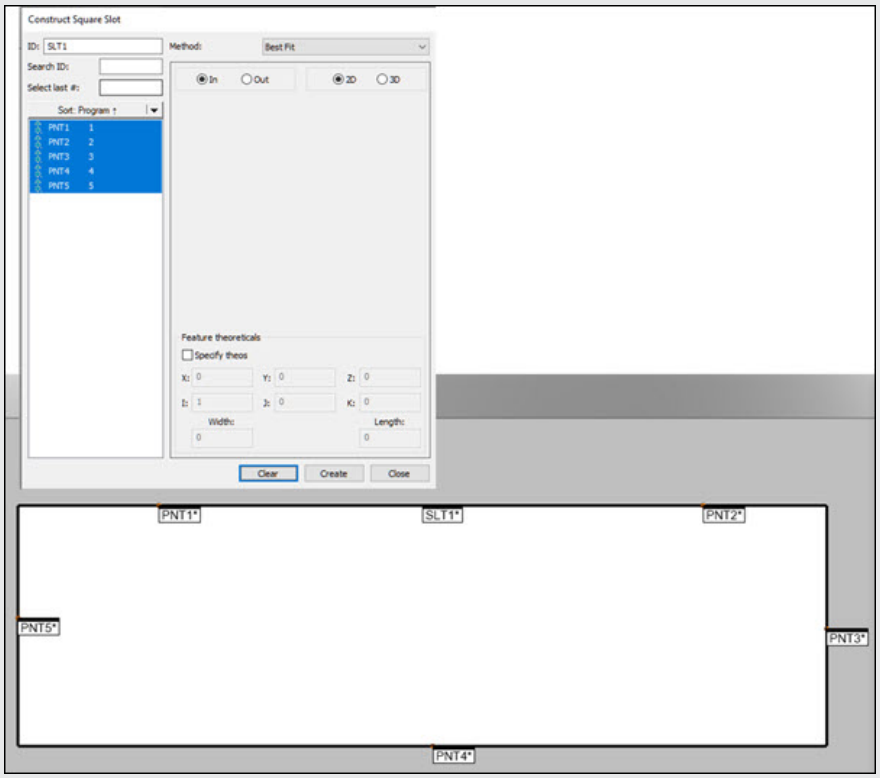


Constructed feature created based on the selected items from the Feature list.

3. Using the table above, select the features from the **Feature** list based on the selected method.

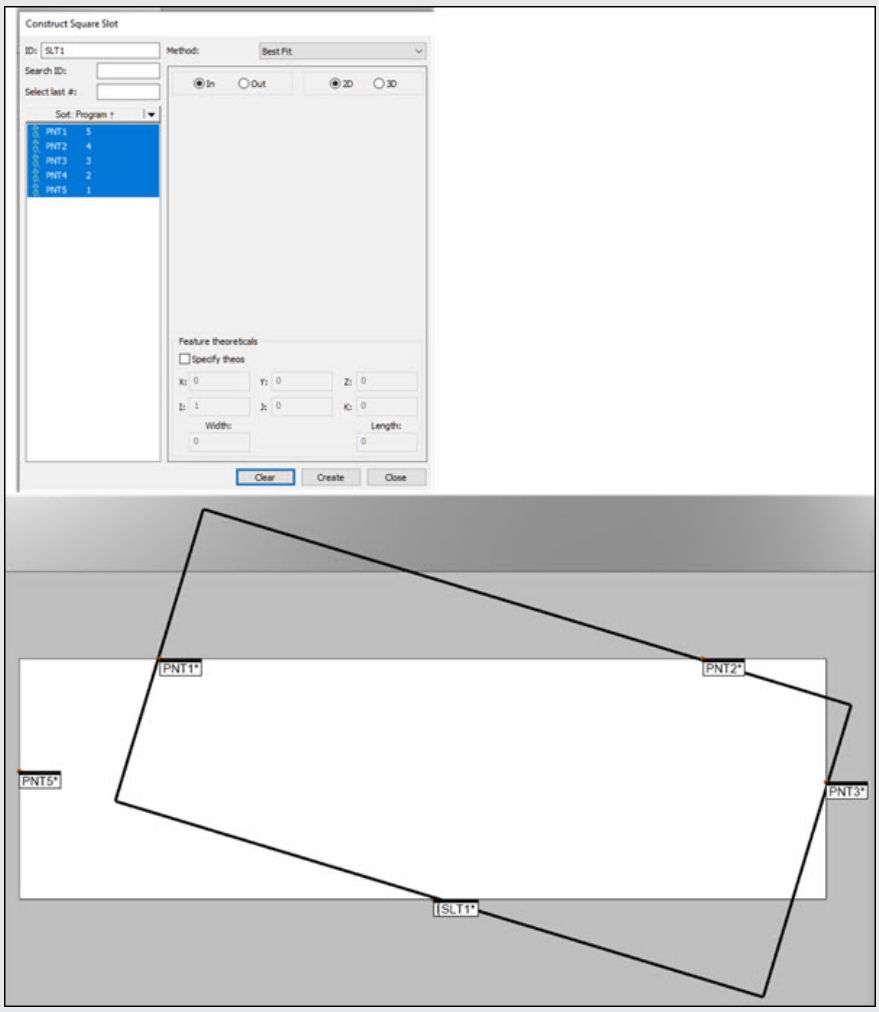
Remember, the order you select the features from the list determines the orientation (CW or CCW) of the Constructed Square Slot. See the examples below.

Example of a Correct Order of Selection



Example showing the correct order of feature selection to orient the slot from the first and second points and resolves the correct length and width.

Example of an Incorrect Order of Selection



Example of an incorrect order of selection that resolves the wrong orientation and incorrect length and width.

4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in the PC-DMIS Core documentation.

5. If you chose one of the best fit methods, select either the **2D** or **3D** option. For details, see the "2D / 3D Slot" topic in the PC-DMIS Core documentation.

6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

7. Click the **Create** button.

The Edit window command line for a sample slot construction would read:

```
feature_name=FEAT/SLOT,TOG1,TOG2,TOG3
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,width,length
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,width,length
CONSTR/TOG4,TOG5,TOG6,TOG7,feat_1,feat_2, …
```

The actual report displays in all capital letters.

TOG1 = CARTESIAN or POLAR

TOG2 = OUT or IN

TOG3 = YES or NO

TOG4 = SLOT (or other construction type)

TOG5 = ROUND or SQUARE

TOG6 = BF or BFRE or PROJ

TOG7 = 2D or 3D (This appears only if TOG6 reads BF or BFRE)

In / Out Square Slot

The **In** and **Out** options tell PC-DMIS whether to construct the slot as an internal or an external slot.

- If you select **In**, PC-DMIS constructs the slot as an internal slot.
- If you select **Out**, PC-DMIS constructs the slot as an external slot.

2D / 3D Square Slot

The **2D** and **3D** options tell PC-DMIS whether to construct the feature as a 2D or 3D slot. These options become available if you select the **Best Fit** or **BF Recomp** option.

- If you select **2D**, PC-DMIS constructs the slot, projecting it onto the workplane.
- If you select **3D**, PC-DMIS constructs a best fit plane from the inputs. Those inputs are then projected to the plane, and a constructed slot is created from the projected points.

Constructing a Best Fit or Best Fit Recompensate Square Slot

The Best Fit (BF) and the Best Fit Recompensate (BFRE) are constructed from five or more features. The constructed slots vector is normal (perpendicular) to the workplane. The BFRE construction uses the ball center combined with the probe radius to compute the slot. The compensation is an integral part of the fitting. The BF construction compensates the measured points prior to fitting.

The height of the slot from the workplane is the average of all the input features.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

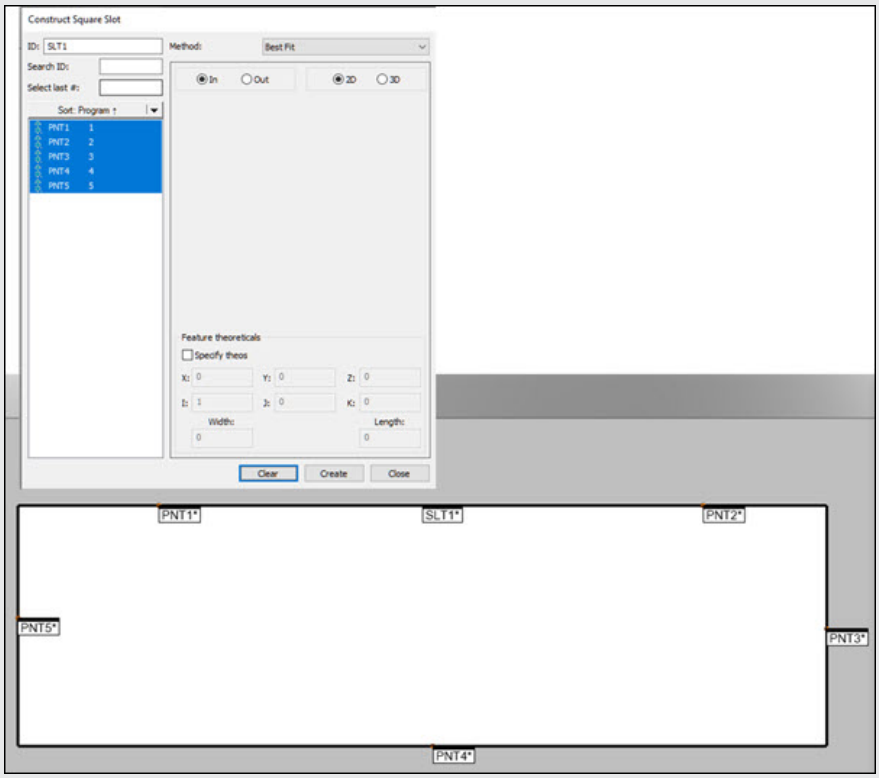
For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a Best Fit or Best Fit Recompensate square slot:

1. Open the **Construct Slot** dialog box (**Insert | Feature | Constructed | Slot**).
2. From the **Method** list, select either the **Best Fit** or **Best Fit Recompensate** option (shown as **Best Fit** and **BF Recomp**).
3. From the **Feature** list, select at least five features.

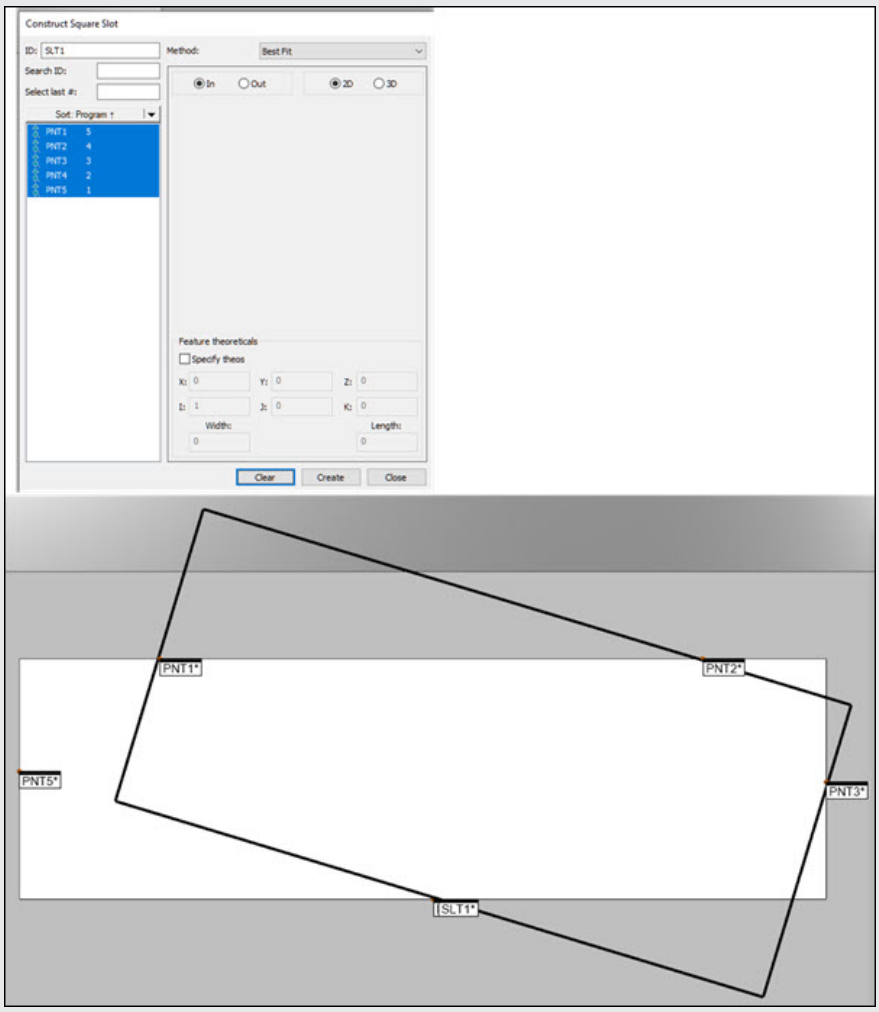
Remember, the order you select the features from the list determines the orientation (CW or CCW) of the Constructed Square Slot. See the examples below.

Example of a Correct Order of Selection



Example showing the correct order of feature selection to orient the slot from the first and second points and resolves the correct length and width.

Example of an Incorrect Order of Selection



Example of an incorrect order of selection that resolves the wrong orientation and incorrect length and width.

4. Select the **In** or **Out** option. For details, see the "In / Out Square Slot" topic in this documentation.

5. Select either the **2D** or **3D** option. For details, see the "2D / 3D Square Slot" topic in the PC-DMIS Core documentation.

6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

7. Click the **Create** button.

The Edit window command line for the Best Fit or Best Fit Recompensate Slot would read:

```
CONSTR/SLOT,BF (or BFRE),feat_1,feat_2, ...
```

Constructing a Projected Square Slot

Similar to a Projected Circle, PC-DMIS can create a slot feature projected onto a specified plane.

To construct a projected square slot:

1. Open the **Construct Square Slot** dialog box (**Insert | Feature | Constructed | Square Slot**).
2. From the **Method** list, select the **Projected** option.
3. From the **Feature** list, select two features. The first should be a slot. The second should be a plane.
4. Select the **In** or **Out** option. For details, see the "In / Out Square Slot" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button. The slot is constructed projected onto the specified plane.

The Edit window command line for the Projected Square Slot would read:

```
CONSTR/SLOT,SQUARE,PROJ,feat_1,feat_2, ...
```

Constructing a Cast Square Slot

Similar to a Cast Circle, you can construct a Cast Square Slot by changing any given feature into a slot. Using the Cast option from the **Construct Square Slot** dialog box, PC-DMIS constructs the slot at the centroid of the input feature.

Guidelines when you construct a cast slot:

- The software uses the diameter or width of the input feature as the width of the slot.
- The software uses the length of the input feature as the length of the slot.
- If the width or diameter of the input feature is 0 (zero), the width of the slot is the probe's diameter.
- If the length of the input feature is 0 (zero), the length of the slot is set to three time the probe's diameter.
- The software uses the vector of a line feature as the slot's angle vector; the software uses the vector of any other feature types as the slot's surface vector.

To construct a Cast Square Slot:

1. Open the **Construct Square Slot** dialog box (**Insert | Feature | Constructed | Square Slot**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.
4. Select the **In** or **Out** option. For details, see the "In / Out Slot" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for the Cast Square Slot would read:

```
CONSTR/SLOT,SQUARE,CAST,feat_1,DEPENDENT
```

Constructing an Extracted Square Slot

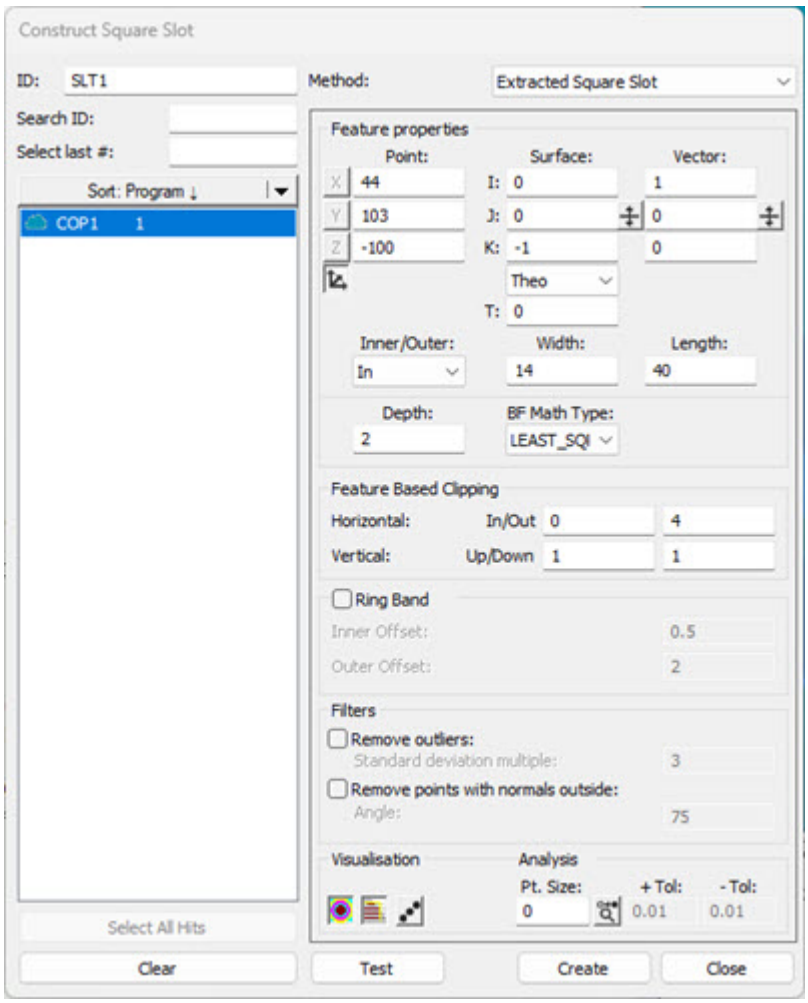
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a square slot that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Square Slot** dialog box (**Insert | Feature | Constructed | Square Slot**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Square Slot dialog box - Extracted Square Slot

3. From the **Method** list, select the **Extracted Square Slot** option.

4. From the **Reference** area, select the COP or Mesh that you want to use to extract the square slot from.

5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.

6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. In the **Angle** section, enter the corresponding vector angle values. You can use the **Material Thickness Type** list and the **T** box below it to type in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

   Flip Vectors

   Polar/Cartesian

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted square slot is an inner or outer square slot type from the **Inner/Outer** list.

8. Enter the width and length of the square slot in the respective boxes. The **Width** value defines the width of the square slot, and the **Length** value defines the length or the longest side of the square slot.

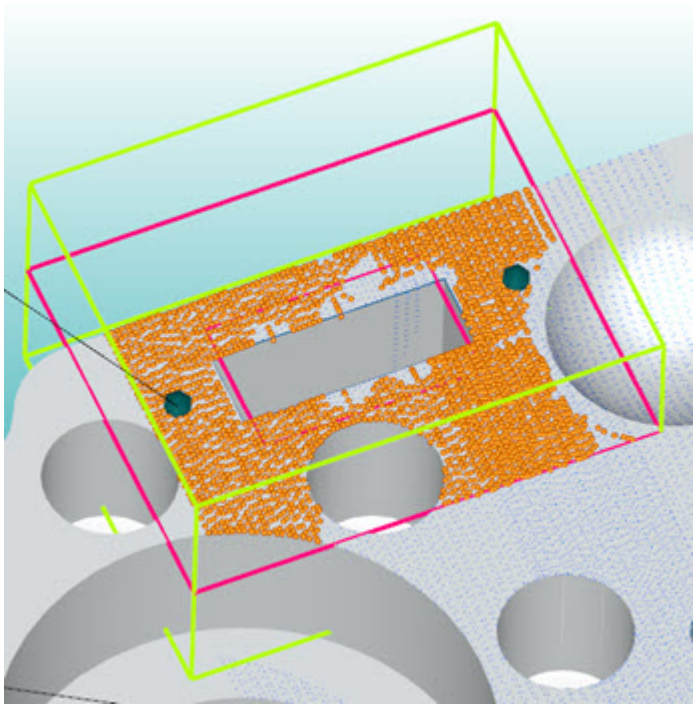
9. Enter the **Depth** value. This parameter controls the location of the laser focal point in relation to the square slot's outside diameter (outer square slots) or the square slot's center axis (inner square slots). This allows you to control how the laser stripes fall on the square slot's surface by specifying how far or close the laser is to the square slot's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height using data found at the lowest possible depth from the surface plane. A depth of any other value causes the software to perform the calculation at that depth.

10. From the **BF Math Type** list, select the type of Best Fit algorithm to use to construct the square slot. The available options are:

    LEAST_SQR
    Least Squares - This calculation type provides a method of fitting in which the average squared radial distance from the data points to the square slot is minimized. The square root of this quantity is the Root Mean Square (RMS) distance. Since the RMS distance is based on an average, some points may be further than the RMS distance from the computed square slot.

    MAX_INSC
    Maximum Inscribed - For Inner slots, this calculation type generates a slot with the largest possible footprint that lies within the data. PC-DMIS first computes a Minimum Circumscribed slot and requires that the center of the Maximum Inscribed slot lies within it. Do not use this calculation type for arcs of less than 90 degrees.

    MIN_CIRCSC
    Minimum Circumscribed - For Outer slots, this calculation type generates a slot with the smallest possible footprint that encloses the input data (or input features). This option could be used when measuring a stud that would fit into a mating feature. The resulting feature would be the smallest slot into which the mating feature would fit. Do not use this calculation type for arcs of less than 180 degrees.



Example of a Constructed Extracted Square Slot showing the candidate points

11. From the **Feature Based Clipping** area, define the **Horizontal In/Out** and **Horizontal** values, and the **Vertical Up/Down** and **Vertical** values. These values set the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

12. If you want to define the Ring Band offsets, click the **Ring Band** check box and enter the **Inner Offset** and **Outer Offset** values. For details on how the **Ring Band** works, see the "Ring Band Parameters" topic in the PC-DMIS Laser documentation.

13. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

14. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

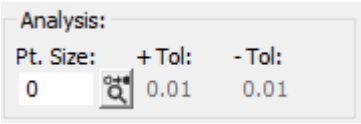
For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

15. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

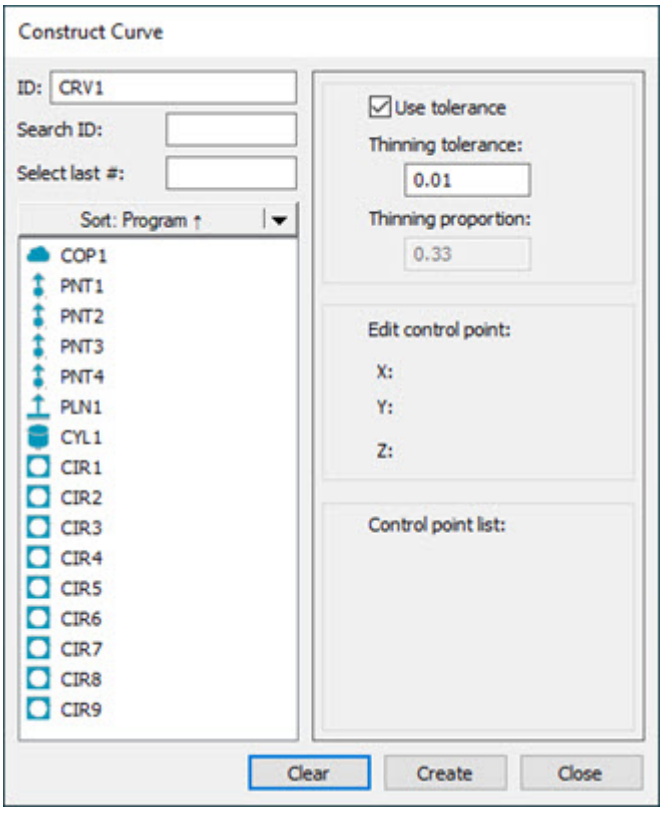
16. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

17. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
SLT1        =FEAT/SQUARE_SLOT,CARTESIAN,INNER,LEAST_SQR
             THEO/<56.303,53.462,0>,<0,0,1>,<1,0,0>,5,15
             MEAS/
<56.152,53.463,0>,<0,0,1>,<0.999733,-0.0231079,0>,5.123,
15.343
             DEPTH=50
             THEO_THICKNESS,0
             HORIZONTAL CLIPPING=7,VERTICAL
CLIPPING=9,INNER HORIZONTAL CLIPPING=0
             RINGBAND=ON,INNER OFFSET=2,OUTER OFFSET=10
             USE OUTLIER REMOVAL=ON,1
             REMOVE POINTS WITH NORMALS OUTSIDE=ON,0.5
             CONSTR/SLOT,SQUARE,EXTRACTED,COP1
```

Constructing a Curve



Construct Curve Dialog Box

Two types of constructed curves (Independent Curves and Dependent Curves) are available in PC-DMIS. The following table displays the two curves along with their necessary inputs. All curves require a set as input. The set can be a measured set, a constructed set, or a scan. The input set must contain at least four features (or input points in the case of a scan).

Construct Feature Type	the Edit Window Symbol	Number of Required Input Sets	Input Set	Comments
Dependent Curve	DEPENDENT	1	Set containing at least 4 inputs	The Curve updates as the input changes. Editing a curve changes a dependent curve to an independent curve.
Independent Curve	INDEPENDENT	1	Set containing at least 4 inputs	Uses the input feature for construction only. You can manually edit control points of the curve.

The Edit window command line for a sample Curve Construction using a *dependent curve type* would read:

```
feature_name=FEAT/CURVE, DEPENDENT, num_control_points,
num_input_feats, thinning_parameter

CONSTR/CURVE,INPUT_TYPE, input_id
```

For an independent curve type the Edit window command line would read:

```
feature_name=FEAT/CURVE, INDEPENDENT, num_control_points,
num_input_feats, thinning_parameter

CONSTR/CURVE,
```

num_control_points = This is the number of control points that define the curve. More control points allow the curve to more closely pass through the features, but too may cause unexpected behavior

num_input_feats = This is the number of features that the curve is attempting to fit.

These two parameters are not editable in the Edit window.

input_id = This is the ID of the set containing the features to be fitted.

thinning_parameter = For information about the thinning parameters, see the "Thinning Parameters" topic in the PC-DMIS Core documentation.

The actual Edit report displays in all capital letters.

Constructing a Dependent / Independent Curve

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

To construct a curve:

1. Open the **Construct Curve** dialog box (**Insert | Feature | Constructed | Curve**).
2. From the **Feature** list, select the desired set.
3. Select the **Use tolerance** check box and then set the value in the **Thinning Tolerance** box. Update the **Thinning proportion** value if necessary. For details, see the "Thinning Parameters" topic in this documentation.
4. Edit the control points if necessary. For details, see the "Edit Control Point" topic in this documentation.
5. Click the **Create** button.

All constructed curves start out as DEPENDENT curves and must be created from a single input, a set. The set can be one of three types:

- A measured set

- A constructed set
- A scan containing a single row of points. (See "Constructing a Set of Features".)

The input set must contain at least four features or points (in the case of a scan).

The resulting curve depends on the order that the features are added to the set (first to last).

You can also construct a curve by simply selecting multiple features from the feature list box rather than selecting a constructed set from the list. In this case, the command's `INPUT_TYPE` field will be empty when viewing the command in the Edit window.

The following paragraphs describe the options for constructing a curve.

Thinning Parameters (Thinning Tolerance or Thinning Proportion)

You can use one of two types of thinning parameters: Thinning Tolerance or Thinning Proportion. The **Use tolerance** check box on the **Construct Surface** and **Construct Curve** dialog boxes (**Insert | Feature | Constructed | Surface** and **Insert | Feature | Constructed | Curve**, respectively) lets you switch between either tolerance or proportion:

- *Thinning tolerance* controls the tightness (or exactness) of the curve or surface fit. Valid thinning tolerances range from 0.0 to 5.0, with the default being 0.01. The smaller the thinning tolerance, the closer the curve comes to passing through the centroids of the features included in the input set. If the thinning tolerance is 0.0, the curve or surface will pass through all of the centroids. A larger thinning tolerance will result in a curve or surface that has fewer fluctuations (at the expense of not lying near the input set's features). To see this, construct a curve or surface and then change the input tolerance and examine how its shape changes.
- *Thinning proportion*, alternately, may be used to control the quality of the fit. Valid thinning proportions range from between 0.0 and 1.0, with a default value of 0.33. The thinning proportion determines the number of degrees of freedom available in fitting the curve or surface to the centroids. At the lower extreme of 0, the algorithm will attempt to fit a straight line or plane to the centroids. At 1, it will compute a fit that passes through all of the centroids.

To make a DEPENDENT curve an INDEPENDENT curve (so that it is no longer associated with the input set):

1. Open the Edit window (**View | Edit Window**).
2. Select the curve feature that you constructed.
3. Navigate to the DEPENDENT field of that feature.
4. Press F7. It will change from DEPENDENT to INDEPENDENT.

You can change the shape of the curve by editing its control points.

Edit Control Point

If you select a preexisting curve, the **Construct Curve** dialog box (**Insert | Feature | Constructed | Curve**) also contains a list of control points in **Control Point list**. Select one of the control points. PC-DMIS places the corresponding values in the **X**, **Y**, and **Z** boxes so that you can edit them.

To edit these control points:

1. Ensure that a curve feature already exists.
2. Open the Edit window (**View | Edit Window**).
3. Select the curve in the Edit window.
4. Press F9 to activate the **Construct Curve** dialog box.
5. Select the control point to be changed in **Control Point list**. For details, see the "Control Point List" topic in this documentation.
6. Edit the individual X, Y, and Z components of the point.
7. Click the **OK** button.

The curve is then updated to reflect the changes.

If you edit the control points on a dependent curve, it automatically becomes an independent curve because it is no longer based on the input set.

Control Point List

The **Control Point list** option in the **Construct Curve** dialog box (**Insert | Feature | Constructed | Curve**) is only visible when there is an existing curve associated with the dialog box. If a curve exists and you want to edit the control points of that curve, the **Control Point list** displays all of the control points for that curve.

Determining the Length Between Two Points on a Scan

To determine the length between two points on a scan, do the following:

1. Open the **Construct Curve** dialog box (**Insert | Feature | Constructed | Curve**).
2. Select a previously created scan for your input.
3. Click the **Create** button. PC-DMIS inserts a constructed curve into the Edit window.

4. On the Edit window (**View | Edit Window**), in Command Mode, locate the last line of the constructed curve code block:
   `CONSTR/CURVE`

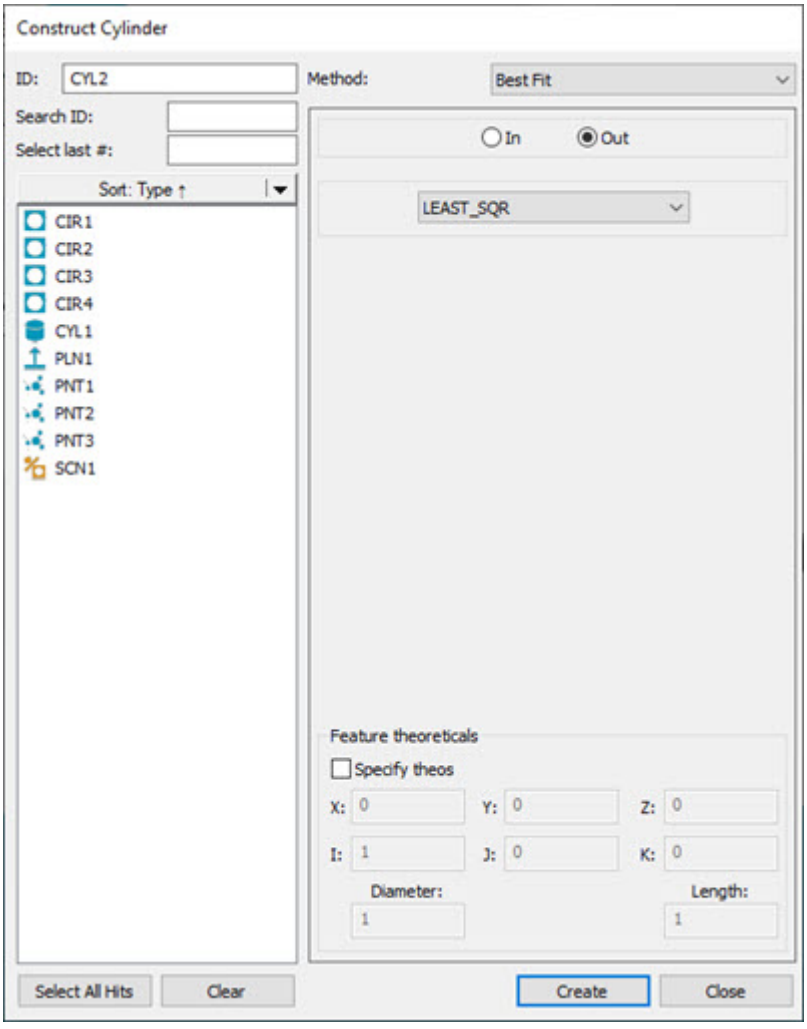
5. Press the Tab key on your keyboard until PC-DMIS highlights the ID for the scan you selected to use as the input.

6. Change the input ID to use specific points from the scan by typing `SCN1.HITS[n..m]` where SCN1 represents your scan ID and n and m represent a range between two points (or hits) on the scan. For example, if you wanted to get the length between hits 50 and 80 of the scan with an ID of SCN12, you would type, `SCN12.HITS[50..80]` .

7. Create a Location dimension and use the constructed curve as the input. Use the dimension to report the L axis (L for Length). The location dimensions would then display the length of the spline between the two hits you specified.

If you omit steps 4, 5, and 6, PC-DMIS reports the entire length of the scan (or curve).

Constructing a Cylinder Feature



Construct Cylinder Dialog Box

PC-DMIS allows several methods for constructing a cylinder. The following table lists the various types of constructed cylinders, along with their necessary inputs. Some features may require no inputs while others may require six inputs or more. In the following table, the term "Any" indicates that the construction can take any type of feature as input for the construction. PC-DMIS allows you to select the features in any order.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Comments
Auto Cylinder	-	-	-	-	See "Auto Cylinder Construction".
Best Fit Cylinder	BF	At least 6 inputs are needed. See Notes below.	-	-	Constructs a best fit cylinder using the given inputs.
Best Fit Recomp Cylinder	BFRE	At least 6 inputs are needed. (1 must be a point) See Notes below.	-	-	Constructs a best fit cylinder using the given inputs.
Cast Cylinder	CAST	1	Any	-	Constructs a cylinder at the centroid of the input feature.
Project Cylinder	PROJ	1 or 2	Any	Plane	One input feature will project the cylinder to the workplane.
Reverse Cylinder	REV	1	Cylinder, Cone, Line, Slot	-	Constructs a cylinder with a reversed vector.
Extracted Cylinder	EXTRACTED_CYL	1	COP or Mesh	-	Constructs an extracted cylinder from the COP or Mesh object at the specified diameter or height.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

You can construct Best Fit and Best Fit Recompensate Cylinders from two circles. The input circles must be of type Constructed Best Fit (BF) / Best Fit Recompensate (BFRE) or Measured circles. The total amount of hits in each circle must be at least three.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

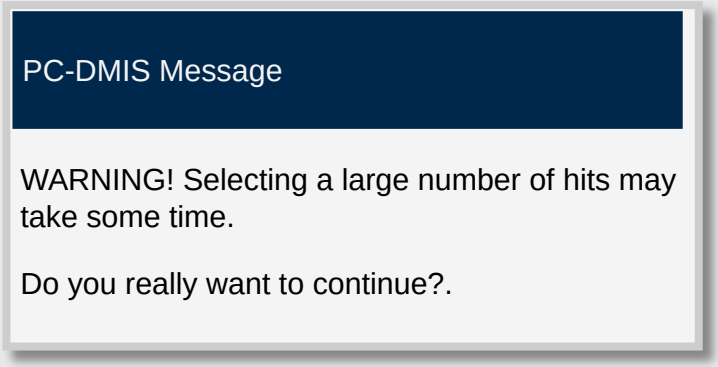
"Cannot construct [feature]. Combination of input features not accepted."

To construct a cylinder:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select the method of the constructed cylinder. The available options are:
    - Best Fit or Best Fit Recompensate Cylinder
    - Cast Cylinder
    - Projected Cylinder
    - Reversed direction Cylinder
    - Auto Cylinder
    - Extracted Cylinder

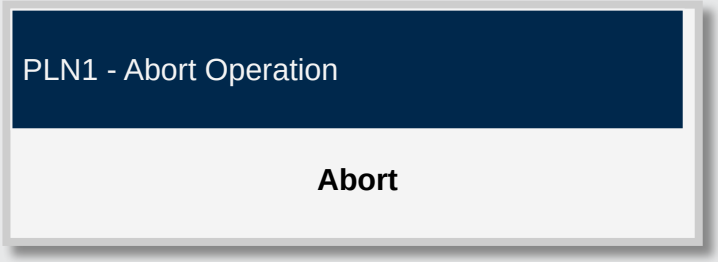
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

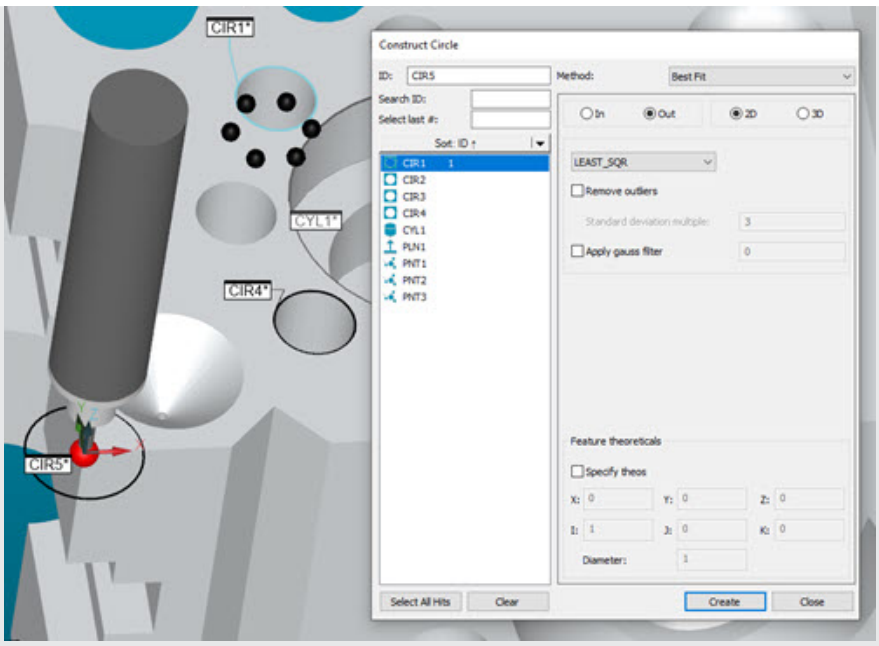


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

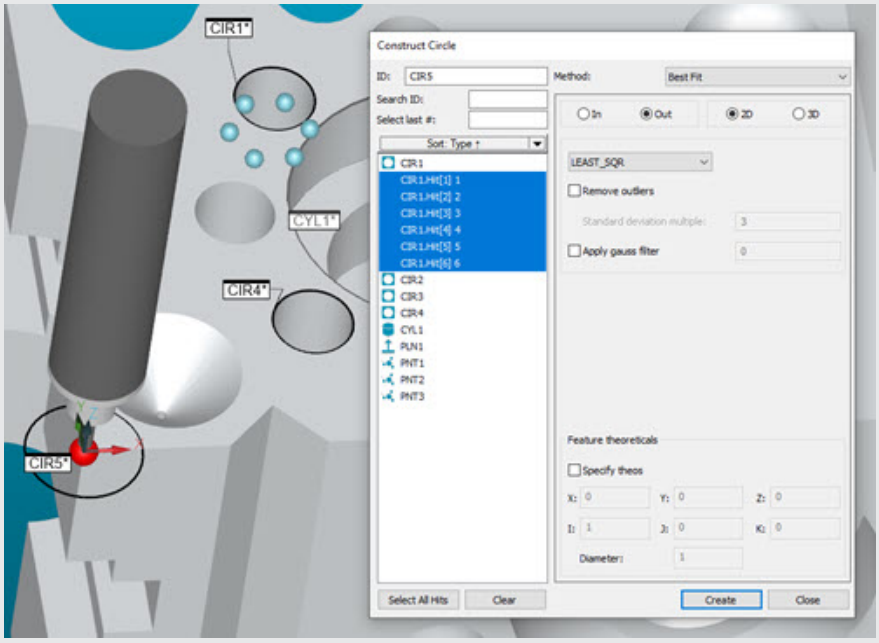
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

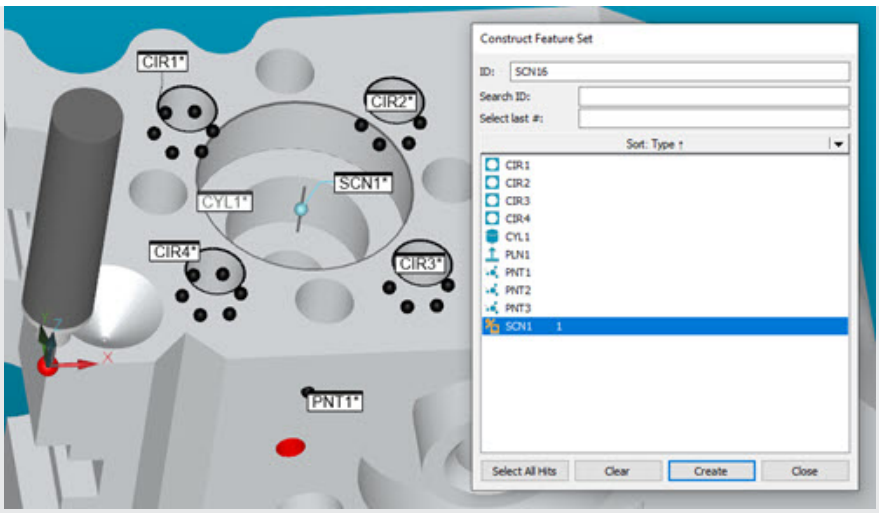
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. Using the table above, select the features from the **Feature** list based on the selected method.
4. Select the **In** or **Out** option. For details, see the "In / Out Cylinder" topic in this documentation.
5. If you selected either the Best Fit or Best Fit Recompensate methods, from the **Best Fit Type** list, select the type of best fit algorithm to use. For details, see the "Best Fit Type" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for a sample Cylinder Construction would read:

```
feature_name=FEAT/CYLINDER, TOG1,TOG4,TOG5

THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam,length

ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam,length

CONSTR/ TOG2,TOG3,.....
```

The actual Edit report displays in all capital letters.

AUTO is the default method of construction. This option automatically determines the best way to construct a cylinder using the input feature(s). See "Auto Cylinder Construction"

The following text is the basic format for cylinders.

TOG1 = POLR or RECT

TOG2 = CYLINDER

TOG3 = BF / BFRE / CAST / PROJ / REV

TOG4 = IN / OUT

TOG5 = LEAST_SQR / MAX_INSC / MIN_CIRCSC / MIN_SEP / FIXED_RAD (only for measured, BF and BFRE circles)

length = This value is measured between the first measured circle (first three hits) and the furthest point away from the first three hits.

The first three lines displayed in the Edit window will be the same for constructed cylinders. The fourth line will be slightly different, according to the type of feature you are constructing. You can switch between the different types of cylinders by placing the cursor on TOG3 and pressing F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

The following topics describe the available options for constructing a cylinder:

In / Out Cylinder

The **In** and **Out** options tell PC-DMIS whether to construct the cylinder as an internal or an external cylinder.

- If you select **In**, PC-DMIS constructs the cylinder as an internal cylinder.
- If you select **Out**, PC-DMIS constructs the cylinder as an external cylinder.

Auto Cylinder Construction

The following list indicates the type of cylinder that PC-DMIS constructs when you select the specified inputs and choose the **Auto** method. The order of the selected features is not important. If you choose an incorrect input feature, PC-DMIS displays an error message and does not automatically construct the indicated feature type.

Be aware that certain patterns of points, such as two rows of three equally spaced points or two rows of four equally spaced points, result in multiple ways to construct or measure a cylinder. The Best Fit algorithm may then construct or measure the cylinder using an unexpected solution. For best results, measured or constructed cylinders should use a pattern of points that will eliminate unwanted solutions.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select the **Auto** option.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.
4. Select the **In** or **Out** option. For details, see the "In / Out Cylinder" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

Input Feature List

Input Feature(s)	Constructs
Five or more Features =	Best Fit Cylinder
Any One Cylinder =	Reverse Cylinder
Any One Feature (except Cylinder or Set) =	Cast Cylinder
Any One Set =	Best Fit Cylinder
Plane + Any Feature =	Project Cylinder

Constructing a Best Fit or Best Fit Recompensate Cylinder

You can construct a best fit cylinder from six or more points. The first three points must be on an approximately planar cross section of the cylinder that is normal to its centerline. PC-DMIS computes a *least squares* cylinder, one for which PC-DMIS minimizes the average squared distance from the data points to the cylinder.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Be aware that certain patterns of points, such as two rows of three equally spaced points or two rows of four equally spaced points, result in multiple ways to construct or measure a cylinder. The Best Fit algorithm may then construct or measure the cylinder using an unexpected solution. For best results, measured or constructed cylinders should use a pattern of points that will eliminate unwanted solutions.

To construct a Best Fit or Best Fit Recompensate Cylinder:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. From the **Feature** list, select your input features by either choosing at least two appropriate circle features or at least six features of any type.
    - *For at least two circle features*, they *must* be constructed Best Fit (BF) / constructed Best Fit Recompensate (BFRE), or measured circles. The total amount of hits in each circle must be at least three.
    - *For at least six features*, they can be of any type.
    - However, if you selected **BF Recomp**, at least one feature must be a point.
4. From the **Best Fit Type** list, select the type of best fit algorithm to use. For details, see the "Best Fit Type" topic in this documentation.

5. Click the **Create** button.

> The math routine that PC-DMIS uses to get the points from the input features behaves differently depending on the type of input feature. Constructed features other than the appropriate input circles described above return a single point. The constructed BF circles, constructed BFRE circles, or measured circles described above return their input points.

The Edit window command line for this option would read:

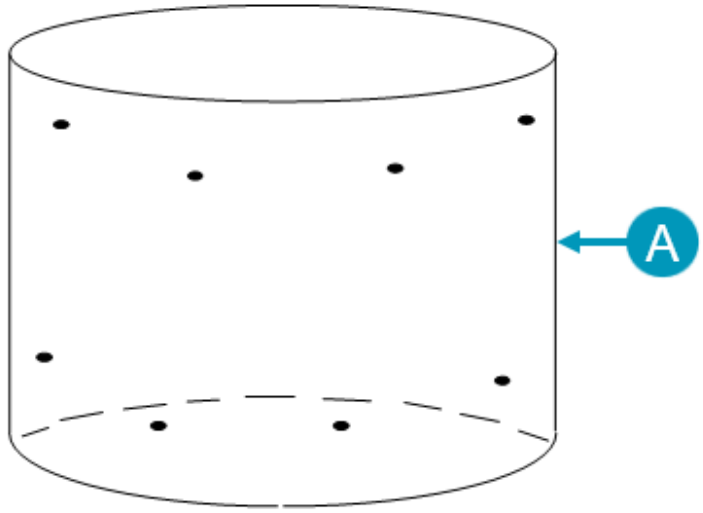
```
CONSTR/CYLINDER,BF, feat_1,feat_2,feat_3,feat_4,feat_5,feat_6
```

(Uses the actual measured points for construction.)

or

```
CONSTR/CYLINDER,BFRE, feat_1,feat_2,feat_3,feat_4,feat_5,...
```

(Uses the center of the probe for measurement.)



A - *Best Fit Cylinder constructed from eight points*
Constructing a Cylinder from Six or More Points

Best Fit Type

This list becomes available if you select **Best Fit** or **BF Recomp** options when you construct the cylinder. It allows you to specify the type of Best Fit construction used. The types are:

- LEAST_SQR
- MIN_SEP
- MAX_INSC

- MIN_CIRCSC
- FIXED_RAD

These calculation types are discussed in the "Best Fit Type" topic for a constructed circle feature.

Constructing a Projected Cylinder

You can construct a cylinder from any feature and a plane. The diameter of the projected cylinder is the diameter of the first input feature (if circular) or twice the probe diameter if not a circular feature. You need to input the length and diameter for bound measurements. If there is only one input feature, the projection is to the workplane.

To construct a Projected Cylinder:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select *either* one or two features. If one feature is selected, the feature can be of any type. If two features are selected, the first feature can be of any type. The second feature *must* be a plane.
4. Select the **In** or **Out** option. For details, see the "In / Out Cylinder" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:
`CONSTR/CYLINDER,PROJ,` *feat_1,(feat_2)*

Constructing a Cast Cylinder

You can construct a cylinder by changing any given feature into a cylinder. PC-DMIS constructs the cylinder at the centroid of the input feature. If a sheet metal point is used, the diameter is the probe diameter. For some sheet metal features (such as slots and notches) the width will be used as the diameter. For features that do not have diameters (lines, points, etc.), a value that is four times the probe diameter is used.

You can change the size of the cylinder; this changes the cylinder from DEPENDENT to INDEPENDENT. This means that when the cylinder is executed, the length and diameter do not change based on the input feature but remain independent of the input feature,

while the position and vector stay dependent on the input feature. This allows you to have control over the cylinder size in cases where the input feature doesn't really have a length and diameter, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field that you may change.

PC-DMIS uses the new attributes for all calculations (for example, if the diameter is changed) instead of the default values as described above.

To construct a Cast Cylinder:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature. The feature can be of any type.
4. Select the **In** or **Out** option. For details, see the "In / Out Cylinder" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CYLINDER,CAST, feat_1 , (DEPENDENT | INDEPENDENT)
```

Changing the Direction of a Cylinder

You can construct a cylinder with a reversed vector.

To construct a cylinder with reverse vectors, follow these steps:

1. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**).
2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select one feature. It *must* be a cylinder.
4. Select the **In** or **Out** option. For details, see the "In / Out Cylinder" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CYLINDER,REV, feat_1
```

Constructing an Extracted Cylinder

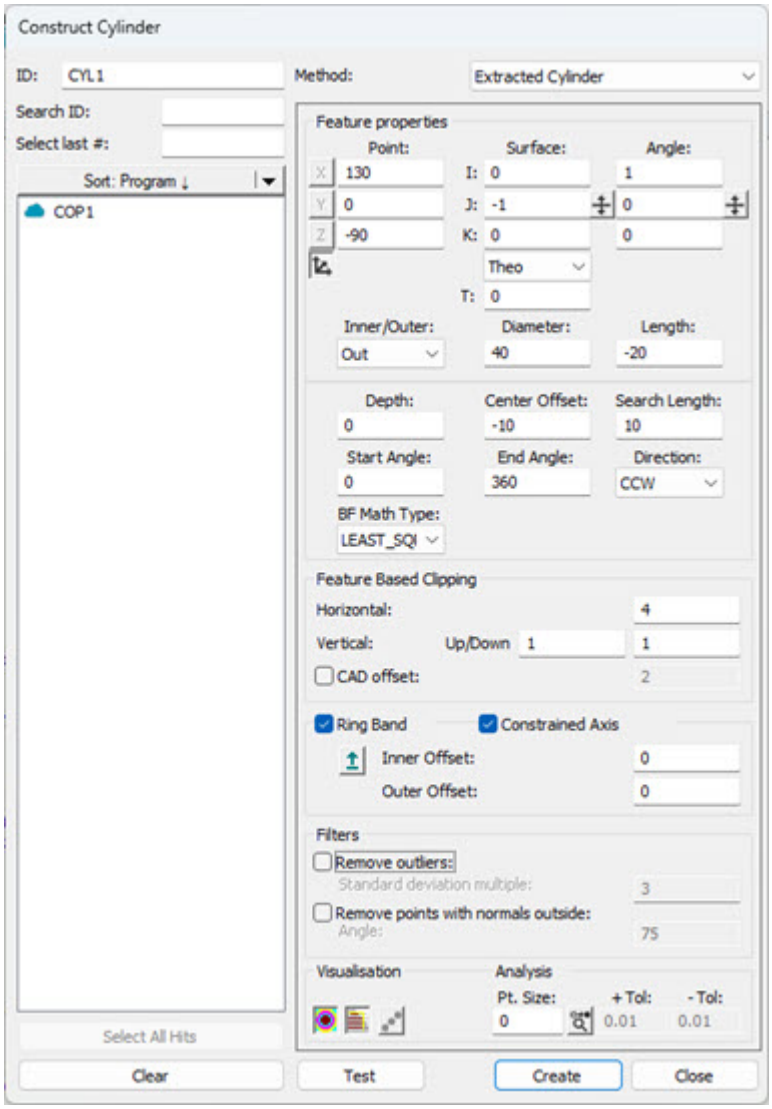
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a cylinder that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Cylinder** dialog box (**Insert | Feature | Constructed | Cylinder**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Cylinder Dialog Box - Extracted Cylinder Option

3. From the **Method** list, select the **Extracted Cylinder** option.

4. From the **Reference** area, select the COP or Mesh that you want to use to extract the cylinder from.

5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.

6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. In the **Angle** section, enter the corresponding vector angle values. You can use the **Material Thickness Type** list and the **T** box below it to type in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

   Flip Vectors

   Polar/Cartesian

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted cylinder is an inner or outer cylinder type from the **Inner/Outer** list.

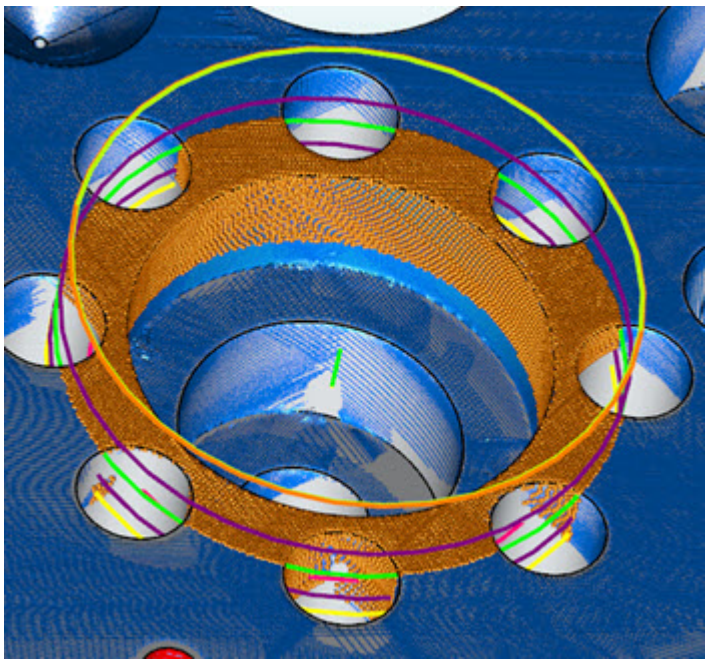
8. Enter the diameter and length of the cylinder in the respective boxes. The **Diameter** value defines the cylinder's initial diameter, and the **Length** value defines the length (height) of the cylinder's axis. The **Length** parameter is only valid as nominal. PC-DMIS does not perform an actual length measurement.

9. Enter the **Depth** value. This parameter controls the location of the laser focal point in relation to the cylinder's outside diameter (outer cylinders) or the cylinder's center axis (inner cylinders). This allows you to control how the laser stripes fall on the cylinder's surface by specifying how far or close the laser is to the cylinder's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height using data found at the lowest possible depth from the surface plane. A depth of any other value causes the software to perform the calculation at that depth.

10. Enter the **Center Offset** value. This value identifies the center location of where the laser starts to measure the cylinder. If you do not enter a **Search Length** value, this value defines where the measurement begins.

11. Enter the **Search Length** value. This value defines the distance from the **Center Offset** the laser measures for the cylinder. For example, if you have a **Center Offset** value of zero and a **Search Length** value of 20, the laser begins measuring +20 units from the **Center Offset** value.

    PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the zone that PC-DMIS uses for the extracted cylinder. The yellow cylinder is the surface. The yellow cylinder is also the horizontal zone and the green cylinder is the vertical zone. The orange points are the candidate points that the extraction considers.



Example of a constructed extracted cylinder showing the candidate points.

12. The **Start Angle** and **End Angle** boxes allow you to change the default starting and ending angles on the feature. For details on these boxes, see the "Start and End Angles" topic in this documentation.

13. You can use the **Direction** list to specify the direction in which PC-DMIS takes the hits. For details, see "Direction List" in this documentation.

14. You can use the **BF Math type** to select the type of best fit algorithm to use. For details see the "Best Fit Type" topic in this documentation.

15. From the **Feature Based Clipping** area, define the **Horizontal**, **Vertical Up/Down**, and **Vertical** values. These values set the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

    Alternatively, you can clip data within an offset boundary around all the CAD elements on a surface with the **CAD offset** option. This is also called *CAD segregation*. For details, see the "CAD offset" section of the "Feature Based Clipping Parameters" topic in the PC-DMIS Laser documentation.

15. If you want to define the Ring Band offsets, click the **Ring Band** check box and type the **Inner Offset** and **Outer Offset** values. Click the **Constrained Axis** check box to constrain the Cylinder feature to the vector of the plane surrounding the cylinder. For details on how the **Ring Band** works, see the "Ring Band Parameters" topic in the PC-DMIS Laser documentation.

16. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

17. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

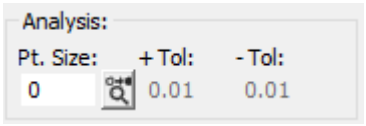
> The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.
>
> For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

18. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

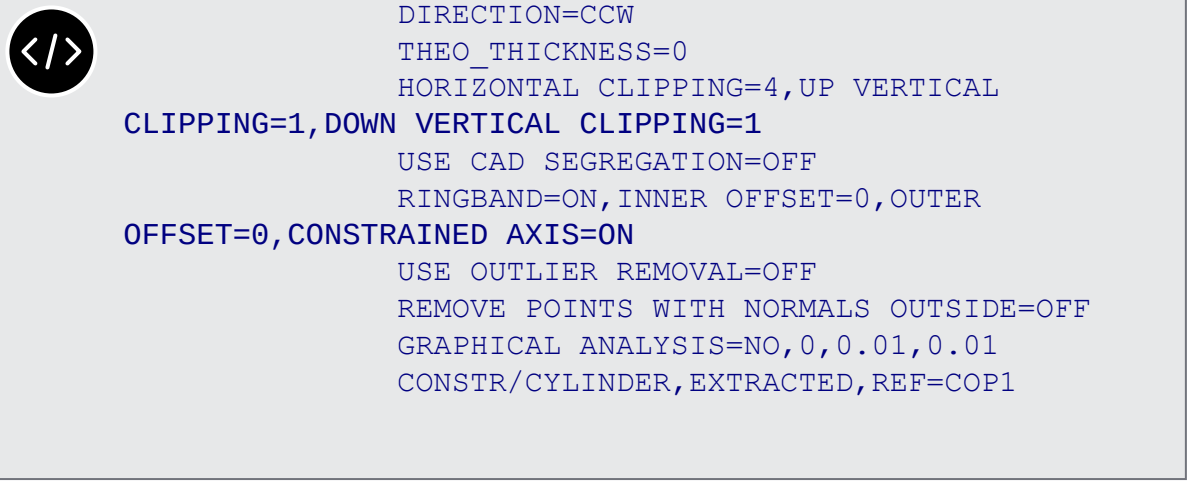


- The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

19. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

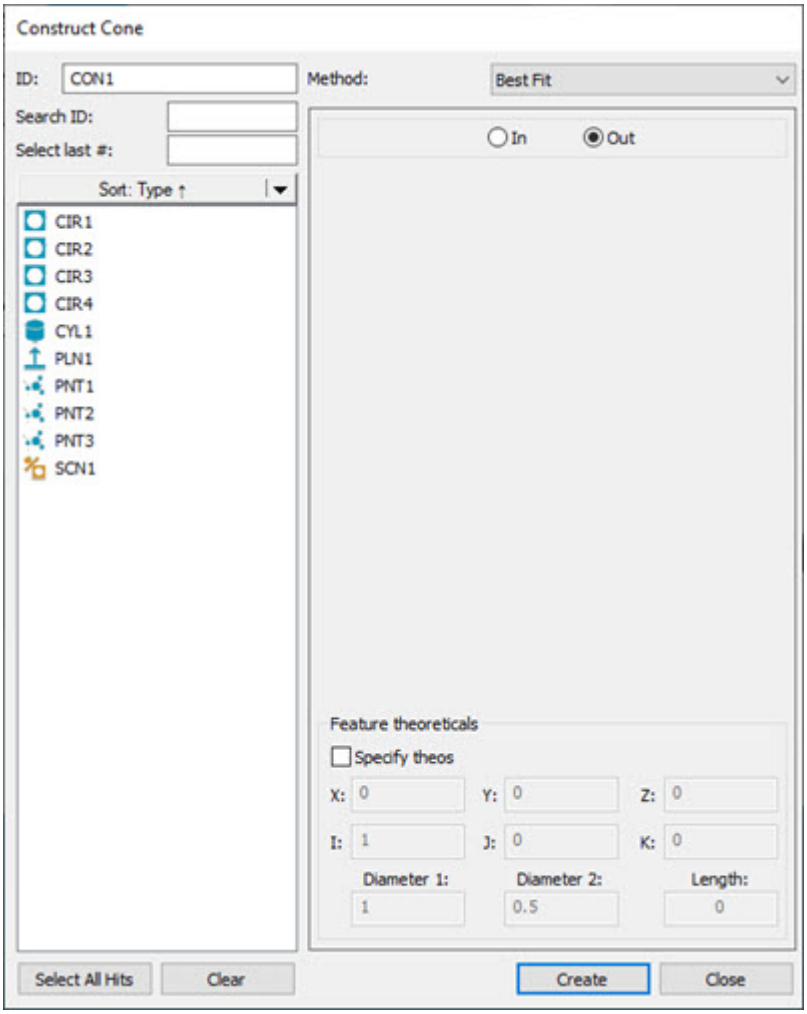
20. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:



```
CYL1       =FEAT/CYLINDER,CARTESIAN,OUT,LEAST_SQR
            THEO/<120,0,-90>,<0,-1,0>,40,20
            ACTL/<120,0,-90>,<0,-1,0>,40,20
            ANGLE VEC=<1,0,0>
            DEPTH=0
            CENTER OFFSET=-10
            SEARCH LENGTH=10
            START ANG=0,END ANG=360
            DIRECTION=CCW
            THEO_THICKNESS=0
            HORIZONTAL CLIPPING=4,UP VERTICAL
CLIPPING=1,DOWN VERTICAL CLIPPING=1
            USE CAD SEGREGATION=OFF
            RINGBAND=ON,INNER OFFSET=0,OUTER
OFFSET=0,CONSTRAINED AXIS=ON
            USE OUTLIER REMOVAL=OFF
            REMOVE POINTS WITH NORMALS OUTSIDE=OFF
            GRAPHICAL ANALYSIS=NO,0,0.01,0.01
            CONSTR/CYLINDER,EXTRACTED,REF=COP1
```

Constructing a Cone Feature



Construct Cone Dialog Box

PC-DMIS allows several methods for constructing a cone. The following table lists the various types of constructed cones, along with their necessary inputs. Some features may require no inputs while others may require six inputs or more. The term "Any" in the table below indicates that the construction can take any type of feature as input for construction. PC-DMIS allows you to select the features in any order.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Comments
Auto Cone	-	-	-	-	See "Auto Cone Construction".
Best Fit Cone	BF	At least 6 inputs are needed.	-	-	Constructs a best fit cone using the given inputs. See the Note below for recommended inputs.
Best Fit with Recomp Cone	BFRE	At least 6 inputs are needed. (1 must be a point)	-	-	Constructs a best fit cone using the given inputs. See the Note below for recommended inputs.
Cast Cone	CAST	1	Any	-	Constructs a cone at the centroid of the input feature.
Project Cone	PROJ	1 or 2	Any	Plane	Using one input feature, projects the cone to the workplane.
Reverse Cone	REV	1	Cone	-	Constructs a cone with a reversed vector for the axis.
Extracted Cone	EXTRACTED_CONE	1	COP or Mesh	-	Constructs an extracted cone from the COP or Mesh object at the specified diameter or height of the cone.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

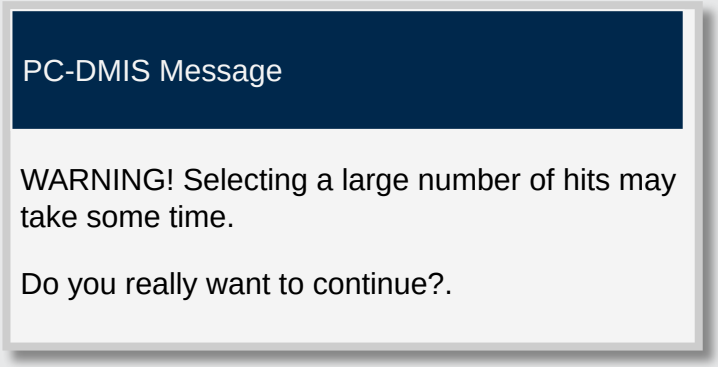
"Cannot construct [feature]. Combination of input features not accepted."

To construct a cone:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select the method of the constructed cone. The available options are:
    - Auto Cone
    - Best Fit
    - Best Fit Recompensate Cone
    - Projected Cone
    - Cast Cone
    - Reversed direction Cone
    - Extracted Cone

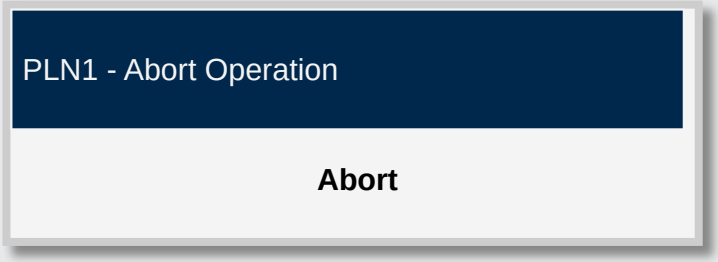
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

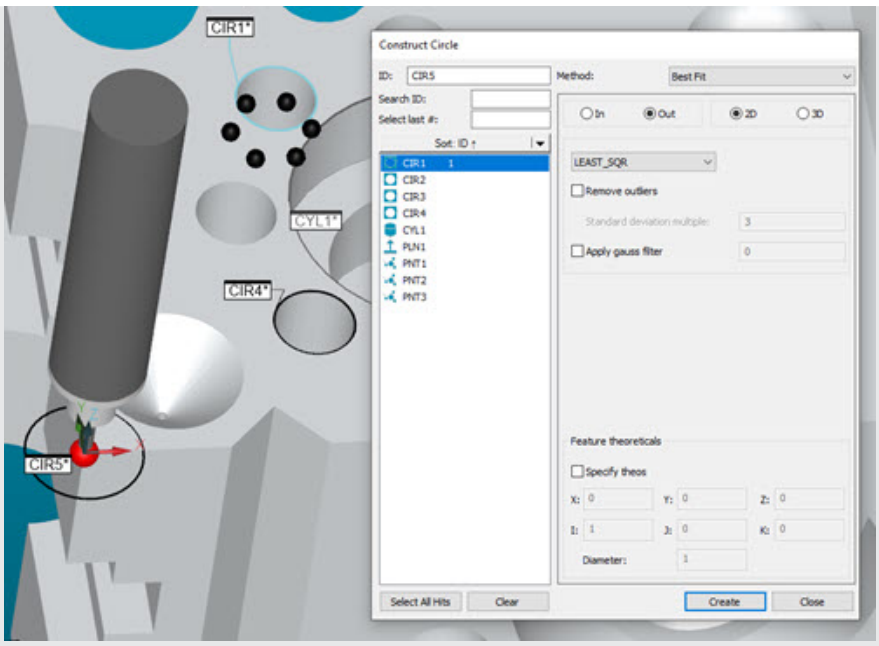


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

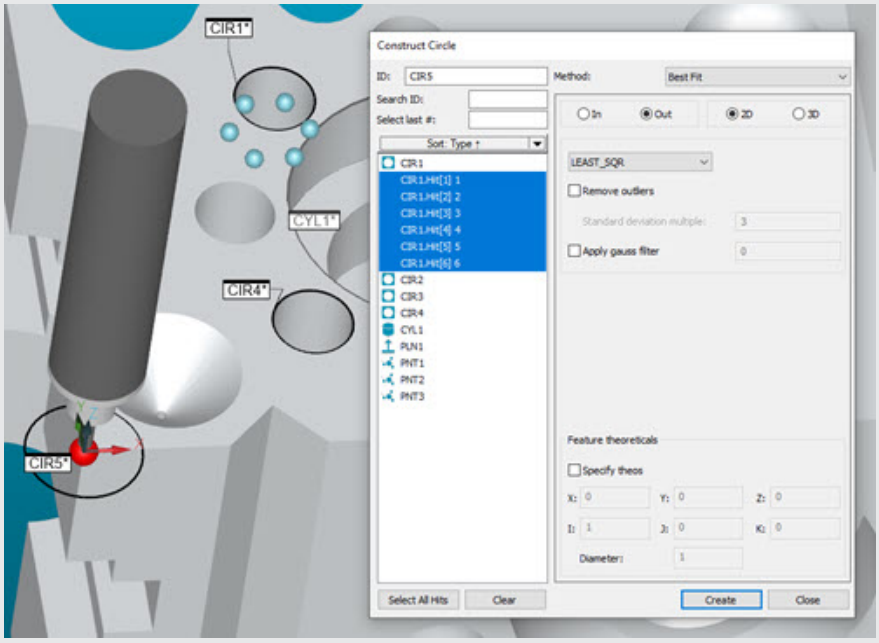
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

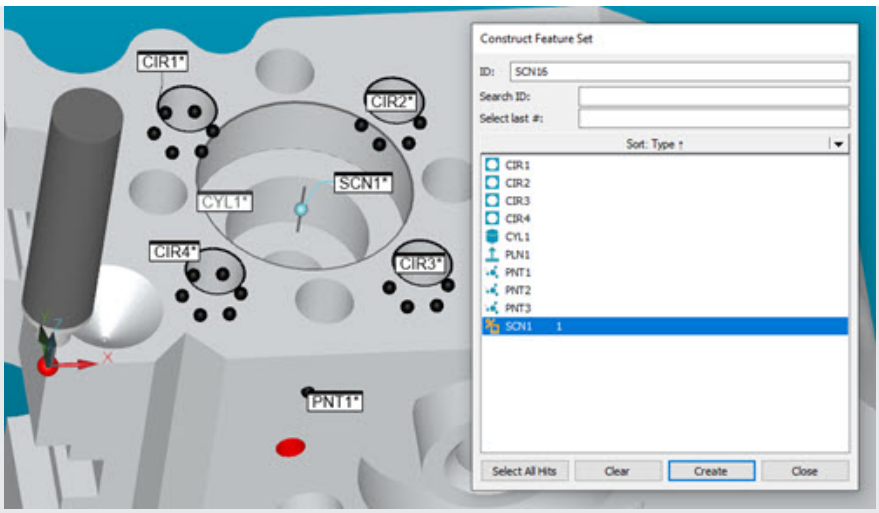
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. Use the table above to select the features from the **Feature** list based on the selected method.
4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
 feature_name=FEAT/CONE, TOG1,TOG4,ANG
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,ang
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,ang
CONSTR/ TOG2,TOG3,.....
```

```
 feature_name=FEAT/CONE, TOG1,TOG4,LENG
THEO/x_cord,y_cord,z_cor,i_vec,j_vec,k_vec,leng,diam_1,diam_2
ACTL/x_cord,y_cord,z_cor,i_vec,j_vec,k_vec,leng,diam_1,diam_2
CONSTR/ TOG2,TOG3,.....
```

The actual Edit report displays in all capital letters.

TOG1 **= POLR or RECT**

TOG2 **= CONE**

TOG3 **= BF / BFRE / CAST / PROJ / REV**

TOG4 **= IN / OUT**

TOG5 **= ANG / LENG**

The first three lines that display in the Edit window are similar for constructed cones. If the feature is bound or unbound, the theoretical and actual values that are displayed will vary. The fourth line will be slightly different, according to the type of feature you are constructing. You can switch between the different types of cones by placing the cursor on TOG and clicking the left mouse button. The keyboard can also be used to switch toggle fields. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

Auto is the default method for constructed cones. This option automatically determines the best way to construct a cone using the input feature(s). See "Auto Cone Construction"

The following paragraphs describe the available options for constructing a cone:

In / Out Cone

The **In** and **Out** options tell PC-DMIS whether to construct the cone as an internal or an external cone.

- If you select **In**, PC-DMIS constructs the cone as an internal cone.
- If you select **Out**, PC-DMIS constructs the cone as an external cone.

Auto Cone Construction

The "Input Feature List" table indicates the type of cone that you can construct when you select the specified inputs and choose the **Auto** method. The order that you select the features is not important. If you select an incorrect input feature(s), PC-DMIS displays an error message and does not automatically construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select the **Auto** option.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.

4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.

5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

6. Click the **Create** button.

Input Feature List

Input Feature(s)	Constructs
Six or more Features	Best Fit Cone
Any One Cone	Reverse Cone
Any One Feature (except Cone or Set)	Cast Cone
Any One Set	Best Fit Cone
Plane + Any Feature	Project Cone

Constructing a Best Fit or Best Fit Recompensate Cone

You can construct a best fit cone using six or more features. The first three inputs must be on an approximately planar cross section of the cone that is normal to its center line. The remainder of the points should lie either above or below the plane defined by the first three points, but not on both sides of the plane. This method of measuring yields the best results. PC-DMIS computes a *least squares* cone, one for which PC-DMIS minimizes the average squared distance from the data points to the cone.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To Construct a Best Fit or Best Fit Recompensate Cone:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select either the **Best Fit** or **BF Recomp** option.
3. From the **Feature** list, select at least six features.
4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

For a Best Fit constructed cone feature, the Edit window command line would read:

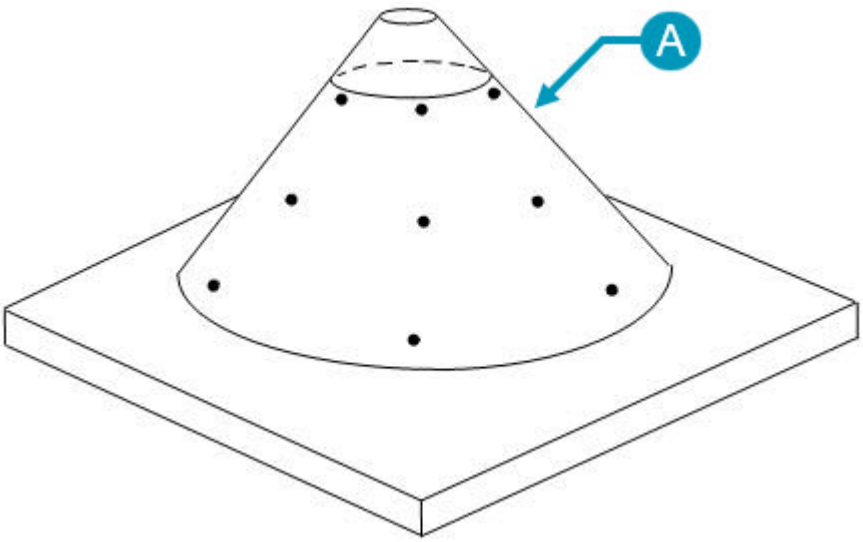
```
CONSTR/CONE,BF, feat_1,feat_2,feat_3,feat_4,feat_5,feat_6
```

PC-DMIS uses the actual measured points for construction.

For a Best Fit Recompensate constructed cone feature, the Edit window command line would read:

```
CONSTR/CONE,BFRE, feat_1,feat_2,feat_3,feat_4,feat_5,feat_6
```

PC-DMIS uses the center of the probe for measurement.



A - *Best Fit Constructed Cone from Nine Points*
Constructing a Cone from Six or More Points

Constructing a Projected Cone

You can construct a cone by projecting any feature onto a plane. If the projected input feature is not a cone, PC-DMIS uses default values for the included angle and two axis lengths. The first length is the distance between the vertex and the first circle. The second

length is the distance between the two circles. If there is only one input, the projection is to the current workplane.

To construct a Projected Cone:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select *either* one or two features. If you only select one feature, the feature can be of any type. If you select two features, the first feature can be of any type, but the second feature *must* be a plane.
4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CONE,PROJ, feat_1,(feat_2)
```

Constructing a Cast Cone

You can construct a cone by changing any given feature into a cone. PC-DMIS constructs the cone at the centroid of the input feature. If the input feature is not a cone, PC-DMIS uses a default value for the included angle. If the input feature is not a line element (line, cylinder, or slot), PC-DMIS uses a default length for the axis length.

You can change the size of the cone; this changes the cone from DEPENDENT to INDEPENDENT. This means that when the cone is executed that the size does not change based on the input feature, but is independent of the input feature, while the position and vector remain dependent on the input feature. This allows you to have control over the cone's size in cases where the input feature doesn't really have a size, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field that you may change.

PC-DMIS uses the new attributes for all calculations (for example, if the half angle is changed) instead of the default values as described above.

To construct a Cast Cone:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.

4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CONE,CAST, feat_1 , (DEPENDENT | INDEPENDENT)
```

Changing the Direction of a Cone

You can construct a cone with a reversed vector.

To construct a Reverse Cone:

1. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**).
2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select one feature. It *must* be a cone.
4. Select the **In** or **Out** option. For details, see the "In / Out Cone" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/CONE,REV, feat_1
```

Constructing an Extracted Cone

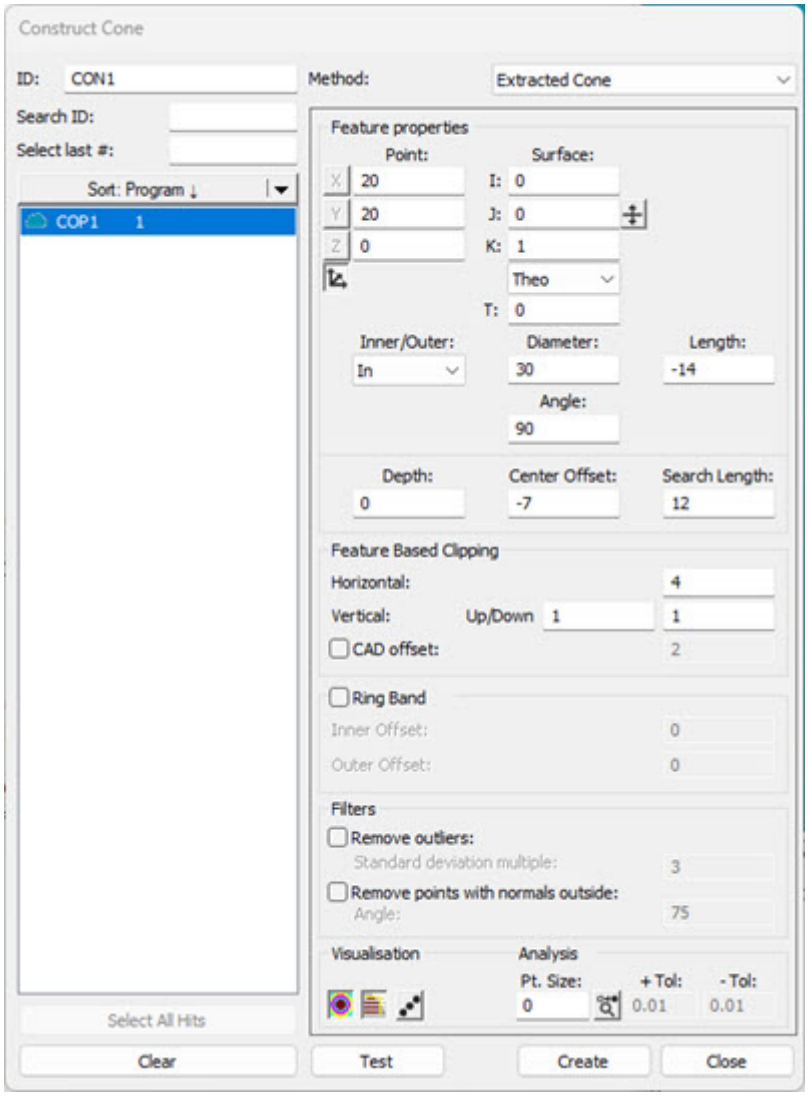
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a cone that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Cone** dialog box (**Insert | Feature | Constructed | Cone**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Cone Dialog Box - Extracted Cone Option

3. From the **Method** list, select the **Extracted Cone** option.
4. From the **Reference** area, select the COP or Mesh that you want to use to extract the cone from.
5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.
6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. In the **Angle** section, enter the corresponding vector angle values. You can use the **Material Thickness Type** list and the **T** box below it to type

in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

You can use these controls to perform the associated functions:

Flip Vectors

Polar/Cartesian

For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted cone is an inner or outer cone type from the **Inner/ Outer** list.

8. Enter the diameter and length of the cone in the respective boxes. The **Diameter** value defines the cone's initial diameter, and the **Length** value defines the length (height) of the cone's axis. The **Length** parameter is only valid as nominal. PC-DMIS does not perform an actual length measurement.

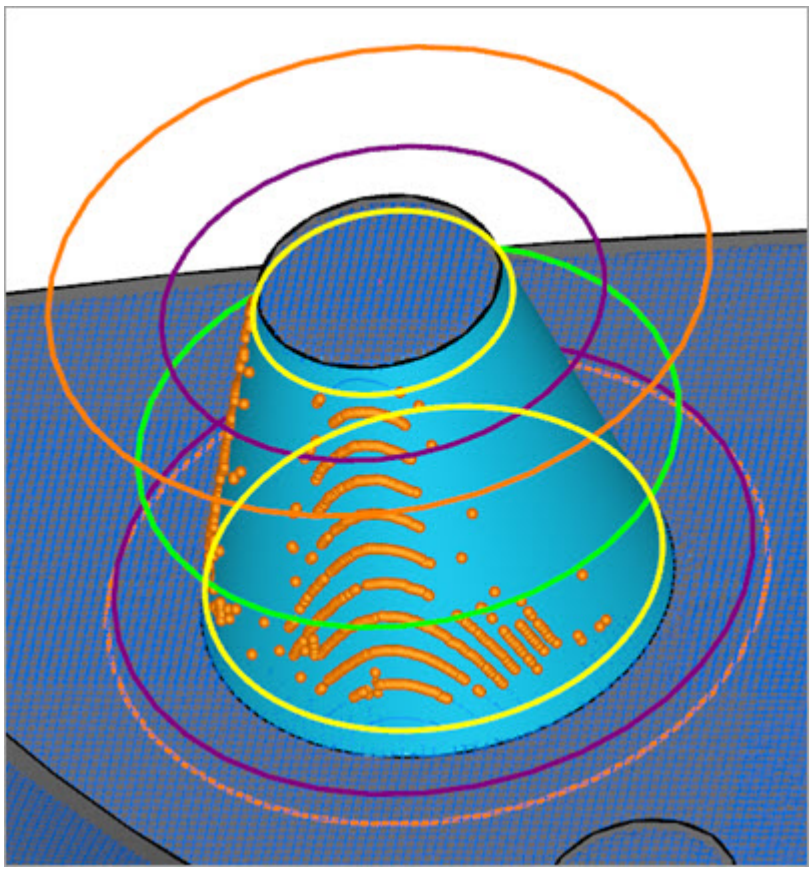
9. Enter the **Angle** value. This is the included angle of the cone.

10. Enter the **Depth** value. This parameter controls the location of the laser focal point in relation to the cone's outside diameter (outer cones) or the cone's center axis (inner cones). This allows you to control how the laser stripes fall on the cone's surface by specifying how far or close the laser is to the cone's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height using data found at the lowest possible depth from the surface plane. A depth of any other value causes the software to perform the calculation at that depth.

11. Enter the **Center Offset** value. This value identifies the center location where the laser starts to measure the cone. If you do not enter a **Search Length** value, this value defines where the measurement begins.

12. Enter the **Search Length** value. This value defines the distance from the **Center Offset** the laser measures for the cone. For example, if you have a **Center Offset** value of zero and a **Search Length** value of 20, the laser begins measuring +20 units from the **Center Offset** value.

    PC-DMIS draws the extraction zone and centers it around the XYZ location point. This box defines the cylinder zone that PC-DMIS uses for the extracted cone. The yellow cylinder is the surface. The yellow cylinder is also the horizontal zone and the green cylinder is the vertical zone. The orange points are the candidate points that the extraction considers.



Example of an extracted cone showing the candidate points.

13. From the **Feature Based Clipping** area, define the **Horizontal**, **Vertical Up/Down**, and **Vertical** values. These values set the dimensions for the green extraction zone region. Consider part variability when you define the extraction zone.

    Alternatively, you can clip data within an offset boundary around all the CAD elements on a surface with the **CAD offset** option. This is also called *CAD segregation*. For details, see the "CAD offset" section of the "Feature Based Clipping Parameters" topic in the PC-DMIS Laser documentation.

14. If you want to define the Ring Band offsets, click the **Ring Band** check box and enter the **Inner Offset** and **Outer Offset** values. For details on how the **Ring Band** works, see the "Ring Band Parameters" topic in the PC-DMIS Laser documentation.

15. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

16. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

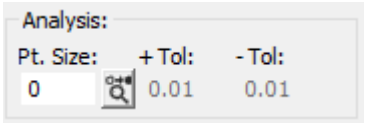
For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

17. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

18. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

19. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
CON1=FEAT/CONE,CARTESIAN,OUT
   THEO/<69,90,13.995>,<0,0,-1>,30,13.995,8
   ACTL/
<68.873,89.926,13.994>,<0.0061449,0.0001119,-0.9999811>,
30.367,13.995,7.893
   DEPTH=0
   CENTER OFFSET=6.998
   SEARCH LENGTH=11.995
   THEO_THICKNESS,0
   USE CAD SEGREGATION=ON,CAD OFFSET=1
   HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=1
   RINGBAND=OFF
   USE OUTLIER REMOVAL=OFF
   REMOVE POINTS WITH NORMALS OUTSIDE=OFF
   CONSTR/CONE,EXTRACTED,REF=COP1
```

Variations

This first example shows the unbounded format for a constructed cone command in the Edit window with TOG5 changed to ANG.

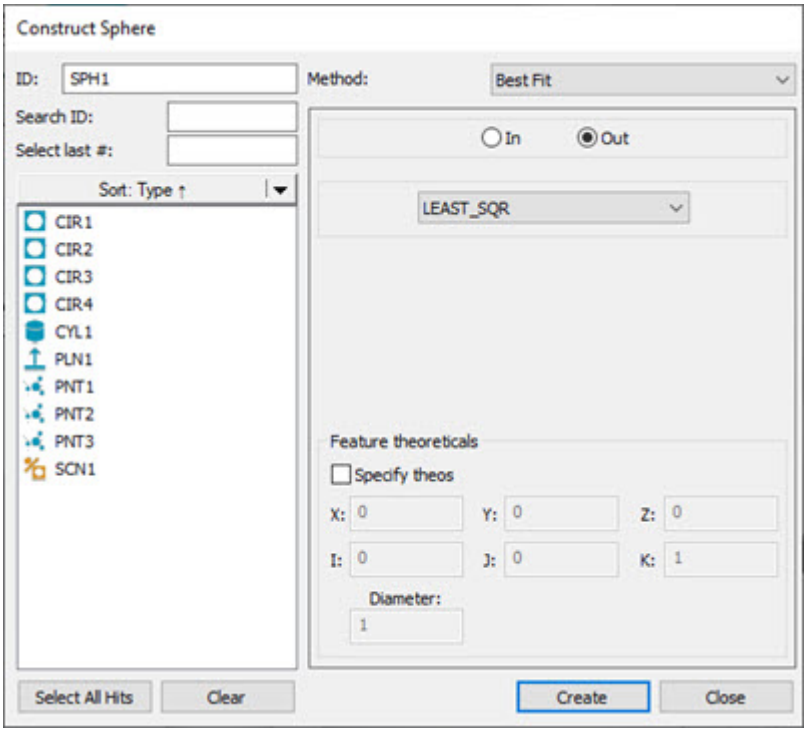
```
feature_name=FEAT/CONE,TOG1,TOG4,ANG THEO/
x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,ang
 ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,ang CONSTR/TOG2,T
OG3,......
```

This example shows the bounded cone format for a constructed cone command in the Edit window with TOG5 changed to LENG.

- Diam_1 is the diameter of the height of the first three hits.
- Diam_2 is the diameter of the furthest point away from the first diameter.
- Length is the distance between the two diameters.

```
feature_name=FEAT/CONE,TOG1,TOG4,LENG THEO/
x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length,diam_1,diam_2
 ACTL/
x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,length,diam_1,diam_2
CONSTR/TOG2,TOG3,......
```

Constructing a Sphere Feature



Construct Sphere Dialog Box

There are several ways to construct a sphere using PC-DMIS. The following table lists the various types of constructed spheres, along with their necessary inputs. Some features may require no inputs while others may require five inputs or more. The term "Any" indicates that the construction can take any type of feature as input for construction. PC-DMIS allows the features to be selected in any order.

Construct Feature Type	Edit Window Symbol	Number of Required Input Features	Primary Feature	Secondary Feature	Comments
Auto Sphere	-	-	-	-	See "Auto Sphere Construction".
Best Fit Sphere	BF	At least 5 inputs are needed.	-	-	Constructs a best fit sphere using the given inputs. See the Note below for recommended inputs.
Best Fit with Recomp Sphere	BFRE	At least 5 inputs are needed. (1 must be a point)	-	-	Constructs a best fit sphere using the given inputs. See the Note below for recommended inputs.
Cast Sphere	CAST	1	Any	-	Constructs a sphere at the centroid of the input feature.
Projected Sphere	PROJ	1 or 2	Any	Plane	One input feature will project the sphere to the workplane.
Reverse Sphere	REV	1	Sphere	-	Constructs a sphere with a reversed vector.
Extracted Sphere	EXTRACTED_SPHERE	1	COP or Mesh	-	Constructs an extracted sphere from the COP or Mesh object at the specified diameter or height.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

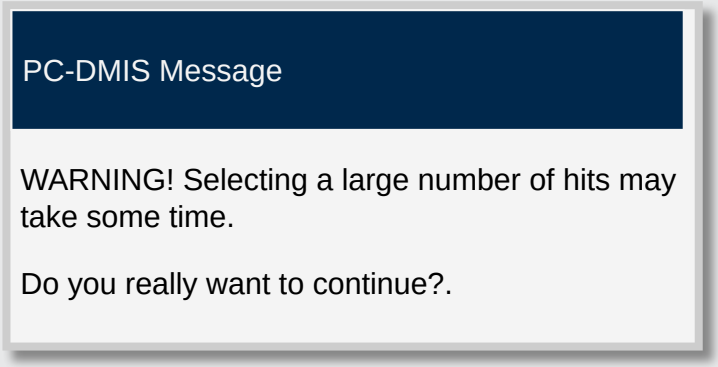
"Cannot construct [feature]. Combination of input features not accepted."

To construct a sphere:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).
2. From the **Method** list, select the method of the constructed sphere. The available options are:
    - Auto Sphere
    - Best Fit
    - Best Fit Recompensate Sphere
    - Projected Sphere
    - Cast Sphere
    - Reversed direction Sphere
    - Extracted Sphere

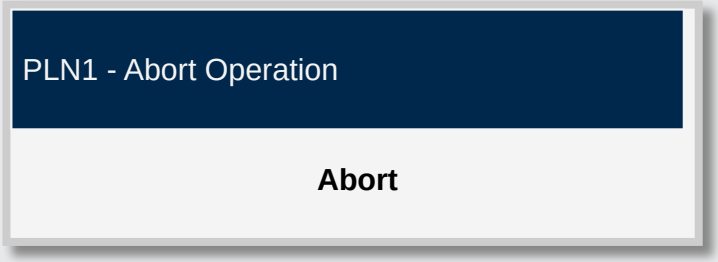
If you select the **Best Fit** or the **BF Recomp** (Best Fit Recomp) method for this feature, PC-DMIS allows you to click the **Select All Hits** button to create the construction from the individual hits of the input features instead of their centroids.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.

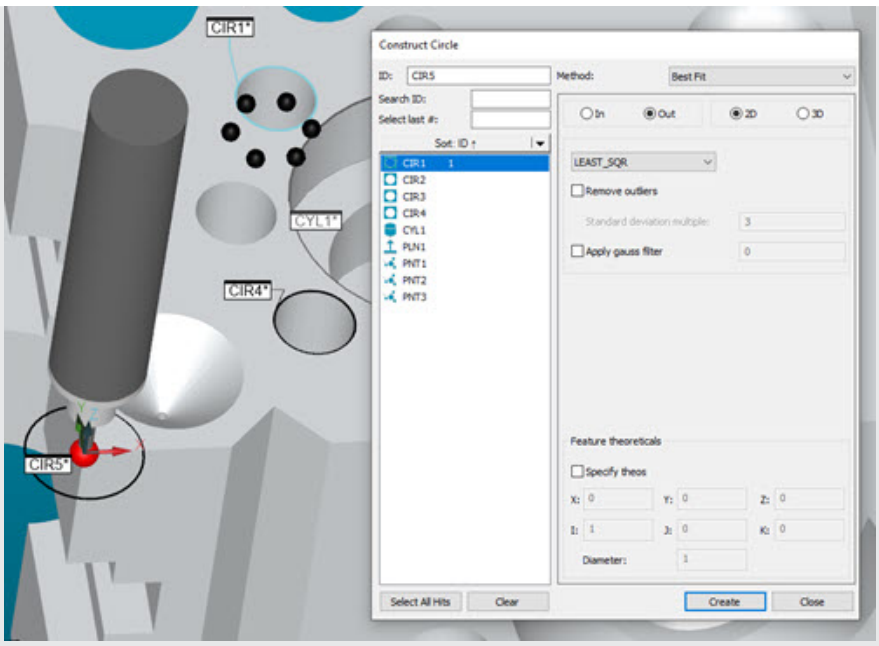


Once all the constructed features are created, the message disappears.

You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

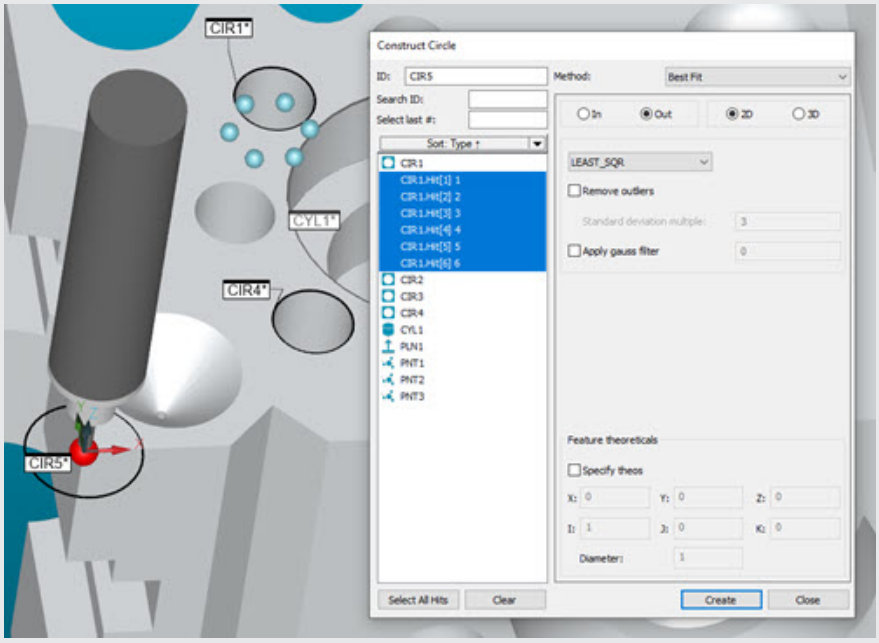
To create constructed features from the individual hits of the input features, do this:

1. From the **Feature list**, select the feature or features that you want to use to create the constructed feature.



Example showing a selected feature prior to clicking the Select All Hits button.

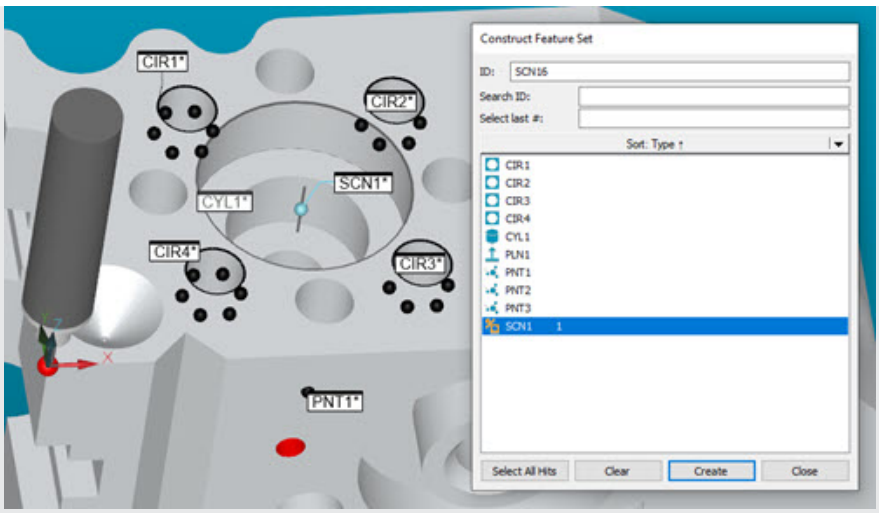
2. Click the **Select All Hits** button to display all the components that comprise the selected feature or features.



Display of items that make up the selected feature are highlighted in the dialog box and in the Graphic Display window.

PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature list** are of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

3. Click the **Create** button to create the Constructed feature based on the final features and feature components that you selected.



Constructed feature created based on the selected items from the Feature list.

3. Use the table above to select the features from the **Feature** list based on the selected method.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.
5. If you selected either the Best Fit or Best Fit Recompensate methods, from the **Best Fit Type** list, select the type of best fit algorithm to use. For details, see the "Best Fit Type" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

The Edit window command line for a sample Sphere Construction would read:

```
feature_name=FEAT/SPHERE, TOG1,TOG4

THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam

ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,diam

CONSTR/ TOG2,TOG3
```

The actual Edit report displays in all capital letters.

TOG1= POLR or RECT

TOG2 = SPHERE

TOG3 = BF / BFRE / CAST / PROJ / REV

TOG4 = IN / OUT

The first three lines that display in the Edit window are the same for constructed spheres. The fourth line is slightly different, according to the type of feature you are constructing. To switch between the different types of spheres, place the cursor on TOG3 and press F7 or F8. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

When two or more features are involved, PC-DMIS automatically determines the necessary order of the input features. This improves the accuracy of the measurement process.

Auto is the default method of construction. This option automatically determines the best way to construct a sphere using the input feature(s). See "Auto Sphere Construction".

The following topics describe the available options for constructing a sphere.

In / Out Sphere

The **In** and **Out** options tell PC-DMIS whether to construct the sphere as an internal or an external sphere.

- If you select **In**, PC-DMIS constructs the sphere as an internal sphere.
- If you select **Out**, PC-DMIS constructs the sphere as an external sphere.

Auto Sphere Construction

The "Input Feature List" table indicates the type of sphere that PC-DMIS constructs when you select the specified inputs and choose the **Auto** option. The order of the selected features is not important. If you choose an incorrect input feature, PC-DMIS displays an error message and does not automatically construct the indicated feature type.

To allow PC-DMIS to automatically determine the best method of construction:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).
2. From the **Method** list, select the **Auto** option.
3. From the **Feature** list, select the desired feature or features based on the "Input Feature List" table below.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.

5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.

6. Click the **Create** button.

Input Feature List

Input Feature(s)	Constructs
Four or more Features	Best Fit Sphere
Any One Sphere	Reverse Sphere
Any One Feature (except Sphere or Set)	Cast Sphere
Any One Set	Best Fit Sphere
Plane + Any Feature	Project Sphere

Constructing a Best Fit or Best Fit Recompensate Sphere

You can construct a "best fit" sphere using five or more features. PC-DMIS computes a *least squares* sphere, one for which PC-DMIS minimizes the average squared distance from the data points to the sphere.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

To construct a Best Fit or a Best Fit Recompensate Sphere:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).

2. From the **Method** list, select either the **Best Fit** or **Best Fit Recompensate** option (shown as **Best Fit** and **BF Recomp**).

3. From the **Feature** list, select five or more features.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.
5. From the **Best Fit Type** list, select the type of best fit algorithm to use. For details, see the "Best Fit Type (for Sphere)" topic in this documentation.
6. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
7. Click the **Create** button.

For a Best Fit constructed sphere, the **Edit** window command line would read:

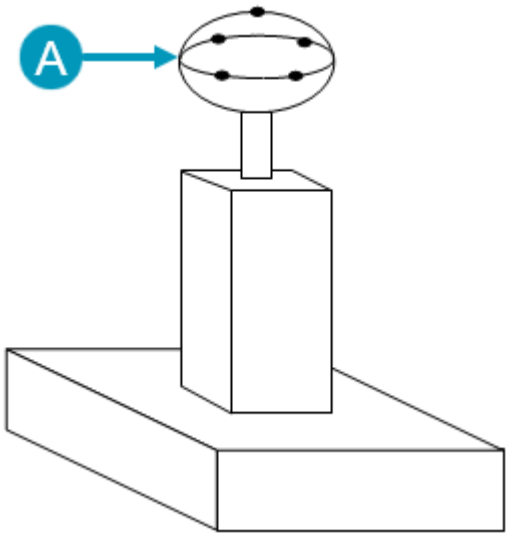
```
CONSTR/SPHERE,BF,feat_1,feat_2,feat_3,feat_4,feat_5...
```

PC-DMIS uses the actual measured points for construction.

For a Best Fit Recompensate constructed sphere, the **Edit** window command line would read:

```
CONSTR/SPHERE,BFRE,feat_1,feat_2,feat_3,feat_4,feat_5...
```

PC-DMIS uses the center of the probe for measurement.



A - *Best Fit Sphere Constructed from Five Points.*
Constructing a Sphere from Five or more Points

Best Fit Type (for Sphere)

A list of Best Fit types becomes available if you select the **Best Fit** or the **BF Recomp** methods when constructing a sphere.

The available Best Fit types for a constructed sphere are:

- LEAST_SQR
- MIN_SEP
- MAX_INSC
- MIN_CIRCSC
- FIXED_RAD
- MIN_BOUND_SPH

These types are discussed below:

LEAST_SQR
Least Squares - This calculation type provides a method of fitting in which the average squared radial distance from the data points to the sphere is minimized. The square root of this quantity is the Root Mean Square (RMS) distance. Since the RMS distance is based on an average, some points may be further than the RMS distance from the computed sphere.

MIN_SEP
Minimum Separation - This calculation type generates a sphere that is halfway between two concentric spheres containing the data points, with the difference of their radii as small as possible. The Min/Max math used by the MIN_SEP calculation minimizes the maximum error, or deviation, from the input data to the sphere. The Min/Max error is one-half of the minimal separation. No input data points (or input features) lie farther than the Min/Max error from the Min/Max sphere. This calculation determines whether or not all the input data (or input features) are within the given tolerances.

MAX_INSC
Maximum Inscribed - This calculation type generates an empty sphere with the largest possible diameter that lies within the data. PC-DMIS first computes a Minimum Circumscribed sphere and requires that the center of the Maximum Inscribed sphere lies within it. This option could be used for an internal spherical feature that requires a mating external sphere. For example, if the input data represents an internal sphere, then this calculation returns a sphere with the diameter of the largest external sphere that will fit inside the internal sphere. Do not use this calculation type for arcs of less than 90 degrees.

MIN_CIRCSC
Minimum Circumscribed - This calculation type generates a sphere with the smallest possible diameter that encloses the input data (or input features). This option could be used when measuring an external sphere that would fit into a mating internal spherical feature. The resulting feature would be the smallest internal sphere into which the external sphere would fit. Do not use this calculation type for arcs of less than 180 degrees.

FIXED_RAD
Fixed Radius - This calculation type creates a sphere of a given diameter, positioned so that the maximal radial distance from the data points to the sphere is minimized. It is similar to the Max/Min math used in the MIN_SEP calculation except that, since the

diameter is known in advance, the radius cannot vary. Only the sphere's position is allowed to vary.

MIN_BOUND_SPH
Minimum Bound Sphere - This calculates the smallest sphere that contains the input points. This is the required math type for ISO 10360 – XX acceptance testing. The input features should be spheres, not points.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

Constructing a Projected Sphere

You can construct a sphere by projecting any feature into the current workplane. PC-DMIS projects the point where the plane intersects the point. If you only have one input feature, it projects the point to the workplane. You should input the desired diameter when projecting a feature into the workplane; otherwise, PC-DMIS uses the probes' diameter.

To construct a projected sphere:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).
2. From the **Method** list, select the **Projection** option.
3. From the **Feature** list, select either one or two features. The first feature can be of any type. The second feature must be a plane.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/SPHERE,PROJ, feat_1,(feat_2)
```

Constructing a Cast Sphere

You can construct a sphere by changing any given feature into a sphere. PC-DMIS constructs the sphere at the centroid of the input feature. If a sheet metal point is used, the diameter is the probe diameter. For some sheet metal features (such as slots and notches), the width is used as the diameter. For features that do not have diameters (lines, points, etc.), a value that is four times the probe diameter is used.

You can modify the diameter of the sphere; this changes the sphere from DEPENDENT to INDEPENDENT. This means that when the sphere is executed that diameter will not change based on the input feature but will be independent of the input feature, while the position and vector remain dependent on the input feature. This allows you to have control over the diameter in cases where the input feature doesn't really have a diameter, such as a point. The DEPENDENT/INDEPENDENT field is a toggle field you may change.

PC-DMIS uses this diameter value for all calculations instead of using the default diameter value as described above.

To construct a Cast Sphere:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).
2. From the **Method** list, select the **Cast** option.
3. From the **Feature** list, select one feature of any type.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/SPHERE,CAST, feat_1,  (Dependent | Independent)
```

Changing the Direction of a Sphere

You can construct a sphere with a reversed vector.

To construct a Reverse Sphere:

1. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**).

2. From the **Method** list, select the **Reverse** option.
3. From the **Feature** list, select one feature. It must be a sphere.
4. Select the **In** or **Out** option. For details, see the "In / Out Sphere" topic in this documentation.
5. If you want to change the feature theoretical values, select the **Feature theoreticals** check box and type in the values. For details, see the "Specifying Feature Theoreticals" topic in the PC-DMIS Core documentation.
6. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/SPHERE,REV, feat_1
```

Constructing an Extracted Sphere

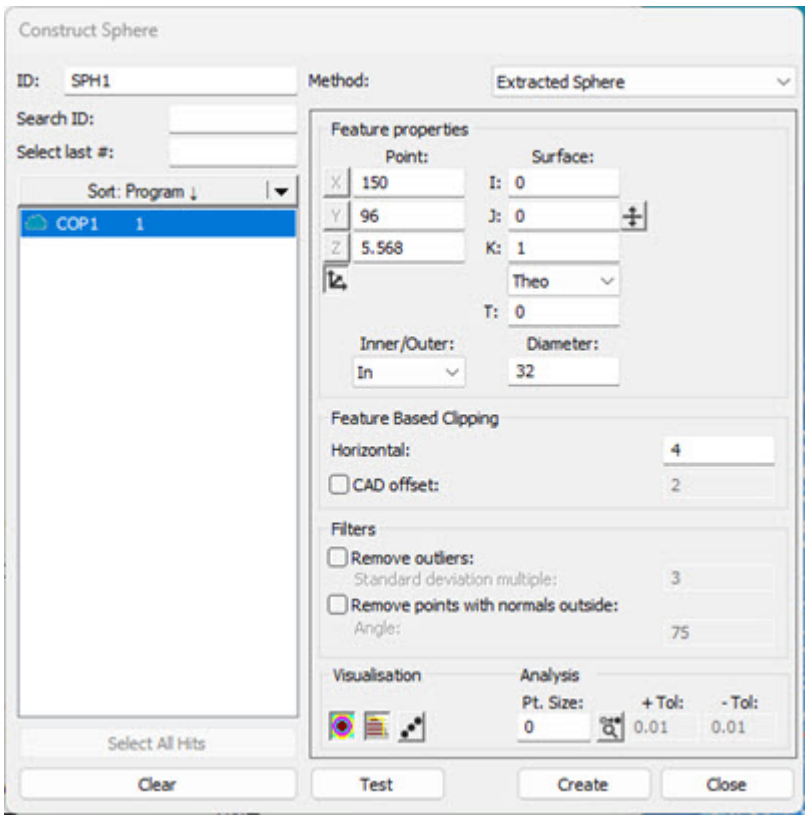
For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

You can construct a sphere that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Sphere** dialog box (**Insert | Feature | Constructed | Sphere**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**).



Construct Sphere Dialog Box - Extracted Sphere Option

3. From the **Method** list, select the **Extracted Sphere** option.

4. From the **Reference** area, select the COP or Mesh that you want to use to extract the sphere from.

5. Click on the CAD model or data to define the nominal or, in the **Point** section of the **Feature properties** area, type the nominal location in the **X**, **Y**, and **Z** boxes.

6. From the **Surface** section of the **Feature properties** area, define the surface vector in the **I**, **J**, and **K** boxes. In the **Angle** section, enter the corresponding vector angle values. You can use the **Material Thickness Type** list and the **T** box below it to type in a material thickness value. For details, see the "Use Thickness" topic in this documentation.

   You can use these controls to perform the associated functions:

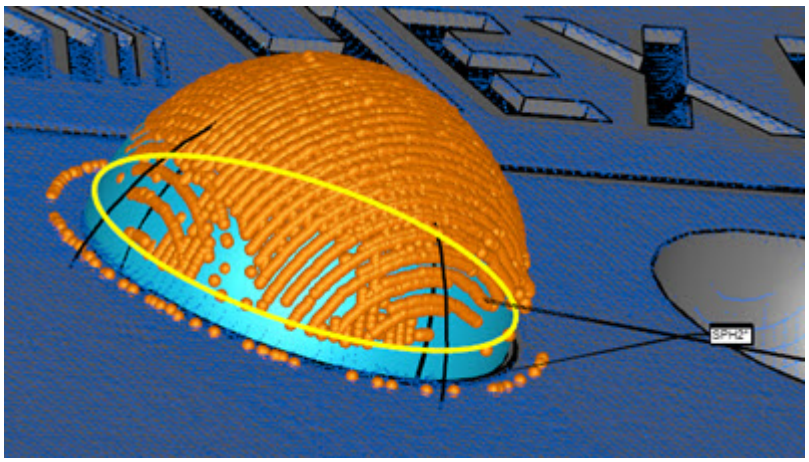
   Flip Vectors

   Polar/Cartesian

   For details on these controls, see the appropriate topic in the "Feature Properties Area" section of the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

7. Select whether the extracted sphere is an inner or outer sphere type from the **Inner/ Outer** list.

8. Type the diameter of the sphere in the **Diameter** box.



Example of a Constructed Extracted Sphere showing the candidate points

9. From the **Feature Based Clipping** area, define the **Horizontal** value. This sets the dimensions for the extraction zone region. Consider part variability when you define the extraction zone.

   Alternatively, you can clip data within an offset boundary around all the CAD elements on a surface with the **CAD offset** option. This is also called *CAD segregation*. For details, see the "CAD offset" section of the "Feature Based Clipping Parameters" topic in the PC-DMIS Laser documentation.

10. If you want to filter out any outlier points, from the **Filters** area, select the **Remove outliers** check box and define the **Standard deviation multiple** to determine which points PC-DMIS excludes as outliers.

11. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

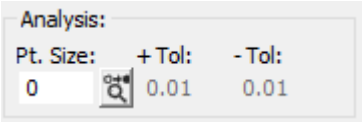
> The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.
>
> For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

12. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.



- The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the "Analysis Area" topic in the PC-DMIS Core documentation.

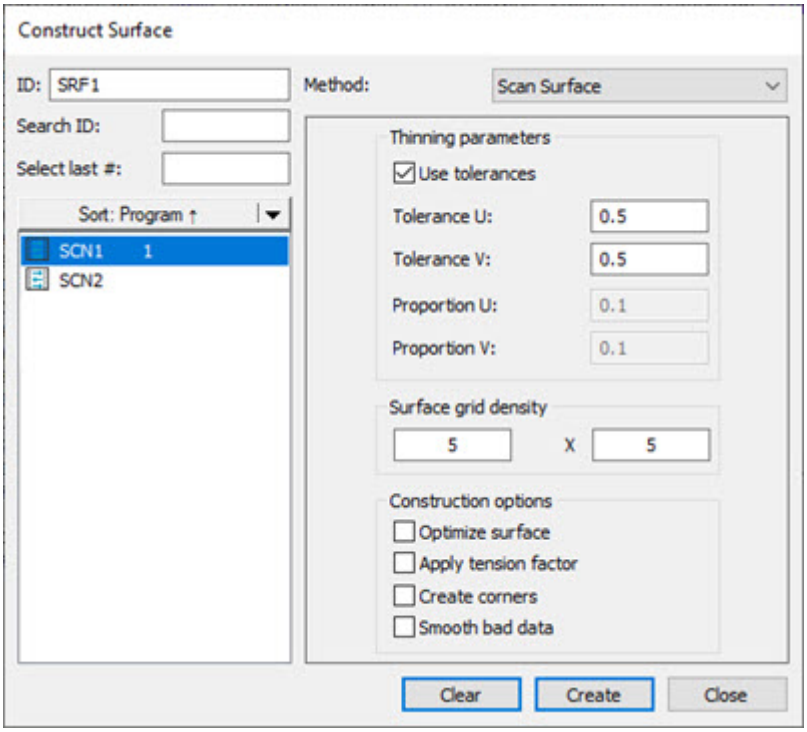
13. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

14. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

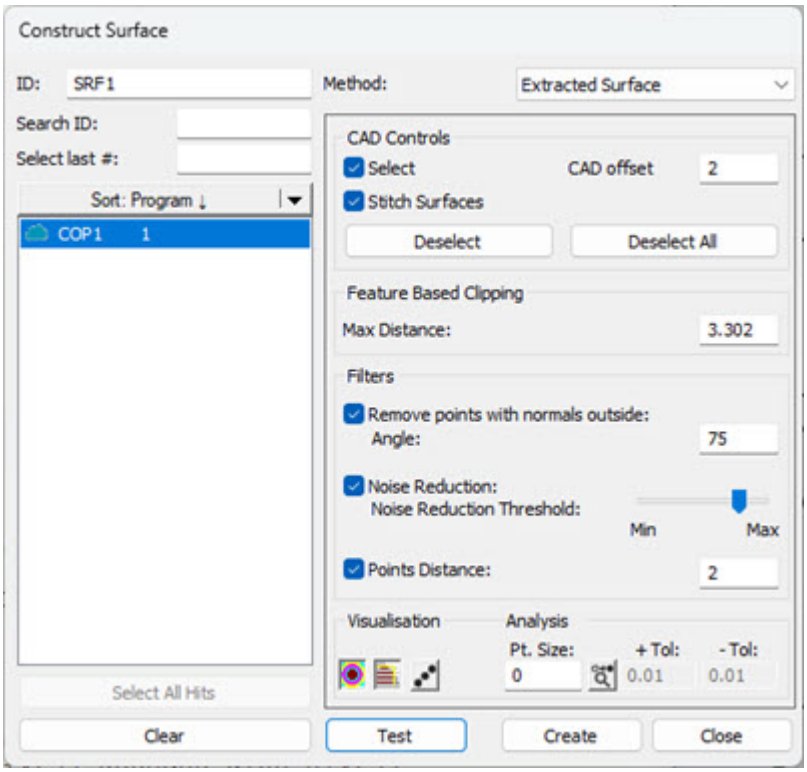
    PC-DMIS creates the command in the Edit window:

```
SPH1        =FEAT/SPHERE,CARTESIAN,OUTER
             THEO/
<168.5,45.66,0>,<0.0021447,0.0058276,0.9999807>,12.7
             ACTL/
<168.586,45.406,0.05>,<0.0021447,0.0058276,0.9999807>,12
.781
             THEO_THICKNESS,0
             USE CAD SEGREGATION=ON,CAD OFFSET=1
             HORIZONTAL CLIPPING=3
             USE OUTLIER REMOVAL=OFF
             REMOVE POINTS WITH NORMALS OUTSIDE=OFF
             CONSTR/SPHERE,EXTRACTED,REF=COP1
```

Constructing a Surface



Construct Surface Dialog Box - Scan Surface Method



Construct Surface Dialog Box - Extracted Surface Method

There are two types of constructed surfaces available in PC-DMIS: **Scan Surface** and **Extracted Surface**.

You typically use the **Scan Surface** for reverse engineering and can be dependent or independent.

Use the **Extracted Surface** for reporting surface profile dimensions and is always dependent.

The following table shows the two surfaces, along with their necessary inputs. The only input that the **Scan Surface** takes is a patch scan. The scan must contain at least two rows of four points per row. The only inputs that an **Extracted Surface** takes are a COP or Mesh object.

Construct Feature Type	Edit Window Symbol	Number of Input Sets	Input Set	Comments
Scan Surface	DEPENDENT	1	Patch scan containing at least 2 rows with 4 points per row	The surface will update as the input feature changes.
	INDEPENDENT	1	Patch scan containing at least 2 rows with 4 points per row.	This uses the input feature for construction only.
Extracted Surface	CONSTR/ SURFACE,EXTRACTED	1	COP or Mesh object	Constructs an extracted surface from the COP or Mesh object.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

To construct a Surface:

1. Select the **Insert | Feature | Constructed | Surface** menu option to open the **Construct Surface** dialog box.

2. Select the type of constructed surface (**Scan Surface** or **Extracted Surface**) from the **Method** list.
3. Select the appropriate feature from the **Feature** list:
    - For the **Scan Surface** method, select the patch scan.
    - For the **Extracted Surface** method, select the Pointcloud or Mesh.
4. Select the various construction options. For details on the options available, see the following topics in this section of the PC-DMIS Core documentation.
5. Click the **Create** button.

The Edit window command line for a Constructed Surface using the **Scan Surface** method would read:

```
feature_name=FEAT/SURFACE,TOG1,CONTROL POINTS U,
             CONTROL POINTS V, NUM POINTS FIT,TOG2
             THINNING PARAMETER U, THINNING PARAMETER V
             CONSTR/SURFACE, INPUT TYPE, INPUT ID
```

The actual Edit report displays in all capital letters.

DEPENDENT is the default construction type for the Scan Surface method. You can use the TOG1 parameter to switch between Dependent and Independent.

TOG1 = Dependent or Independent.

TOG2 = Tolerance or Proportion

The Edit window command line for a Constructed Surface using the **Extracted Surface** method would read:

```
SRF1=FEAT/SURFACE,THEO_THICKNESS=0,MAX DISTANCE=2
     CAD OFFSET=2,REMOVE POINTS WITH NORMALS OUTSIDE=ON,75
     NOISE REDUCTION=ON,20,DISTANCE=ON,2
     CONSTR/SURFACE,EXTRACTED,REF=COP1,SIZE=0
```

Notice that there is no Dependent or Independent toggle for an Extracted Surface. This is because the Extracted Surface is always dependent – it is always associated with the COP or MESH you use to create it and updates if you change the input.

The following topics describe the available options for constructing a surface:

Constructing a Dependent/Independent Scan Surface

All constructed scan surfaces start out as dependent surfaces and must be created from a single input -- a patch scan. The patch scan must contain at least two rows with four points per row. PC-DMIS uses thinning tolerances to control the tightness of the surface fit.

- **Small tolerances:** If the thinning tolerances are small, the algorithm attempts to fit the surface through all of the points in the scan.
- **Large tolerances:** If the thinning tolerances are large, the surface is more of an approximation to the scan. The best way to see this is to construct a surface, change the thinning tolerances, and see how the shape of the surface changes.

The smaller the thinning tolerance, the longer it takes to create the surface. Be aware that the small tolerances (.01 - .05) may take a considerable amount of time (one hour) to create the surface if the input scan is large or not well behaved. Valid thinning tolerances range from 0.01 to 5.0 with a default of 0.5.

You can control the appearance of the surface with the surface grid density values. The surface displays as an N x M mesh of polylines with the default being a 5x5 mesh and the lowest value being a 2x2 mesh. To make a dependent surface independent so that it is no longer associated with the input scan, change the DEPENDENT field in the Edit window.

The shape of the surface cannot be changed.

To construct a Dependent or Independent Surface:

1. Open the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**).
2. Select the **Scan Surface** option from the **Method** list.
3. Set the value for the **Thinning Tol U:** box.
4. Set the value for the **Thinning Tol V:** box. This applies a tolerance value to the V axis.
5. Set the values for the surface grid density.

6. Select the desired construction options. These include:
    - Optimize Surface
    - Apply Tension Factor
    - Create Corners
    - Smooth Bad Data
7. Select a patch scan feature set that contains at least two rows with four points per row.
8. Click the **Create** button.

Thinning Parameters (Thinning Tolerance or Thinning Proportion)

You can use one of two types of thinning parameters: Thinning Tolerance or Thinning Proportion. The **Use tolerance** check box on the **Construct Surface** and **Construct Curve** dialog boxes (**Insert | Feature | Constructed | Surface** and **Insert | Feature | Constructed | Curve**, respectively) lets you switch between either tolerance or proportion:

- *Thinning tolerance* controls the tightness (or exactness) of the curve or surface fit. Valid thinning tolerances range from 0.0 to 5.0, with the default being 0.01. The smaller the thinning tolerance, the closer the curve comes to passing through the centroids of the features included in the input set. If the thinning tolerance is 0.0, the curve or surface will pass through all of the centroids. A larger thinning tolerance will result in a curve or surface that has fewer fluctuations (at the expense of not lying near the input set's features). To see this, construct a curve or surface and then change the input tolerance and examine how its shape changes.
- *Thinning proportion*, alternately, may be used to control the quality of the fit. Valid thinning proportions range from between 0.0 and 1.0, with a default value of 0.33. The thinning proportion determines the number of degrees of freedom available in fitting the curve or surface to the centroids. At the lower extreme of 0, the algorithm will attempt to fit a straight line or plane to the centroids. At 1, it will compute a fit that passes through all of the centroids.

To make a DEPENDENT curve an INDEPENDENT curve (so that it is no longer associated with the input set):

1. Open the Edit window (**View | Edit Window**).
2. Select the curve feature that you constructed.
3. Navigate to the DEPENDENT field of that feature.
4. Press F7. It will change from DEPENDENT to INDEPENDENT.

    You can change the shape of the curve by editing its control points.

Thinning Parameter U

This box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) enables you to set a thinning parameter value applying to the U axis of the surface.

Thinning Parameter V

This box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) enables you to set a thinning parameter value applying to the V axis of the surface.

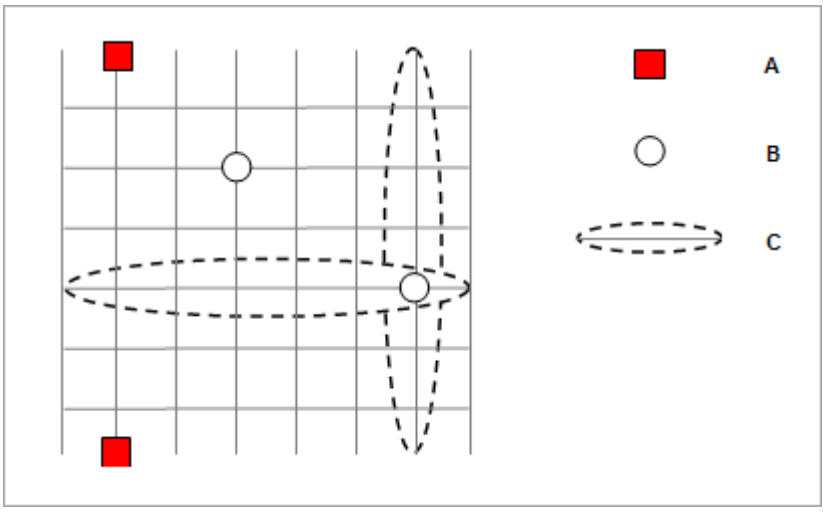
Surface Grid Density

This box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) enables you to set the density of a surface scan. The higher the values, the more splines there will be.

Optimize Surface

Control Points are points that mark begin and end points of splines in a surface grid.

This check box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) attempts to optimize the knots and control points of the initial splines from which the surface is constructed.



Example showing the surface grid with control points (A), knots (B), and splines (C).

Apply Tension Factor

Surfaces that you create by using the **Apply tension factor** check box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) tend to be tauter (tighter) and shorter, but may fit the data slightly less closely.

Create Corners

This check box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) enables corners to be added to the surface at areas where the data seems to make an abrupt change in direction.

Smooth Bad Data

This check box in the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) attempts to throw out bad data. Bad data is evaluated as data that has an abrupt change in direction. This option is almost the opposite of the Create Corners option above.

Constructing an Extracted Surface

For details on extracting Auto features from a Mesh, see the "Extracting Auto Features from a Mesh" topic in the PC-DMIS Laser documentation.

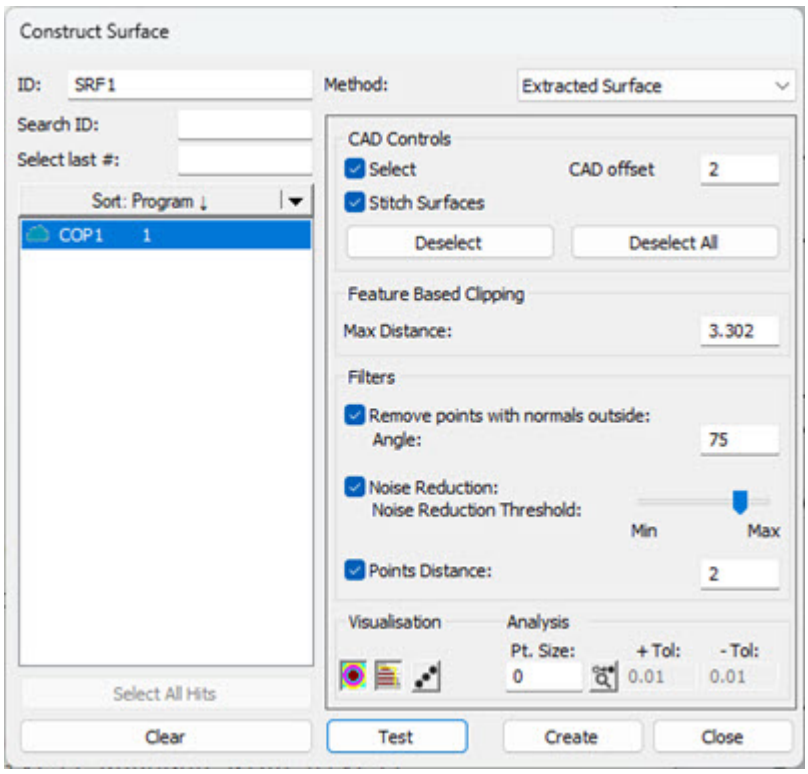
For details on extracting Auto features from a Pointcloud (COP), see the "Extracting Auto Features from Pointclouds" topic in the PC-DMIS Laser documentation.

A Constructed Extracted Surface is a collection of segregated points related to a surface on the CAD model. It does not create a CAD surface. Its intended use is for dimensioning purposes only.

You can construct a surface that is extracted from a scanned pointcloud (COP) or mesh.

To do this, follow these steps:

1. Ensure that your measurement routine has a Pointcloud (COP) or Mesh command.
2. Open the **Construct Surface** dialog box (**Insert | Feature | Constructed | Surface**) or from the **Constructed Feature** toolbar (**View | Toolbars | Constructed Features**) and select the **Extracted Surface** option from the **Method** list.



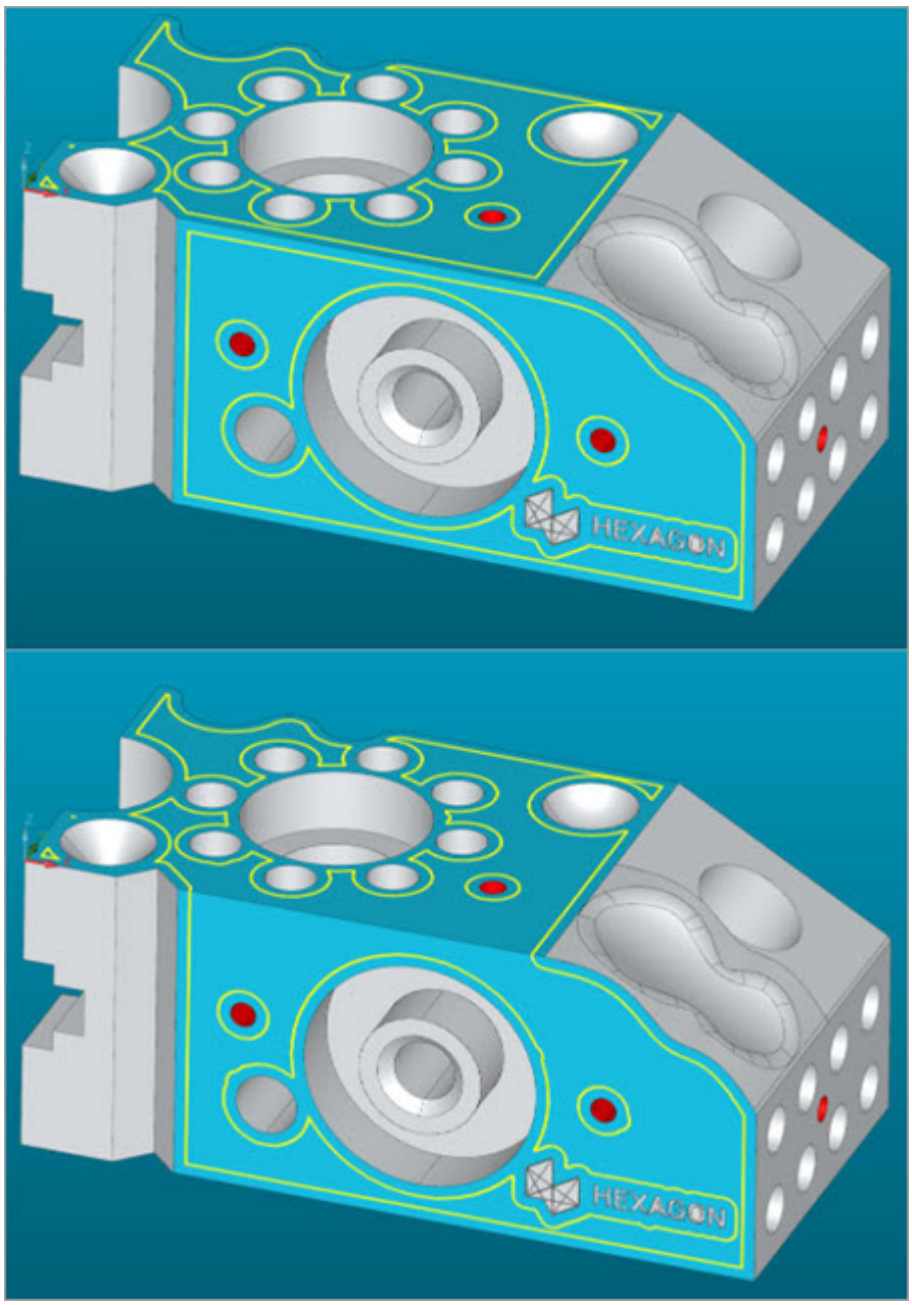
Construct Surface dialog box for Extracted Surface method

3. From the **Reference** area, select the COP or Mesh that you want to use to extract the surface from.

4. From the **CAD Controls** area, select the **Select** check box to select the CAD surface(s) in the Graphic Display window that you want to use to extract the Constructed Surface from.

5. You can clip data within an offset boundary around all the CAD elements on the selected surface(s) with the **CAD offset** option in the **CAD Controls** area. This is also called *CAD segregation*. For details, see the "CAD offset" section of the "Feature Based Clipping Parameters" topic in the PC-DMIS Laser documentation.

6. You can stitch surfaces together with the **Stitch Surfaces** option. If you select multiple contiguous surfaces on the CAD model, the software stitches these surfaces together and processes them as one surface.



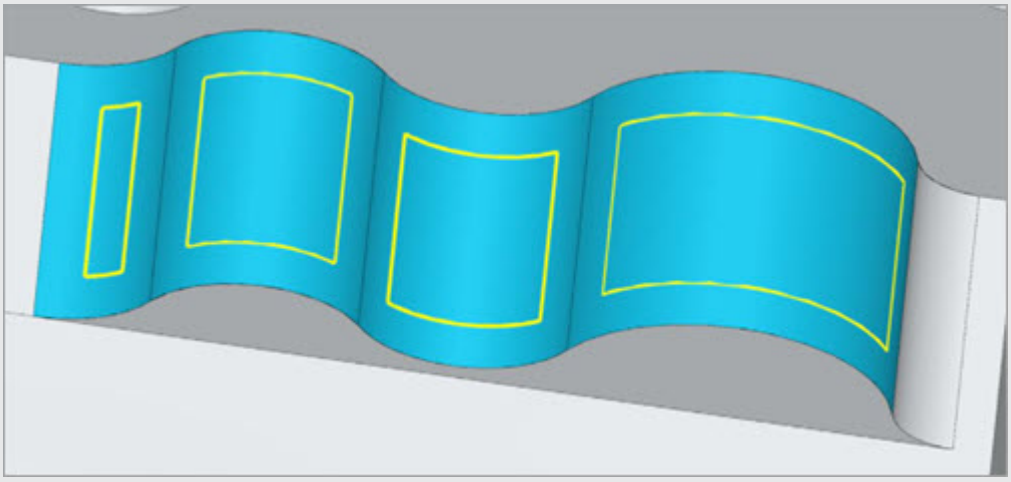
Examples before (top) and after (bottom) applying the Stitching Surfaces option

The advantage of stitching surfaces together is that you would get one continuous pointcloud rather than one with gaps where the surfaces meet if they were processed as individual surfaces.

This is especially evident with the Extracted Surface command. The main aim of the Extracted Surface command is to provide a way for you to report the profile of a surface. Stitching ensures that there is no loss of data where the surfaces meet.

If you want to check the transition between the surfaces of each of the curves on a CAD model, stitching provides a way for you to detect whether there was a smooth transition or a step.

In the image below, you can see that without stitching, there are gaps (twice the CAD offset) between each curve:



7. In the **Feature Based Clipping** area, type the **Max Distance** value. This is the maximum distance in measurement routine units that a point can be from the CAD model to be considered a valid point.

8. If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area, select the **Remove points with normals outside** check box and type the value in the **Angle** (Max Incidence Angle) box.

The Max Incidence Angle filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed Extracted Cone, Cylinder, Plane, Sphere, Surface Point), the filter now compares the estimated normal of the pointcloud point to the nominal normal of the feature, which greatly improves the results. 2D laser features remain unchanged, using the legacy incidence angle filter method.

For details on the **Filters** area, see the "Filters" topic in the PC-DMIS Laser documentation.

9. In the **Filters** area, if you want to filter out the noise in your data, select the **Noise Reduction** check box and use the **Noise Reduction Threshold** slider to set the level of noise reduction you want to apply.

10. In the **Filters** area, you can set the minimum distance a point can be from any other point before PC-DMIS discards it. If the distance between a point and any of its neighboring points is less than this value, the point is discarded. To do this, select the **Points Distance** check box and set a value in the box.

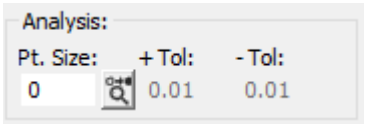
    Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) the extracted constructed surface and projects it to the surface.

11. In the **Visualisation** section, PC-DMIS provides these options to perform the described function:

    Visualisation Tools On/Off - This button toggles the display of the colored visualisation tools. For more information, see "Understanding the Visualisation Tools".

    Show/Hide segregated points - This button toggles the display of the points that the software passes to the feature extractor engine based on the current settings.

    Show/Hide measured points - This button toggles the display of the measured points. It remains greyed out until you click **Test** or **Create**. For more information, see the "Show Measured Points" topic in the PC-DMIS Core documentation.

    - The **Analysis** area allows you to determine how each measured hit/point is displayed. For more information, see the “Analysis Area” topic in the PC-DMIS Core documentation.

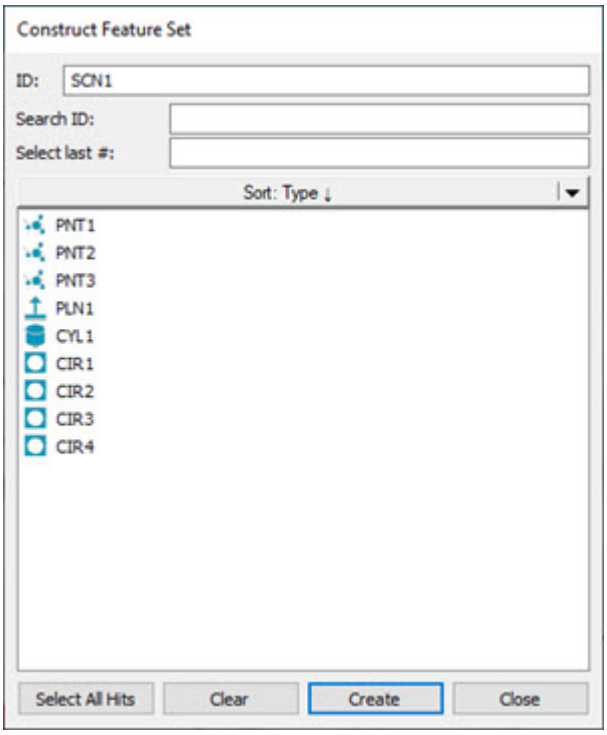
12. Click the **Test** button to test a feature's creation and preview its dimensional data before you create it. PC-DMIS performs a measurement using the current parameters. You can change the parameters and click **Test** repeatedly until you have an acceptable measurement. Then, when you click **Create**, the software converts the temporary feature into a normal feature in the measurement routine.

13. Click the **Create** button. Based on the parameters that you specified in the dialog box, PC-DMIS does an analysis of the candidate points and returns (or extracts) each measured point and projects them to the surface.

    PC-DMIS creates the command in the Edit window:

```
SRF1 =FEAT/SURFACE,THEO_THICKNESS=0,MAX DISTANCE=1
      CAD OFFSET=2,STITCH SURFACES=YES,REMOVE POINTS
WITH NORMALS
      OUTSIDE=OFF
      NOISE REDUCTION=OFF,DISTANCE=OFF
```

Constructing a Set of Features



Construct Feature Set Dialog Box

The **Set** menu command allows you to construct a set of features. You can do this by selecting (or keying in) all the features to be used in the set. When you click the **Create** button, PC-DMIS averages all of the input centroids and displays a set marker along with the new ID for the set.

If you select inappropriate feature types, PC-DMIS displays this message on the Status bar:

"Cannot construct [feature]. Combination of input features not accepted."

To construct a feature set:

1. Access the **Construct Feature Set** dialog box (**Insert | Feature | Constructed | Set**).
2. Select the features you want in your feature set. Optionally, if you click the **Select All Hits** button, PC-DMIS allows you to create the constructed Feature Set from the individual hits of the input features instead of their centroids. For details, see the "Using the Select All Hits Button to Construct a Feature Set" in the PC-DMIS Core documentation.
3. Click the **Create** button. The new feature set is given a feature ID and is placed in the Graphic Display window.

The Edit window command line for a sample Surface Construction would read:

```
 feature_name=FEAT/SET,TOG1,
THEO/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,
ACTL/x_cord,y_cord,z_cord,i_vec,j_vec,k_vec,
CONSTR/TOG2,feat_1, feat_2, feat_3...
```

TOG1= POLR or RECT

TOG2 = SET

The first three lines that display in the Edit window are the same for constructed sets. The fourth line is slightly different, according to the number of features used in the set.

The next few topics cover uses for feature sets in PC-DMIS and how to use feature hits to create a feature set.

Profile Error from a Set

If you are using CAD data, you can construct a set from measured points on a surface. When you ask for the PROFILE of the feature set, PC-DMIS reports the zone between the minimum normal to surface error and the maximum normal to surface error. (See "Dimensioning Surface or Line Profile" in the "Using Legacy Dimensions" chapter for more information.)

Average Values from a Set

When you make a set from input features, PC-DMIS averages the X, Y, and Z values of the input features. For example, the set can be used to get the average Z value of a series of measured points.

Using a Range of Hits from a Scan as Inputs

You can use a range of hits from an existing scan for your feature set inputs instead of selecting individual features.

To do this:

1. Access the **Construct Feature Set** dialog box (**Insert | Feature | Constructed | Set**).
2. Select a scan to use as the input.
3. Click **OK**. The command appears in the Edit window.
4. Place the Edit window into Command Mode.
5. Navigate to the feature set command in the Edit window.
6. Select the scan ID on the CONSTR/SET command line.
7. Modify the scan ID to take a range of hits, using a syntax like this:

   ```
   <ID>.HIT<STARTHIT>..<ENDHIT>
   ```

   <ID> - Specifies the ID of the scan.

   <STARTHIT> - Number that specifies the beginning hit in the range of hits.

   <ENDHIT> - Number that specifies the ending hit in the range of hits.

For example, the following code shows a constructed set using hits 1 through 10 of a scan named SCN1 for its input features.

```
SET1=FEAT/SET,RECT
THEO/2.2953,3.7467,0.95,0,0,1
ACTL/2.2953,3.7467,0.95,0,0,1
CONSTR/SET,BASIC,SCN1.HIT[1..10],,
```

You can use an actual expression (similar to the input code) to assign the X values of the first five features of the constructed set into an array. For example, the following code would assign just the X values of the first five hits to variable V2 and then display the values in an operator comment.

```
ASSIGN/V2=SET1.HIT[1..5].X
COMMENT/OPER,YES,V2 is:
,V2
```

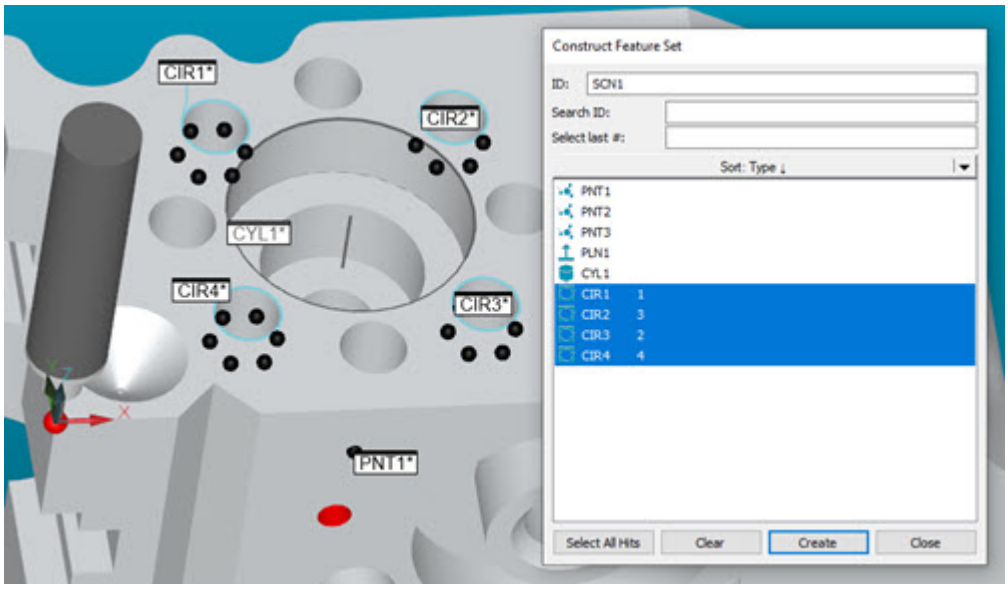
For information on using expressions to return a range of hits as an array, see the "Hit Arrays" topic in the "Using Expressions and Variables" chapter.

Using the Select All Hits Button to Construct a Feature Set

If you click the **Select All Hits** button, PC-DMIS allows you to create the constructed Feature Set from the individual hits of the input features instead of their centroids.

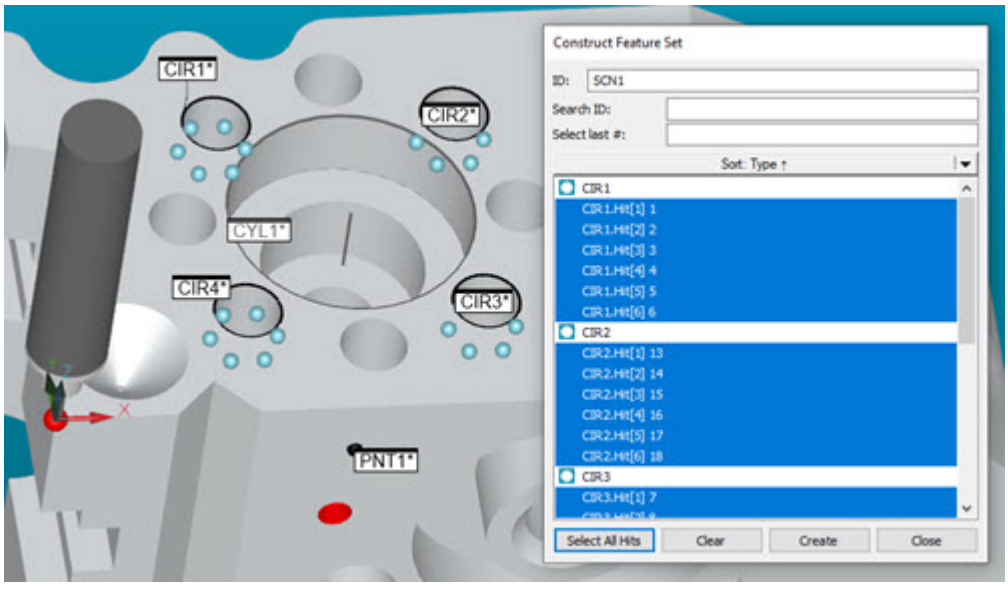
To do this:

1. From the **Feature** list, select the feature or features that you want to use to create the constructed Feature Set.



Example showing the selected features prior to clicking the Select All Hits button

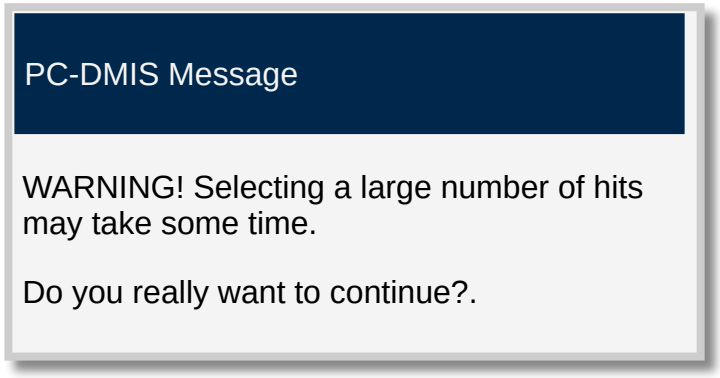
2. Click the **Select All Hits** button to display all the components that comprise the selected features.



Display of items that make up the constructed Feature Set are highlighted in the dialog box and in the Graphic Display window

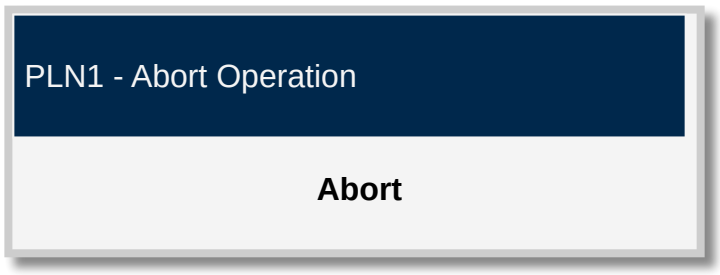
PC-DMIS displays and highlights all the components of the selected feature (or features) in the **Feature** list of the dialog box. You can select or de-select any of the features or feature components shown in the list to include or exclude them.

If there are a large number of points involved (about 10,000 or more), PC-DMIS shows a message asking if you are sure you want to proceed as this may take some time to complete.



Click **Yes** to proceed, click **No** to abort the process. You can select the **Don't ask me again** check box to prevent this message from recurring.

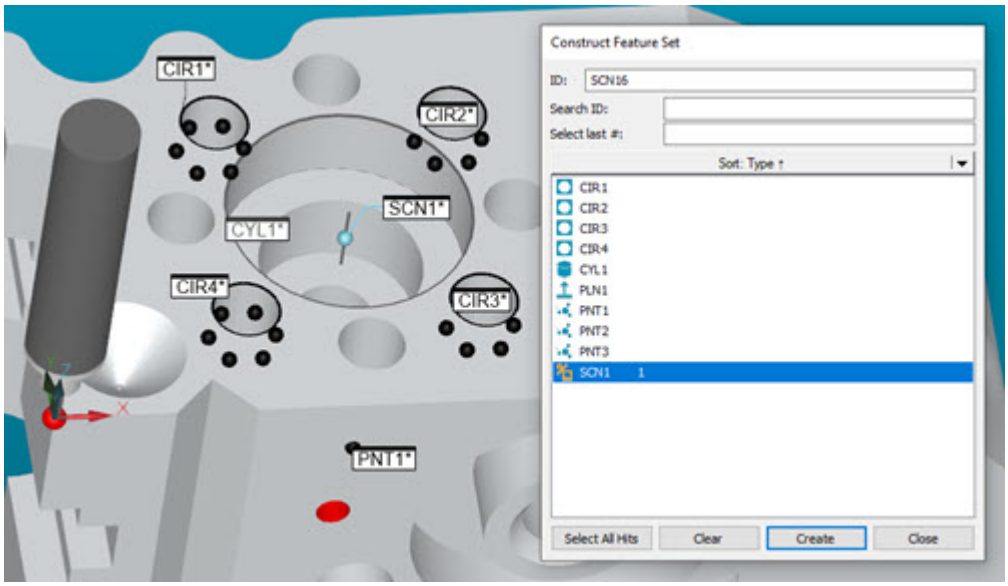
If you click **Yes**, another message is shown similar to the one shown below to let you know that PC-DMIS is generating the features from the hit points.



Once all the constructed features are created, the message disappears.

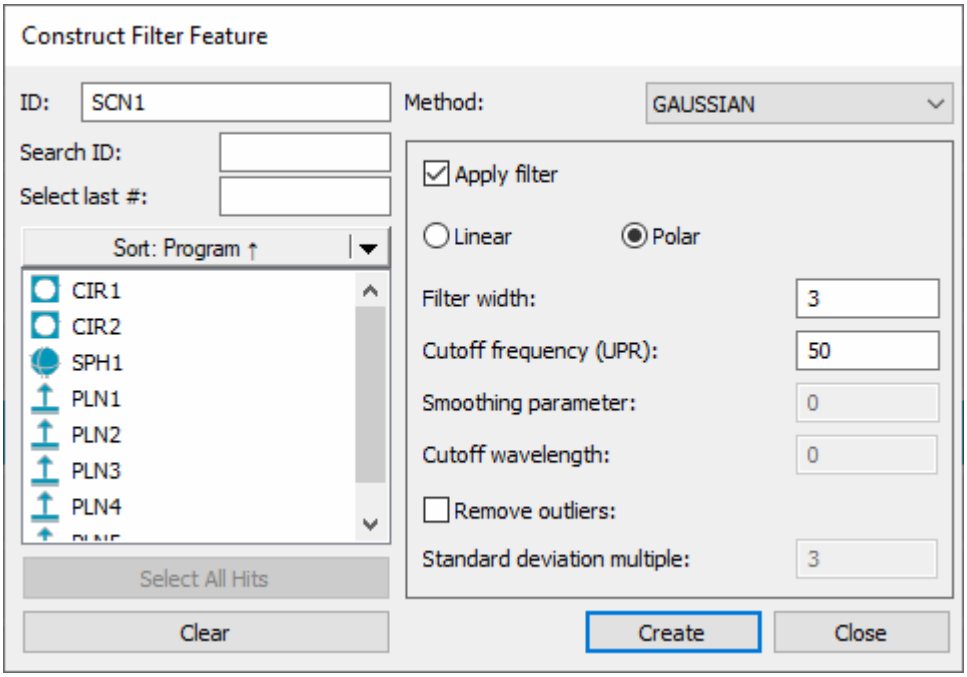
You can click the **Abort** button at any time to stop the process. Any features created up to this point are listed in the **Feature list** area of the dialog box.

3. Click the **Create** button to create the constructed Feature Set based on the final features and feature components that you selected.



Constructed Feature Set created based on selected items from the Feature list

Constructing a Filter Set



Construct Filter Feature Dialog Box

This command allows you to construct a filter set from a scan, certain constructed features, or another filter set. To do this, select (or key in) the input feature, the type of filter (method) desired, and the parameters to be used with the filter. When you click the **Create** button, PC-DMIS applies the filter routine to the data in the input feature and displays a set marker along with the new ID for the set.

This command is typically used to smooth ball center data from a scan. PC-DMIS applies a lowpass Gaussian or another lowpass filter to smooth the data.

If you select inappropriate feature types, PC-DMIS displays an error message that says it cannot construct the feature.

To construct a filter set:

1. Open the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**).
2. Select an input for the filter set.
3. Select either the **Polar** or **Linear** option to smooth circle or linear data, respectively.
4. Select the type of filter from the **Method** list.
5. Type in values for any of the filter parameters.

6. If you want to remove outliers before filtering, select the **Outlier Removal** check box and the **Standard Deviation Multiple** check box.
7. Click the **Create** button.

The Edit window command line for this option would read:

```
CONSTR/SET/FILTER,TOG1,TOG2,feat_1,
VAL1,VAL2,OUTLIER_REMOVAL/TOG3,VAL3
```

For example:

```
CONSTR/SET,FILTER,GAUSSIAN,POLAR,SCN1,
WIDTH=3,UPR =50,OUTLIER_REMOVAL/ON,3
```

TOG1 = `ISO_16610_GAUSSIAN / SPLINE / UNIFORM / TRIANGLE / CYLINDRICAL / GAUSSIAN`

TOG2 = `POLAR / LINEAR`

Feat_1 = This is the input feature for this filter.

VAL1 = Filter width.

VAL2 = Cutoff frequency in undulations per revolution (UPR).

TOG3 = This value switches between ON or OFF. It determines if outliers should be removed before filtering.

VAL3 = Standard deviation multiplier. If *TOG3* is ON then all the points in the input that have more than this number of standard deviations from the least squares substitute feature (for example, circle, or line) will be removed before filtering.

Linear option

The **Linear** option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) enables you to filter the data in a feature that is not a circular scan. The deviations affected in this case are perpendicular to the workplane.

For the linear filters, the number of points in the filter set may be less than the number of input points. PC-DMIS removes points at both ends for which not enough data (to the left or right) is available. To compute a valid output point, see "Filter Width box".

Polar option

The **Polar** option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) enables you to apply a filter to data in a circular scan. PC-DMIS assumes that the data is periodic (meaning it forms one complete, closed circle). The deviations affected in this case are radial deviations.

Apply Filter Check Box

The **Apply filter** check box in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) allows you to construct the feature set independent of whether or not the filtering is applied to the data. Use this check box to choose to remove outliers, for example, but not filter the data.

Method List

The **Method** list allows you to select from the following filter types:

- ISO_16610_GAUSSIAN
- SPLINE
- UNIFORM
- TRIANGLE
- CYLINDRICAL
- GAUSSIAN

If you select GAUSSIAN, UNIFORM, or TRIANGLE then the workplane is critical. For these three filter types, the following applies:

- If you select the **Linear** option, the filtering occurs normal (i.e., perpendicular) to the workplane.
- If you select **Polar** option, then the filtering occurs in a radial direction within the workplane.

ISO_16610_Gaussian

Starting with PC-DMIS 2025.1, the **ISO_16610_GAUSSIAN** filter option is available. This filter option is intended to supercede the **GAUSSIAN** filter option. The older version of the **GAUSSIAN** filter option was written in accordance with ISO 11562:1996 which was later cancelled and replaced by ISO 16610-21:2011. PC-DMIS retains this older **GAUSSIAN** filter option for migration purposes. Starting with PC-DMIS version 2025.1, we recommend that you select the **ISO_16610_GAUSSIAN** filter option when you create new measurement routines. You can not back-save measurement routines that use the newer **ISO_16610_GAUSSIAN** filter option to versions of PC-DMIS earlier than 2025.1.

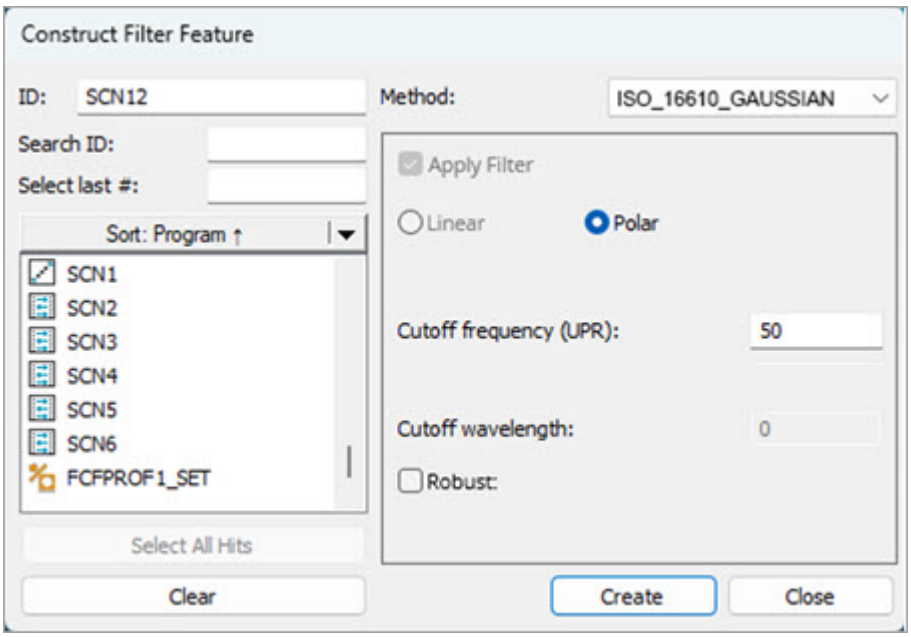
The newer **ISO_16610_GAUSSIAN** filter option combines concepts from these standards in the ISO 16610 series:

- ISO 16610-21
- ISO 16610-28
- ISO 16610-31
- ISO 16610-61
- ISO 16610-68 (not published yet)
- ISO 16610-71

This combination allows the following capabilities that were not previously available:

- You can now filter any kind of surface, not just 2D lines and 2D circles.
- Outlier handling is now standards compliant.
- Points can be irregularly spaced.
- Points are not required to be in any kind of order.
- Both linear and areal datasets are acceptable.

You can access the newer **ISO_16610_GAUSSIAN** filter option from the **Constructed Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**). This newer version of the filter smooths your data by applying a Gaussian lowpass filter to it according to the ISO 16610 collection of standards.

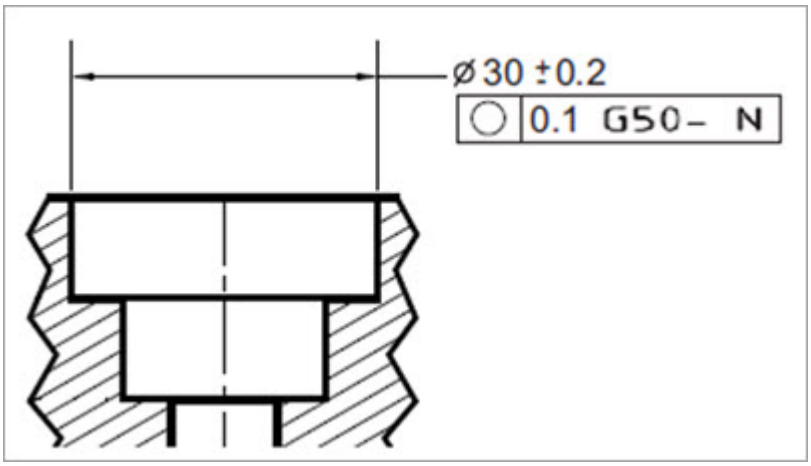


Construct Filter Feature dialog box showing the ISO_16610_GAUSSIAN filter method selected

The **Apply Filter** option is always active for this filter type. It is grayed out so it can't be deselected.

You can either type a **Cutoff frequency** (Polar) or **Cutoff wavelength** (Linear) to control the amount of smoothing.

The following example shows how the **Cutoff frequency** or **Cutoff wavelength** might be defined on the print.



In this example, a roundness specification applies relative to the minimum circumscribed reference feature (N), after the application of a Gaussian longwave pass filter with a cutoff value of 50 UPR (G50). To satisfy this requirement in PC-DMIS, you should:

- Gather surface data with a sufficient point density to satisfy the filter requirements (see "Points to Note"). You could use:
  - An Auto Cylinder with the adaptive concentric circle scan strategy, or
  - A series of linear closed or basic circle scans, or

- Extract the data from a pointcloud using a laser Auto Cylinder or a Constructed Extracted Cylinder

> Multiple individual scans must be combined into a constructed feature set before filtering.

- Construct an ISO_16610_GAUSSIAN filter set from the cylinder's hits. Set the filter type to **Polar** and type 50 in the **UPR** box. Decide whether to apply the **Robust** option or not. In most cases, Hexagon recommends that you apply this option.
- Construct a BFRE Cylinder using the ISO_16610_GAUSSIAN filter feature as the input.
- Create an ISO 1101 Geometric Tolerance Circularity command for your constructed BFRE Cylinder. Type 0.1 for the tolerance and select **N** from the **Reference Feature Association** list.

The title block might show a general filter requirement ISO 1101 TF:G0,8-x50. This means that for all tolerance features (TF), an ISO_16610_GAUSSIAN filter with a cutoff wavelength of 0.8 should be applied to all open profiles (Linear) and a 50 UPR frequency should be applied to all closed profiles (Polar). For more information, refer to ISO 1101:2017 section 8.6.

In some cases, neither the print nor the corporate standards define the critical wavelength. This is an error, because prints need to clarify the dividing line (the critical wavelength) between form error and surface texture. However, real prints often have errors, so it is helpful to have several rules of thumb to aid inspection engineers who need to select a useful critical wavelength when none is specified.

- For round features like circles and cylinders, 50 UPR is a common choice.
- When you use a contact sensor, it often makes sense to choose a critical wavelength equal to the probe diameter.
- For closed features like ellipses and slots, it often makes sense to choose a critical wavelength equal to the circumference divided by 100.
- For open features like planes and freeform surfaces, it often makes sense to choose a critical wavelength equal to the length of the feature divided by 100.

Robust

Unlike other filter types, rather than removing outliers, ISO 16610-31 and ISO 16610-71 allow the filter to be applied in a way that makes it robust to the effects of outliers. For more information, refer to ISO-16610-31 ("Robust profile filters: Gaussian regression filters") and ISO-16610-71 ("Robust areal filters: Gaussian regression filters").

The Robust check box is a simple On/Off toggle that determines whether to make the filter robust to outliers or not. No points are removed.

Points to Note

Gaussian filtering does not work well unless every point has a reasonable number of neighbors: at least four. The neighbors of a point are defined to be all points within a radius equal to the cutoff wavelength. In some cases, scans do not have enough point density, either for individual areas or for the whole scan. This results in the filter performing poorly, such as not filtering out sensor noise, surface texture, outliers, etc. In extreme cases, if a point has no neighbors, the filter does nothing for that point.

You can use the following formulas as guides.

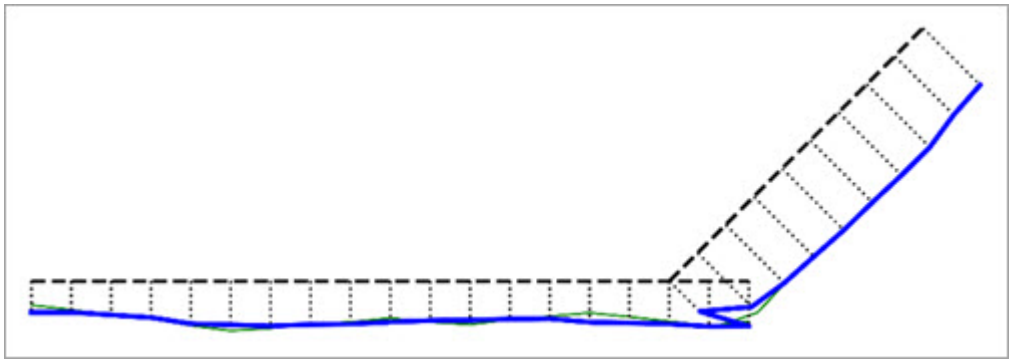
Minimum Number of Points = 5 / Cutoff Wavelength (mm)

*Minimum Number of Points = (5 * UPR) / (π * Diameter (mm))*

Gaussian filtering can also be sensitive to location errors as described in the examples below. These examples show that when PC-DMIS detects a surface that exhibits a kink, sharp corner, or tight radius, it provides a SENSITIVITY_FACTOR to indicate how much the filter has been affected.

Example 1 - Surface Exhibits a Kink

In this example, the two CAD surfaces overlap slightly, causing a kink in the scan path. This results in inconsistent vectors that cross over each other as the scan path goes back on itself.



----- Nominal Surface

_____ Measured Points

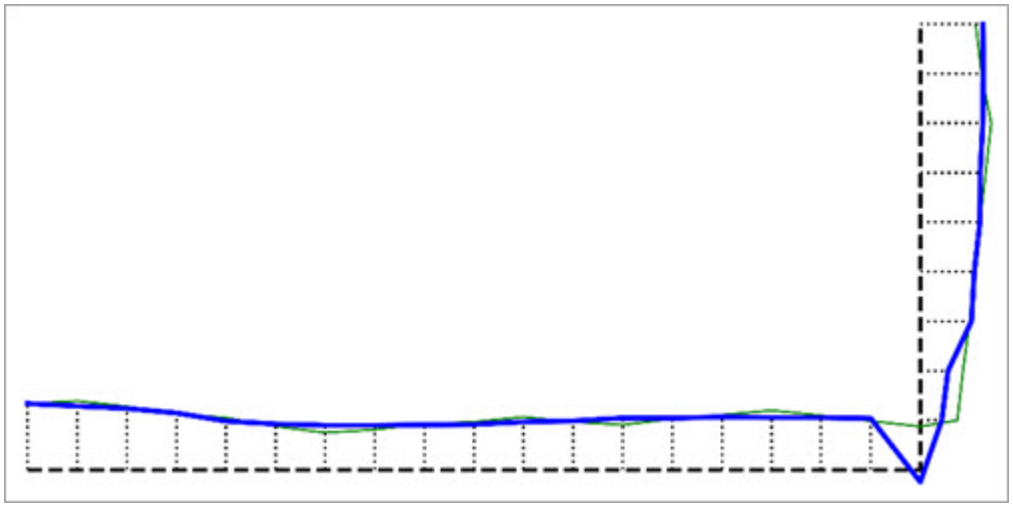
••••• Filtered Deviations

_____ Filtered Points

To fix this problem, adjust the scan path or point density to ensure the nominal vectors remain continuous near boundaries between CAD elements.

Example 2 - Surface Exhibits a Sharp Corner

In this example, the scan is off location while also going around a sharp corner. This results in significant distortion being introduced when attempting to filter.



----- Nominal Surface

_____ Measured Points

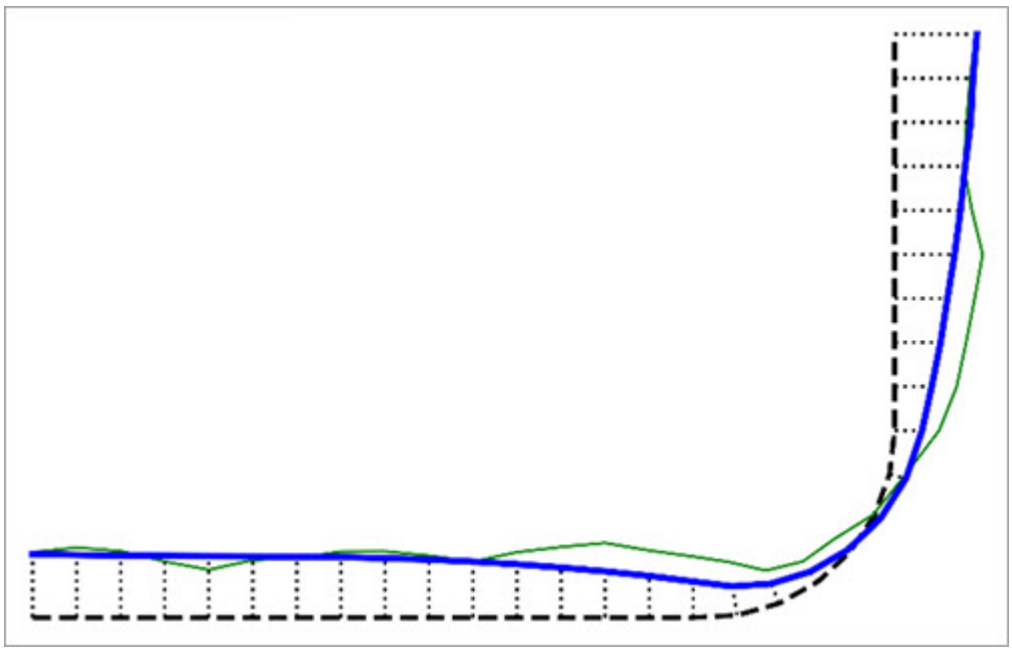
••••• Filtered Deviations

_____ Filtered Points

To fix this problem, avoid measuring near sharp corners. Maybe consider breaking the scan up into two separate scans, one for each edge.

Example 3 - Surface Exhibits a Tight Radius

In this example, the scan is off location while the critical wavelength is larger than the tight radius. This causes distortions near the radius.



----- Nominal Surface

_____ Measured Points

••••• Filtered Deviations

_____ Filtered Points

To fix this problem, ensure the critical wavelength is comparable to, or smaller than any tight radii.

The following is an example of a warning message that PC-DMIS might show when you create an ISO_16610_GAUSSIAN filter command and it detects any of these situations:

- The sensitivity factor exceeds 0.3.
- One or more points are found to have no neighbors.
- The number of points with less than four neighbors exceeds 2 + 0.01 * N, where N is the number of input points.

PC-DMIS

The filter output is sensitive to location errors by a factor of X.X which means the surface exhibits some kind of kink, sharp corner, or tight radius. To fix the problem, (1) ensure the nominal vectors remain continuous near boundaries between CAD elements, (2) avoid

measuring near sharp corners, and (3) ensure the critical wavelength is much less than any tight radii.

The following is an example of a warning message that PC-DMIS might show if it detects insufficient point density:

PC-DMIS

Warning: The point density is insufficient for the specified critical wavelength. There are 7 points with no neighbors, and 280 points with less than four neighbors.

These warnings are not triggered during execution. Instead, this command provides a number of feedback parameters to aid troubleshooting. With each execution, PC-DMIS stores the information in the Edit window command (visible in Command Mode), and is non-editable:

- **Sensitivity Factor** - This represents the proportion of the total error that this filter adds on top of ISO-16610-21 and ISO-16610-28 as a result of location errors in the input data.
- **Points With No Neighbors** - This is the number of points with no neighboring points.
- **Less Than 4 Neighbors** - This is the number of points with less than four neighboring points.

While PC-DMIS displays a warning during command creation, it does not prevent you from executing a constructed **ISO_16610_Gaussian** filter command based on this feedback. It does make the values available via GETTEXT expressions if you wish to add your own logic to test for undesirable results. The example below shows you how to do this.

```
NEW_UPR50_ROB_ON=FEAT/SET,CARTESIAN
    THEO/<-0.027007,-0.000578,-2>,<0,0,1>
    ACTL/<-0.027374,-0.00068,-1.999392>,<0,0,1>
    CONSTR/SET,FILTER,ISO_16610 GAUSSIAN,POLAR,ADP_CIRCLE,
    FILTER/ON,UPR=400,ROBUST/ON

SENSITIVITY_FACTOR=0.000005,POINTS_WITH_NO_NEIGHBORS=0,LESS_
THAN_4_NEIGHBORS=507
    ASSIGN/V1=GETCOMMAND("Constructed Filter Set", "UP",1)
    ASSIGN/FILTER_VALUE=GETTEXT("DVALUE_1",1,V1)
    ASSIGN/NO_NEIGHBORS=GETTEXT("DVALUE_1",2,V1)
    ASSIGN/LESS_NEIGHBORS=GETTEXT("DVALUE_1",3,V1)

    COMMENT/REPT,
    "FILTER_VALUE = " +FILTER_VALUE
    "NO NEIGHBORS = " +NO_NEIGHBORS
    "LESS_NEIGHBORS = " +LESS_NEIGHBORS
```

In rare cases, you might encounter an error during execution of your measurement routine similar to this one:

Construction Error

Math failed for SCN86: There are more than 100 million connections between points and their neighbors. This can be interpreted as the product of the number of points and the average number of connections between points. Please reduce the number of points or reduce the wavelength.

This can happen when you have both a very high point density and a large wavelength which would lead to unacceptable performance delays (extremely long calculation times of several minutes). For these cases, PC-DMIS is unable to execute the command, and the SENSITIVITY_FACTOR displays as an “Error” in the Edit window as shown in this code snippet:

```
SCN86=  =FEAT/SET, CARTISIAN
         THEO/<0,0,0>,<0,0,-1>
         ACTL/<0,0,0>,<0,0,-1>
         CONSTR/SET,FILTER,ISO_16610 GAUSSIAN,LINEAR,CYL1,
         FILTER/ON,WAVELENGTH=16,ROBUST/ON
         SENSITIVITY_FACTOR=Error,POINTS_WITH_NO_NEIGHBORS=0
,LESS_THAN_4_NEIGHBORS=0
```

To continue, you either need to reduce the feature’s point density or reduce the filter command’s wavelength.

Spline

The **Spline** filter option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) allows you to smooth the data by fitting a smoothing spline to it. The Smoothing Parameter value controls the amount of smoothing. However, if you want to use the "optimal" value computed by *Generalized Cross-Validation* (GCV) technique, you should set this value to 0. The Spline filter option is a 3D filtering option. This means deviations in all directions perpendicular to the line are affected.

The spline filter fits a spline approximation to the data and resamples it. This way, it smooths the data in all directions, not just radially or perpendicularly to the workplane. The fitted spline is a *natural smoothing spline*. It has a parameter that mediates between interpolating the data - trying to pass through every data point, which would retain all of the oscillations - and approximating the data with a successively smoother spline, in the sense that the oscillations are damped. The two limits of the smoothing parameter yield an interpolator (with all of the original oscillations) and a straight line. A zero value for the spline filter results in a spline that (asymptotically) minimizes the expected mean square error between it and the underlying, unknown, curve. You should usually use the zero value since this removes the noise yet leaves the underlying shape reasonably intact.

Math Information: Since the range of interest for the actual (internal) smoothing parameter lambda usually involves small values, the **Construct Filter Feature** dialog box accepts

```
-log10(lambda).
```

So, instead of `1-e-6`, you just type 6. You get more smoothing for smaller values of the smoothing parameter. For example, a value of 5 for the spline filter would give you more smoothing than a value of 6.

Uniform

The **Uniform** filter option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) filters the data by averaging all the points in a moving window. You can specify the width of the window through either the **Smoothing parameter** or the **Filter width** value.

If the **Filter width** value is `m`, then the window is `2m * delta` wide, where `delta` is the point spacing.

Triangle

The **Triangle** filter option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) filters the data by a weighted moving average of all the points in a moving window. The weights are determined by a triangle function with the peak in the middle of the window. You can specify the width of the window through either the **Smoothing parameter** or the **Filter width** value.

If the **Filter width** is *m* then the window is `2m·* delta` wide, where `delta` is the point spacing.

Cylindrical

The **Cylindrical** filter option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) filters data in a spiral scan or a circular scan that spans more or less than a full circle. The deviations in this case are radial. As PC-DMIS does not consider the data to be periodic (meaning it does not form a complete, closed circle), a number of points equal to the filter width at the beginning and at the end of the scan are absent from the filter set.

GAUSSIAN

The GAUSSIAN filter option is the most widely used filter option.

Starting with PC-DMIS 2025.1, the **ISO_16610_GAUSSIAN** filter option is available. This filter option is intended to supercede the **GAUSSIAN** filter option. The older version of the **GAUSSIAN** filter option was written in accordance with ISO 11562:1996 which was later cancelled and replaced by ISO 16610-21:2011. PC-DMIS retains this older **GAUSSIAN** filter option for migration purposes. Starting with PC-DMIS version 2025.1, we recommend that you select the **ISO_16610_GAUSSIAN** filter option when you create new measurement routines. You can not back-save measurement routines that use the newer **ISO_16610_GAUSSIAN** filter option to versions of PC-DMIS earlier than 2025.1.

You can access the newer **ISO_16610_GAUSSIAN** filter option from the **Constructed Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**). This newer version of the filter smooths your data by applying a Gaussian lowpass filter to it according to the ISO 16610 collection of standards.

The **GAUSSIAN** filter option in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) enables you to smooth the data by applying a Linear or Polar Gaussian lowpass filter to it according to the ISO 11562 standard. The **Cutoff wavelength**, **Cutoff frequency**, or **Filter width** value controls the amount of smoothing.

For the polar Gaussian filter, the data should be a *complete* circle scan, with radial deviations. A partial circle scan will not work correctly with this filter. The appropriate filter for a partial circle scan is the cylindrical filter discussed below.

For the linear Gaussian filter, the data should nominally lie in a plane with the deviations perpendicular to the plane. For this filter, the cutoff wavelength is in units of length. The following paragraphs describe how a linear Gauss filter functions:

- It estimates the point spacing delta as the average distance between the X and Y coordinates of the 3D points. These should be evenly spaced and coplanar. It will smooth the Z coordinates.
- If the smoothing parameter is

`m = filter width`
(i.e., the smoothed value is at the center of `2m+1` points used in a weighted average, starting at point `m`), then the cutoff wavelength `lambda` is computed as:

`lambda = m * delta / const`
(where `const` is a numerical constant.)

- If the input parameter is `lambda = cutoff wavelength`, then the filter width `m` is computed as:

  `m = lambda * const / delta` (actually, the next higher integer value).

So, the cutoff wavelength has the same units as the point spacing, but the filter width is a pure number.

Filter Width box

The **Filter width** value in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) is an optional smoothing parameter for all of the filter types except the Spline filter. The value in this box determines the number of data points used for filtering to the right and to the left of each smoothed data point. For example, if the Filter Width value is `m` then the window is `2m * delta` wide, where `delta` is the point spacing. The default value for this parameter is 3.

The value you type can be zero or any positive value.

- If there's no value at all (or if the value is zero), either the **Cutoff frequency** or the **Cutoff wavelength** value will be used to set the amount of smoothing.
- If either the **Cutoff frequency** or the **Cutoff wavelength** value is any non-zero, positive value, then the filter width corresponding to the cutoff frequency entered is displayed.
- If **Filter width** and the **Cutoff frequency** or **Cutoff wavelength** don't have values (or are zero), PC-DMIS will not filter the data.

The **Filter width** value also determines how many points will be absent from the filter set when doing *linear* filtering. PC-DMIS removes those points that don't have enough data to the right or left to fill up the data window.

A lowpass filter, one that filters out high frequency noise but transmits the lower frequency form and waviness, works by replacing the value at a data point by a weighted average of the values at a sequence of its neighbors.

For the *Gauss* filter, for instance, the **Filter width** specifies how many points to the left and right of a point are included in the weighted average.

The weights (which are positive and add up to 1) are values of a Gaussian distribution function (also known as a Bell Curve).

If the **Filter width** is equal to `m`, then the smoothed point is at the center of `2m+1` points used in the weighted average.

If the data is periodic, then the points wrap around and there are always enough points to the left and right of a point to compute this average. This is the case for a *Polar* filter. But for a *Linear* filter, the first and last `m` points do not have enough neighbors to compute the full weighted average, so these points are left out of the filtered dataset.

Cutoff Wavelength box

The **Cutoff wavelength** value in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) determines the wavelength of the oscillations in the data below which the amplitudes of the oscillations are reduced when you apply a *linear* Gaussian filter.

Cutoff Frequency box

The **Cutoff frequency** value in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) determines the number of "undulations per revolution" (or UPR) in the circle data above which the amplitudes of the oscillations in the data are reduced when you apply a *polar* Gaussian filter or a cylindrical filter.

Smoothing Parameter box

The **Smoothing parameter** value in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) determines the degree of smoothing applied to the data for the spline, uniform, and triangle filters.

- *For the spline filter*, it is best to set this value to 0 to indicate that the value computed via GCV should be used. The computed value then replaces the 0 in the Edit window.
- *For uniform and triangle filters*, the smoothing parameter represents one-half of the width of the window used in the (weighted) moving average.

Remove Outliers check box

If you select the **Remove outliers** check box in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**), PC-DMIS attempts to remove all points a given number of standard deviations from the substitute feature (usually a circle or a line). You specify the number of standard deviations in the **Standard deviation multiple** box. See the "Standard Deviation Multiple box".

Outlier removal behaves similar to filtering:

- If you select the **Linear** option, outlier removal is based on the 3D distance from the point in question to the line (best fit line through the data).
- If you select the **Polar** option, outlier removal occurs in the radial direction (parallel to the workplane).

Standard Deviation Multiple box

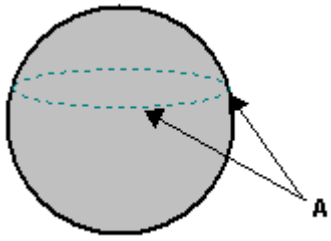
The **Standard deviation multiple** value in the **Construct Filter Feature** dialog box (**Insert | Feature | Constructed | Filter**) determines the number of standard deviations from the substitute (least squares) feature beyond which points are identified as outliers. The default value is 3.

Constructing an Adjusted Filter

The **Adjust Filter** construction type lets you adjust scan data that was gathered while scanning around these standard geometric features:

- Sphere
- Cone
- Cylinder

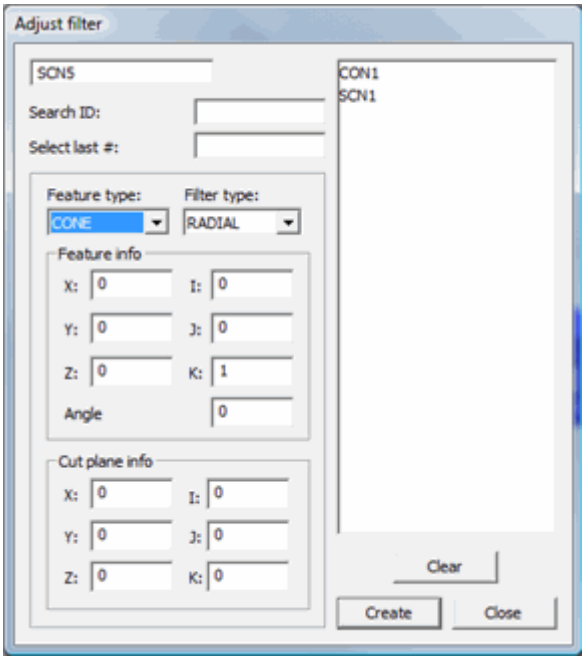
Generally, you perform these scans using an analog probe, such as an SP600. If you imagine a scan around the top quarter of a sphere, for example, you would ideally get points that lie within a plane that cuts through the sphere, like this:



A - Scan Path

However, in reality, as the probe scans around the sphere, it is physically impossible for all the points to remain inside the cut plane due to a natural rise and fall floating motion of the CMM.

The **Adjust Filter** construction can take the pre-existing scanned points and, using the known mathematical properties of the feature, better compensate the points gathered during the measurement process, adjusting them to lie more within the cut plane. You use the **Adjust filter** dialog box to do this. To access this dialog box, select **Insert | Feature | Constructed | Adjust Filter**.



Adjust filter Dialog Box

The dialog has these options:

Feature Type
This defines the primitive (simple geometric feature) being scanned. You can only select **Sphere**, **Cone**, or **Cylinder**.

Filter Type
This defines the type of filter to use, either **Axial** or **Radial** (applies to Cone and Cylinder feature only - see step 4 below).

Feature Info
This lets you define the information about the feature.

XYZ - The feature's nominal position.
IJK - The feature's normal vector.
Radius/Angle - This sets the Radius of the Sphere or Cylinder or the angle of the Cone.

Cut Plane Info
This area defines the location and vector of the cut plane.

XYZ - Location of the Cut Plane
IJK - Vector of the Cut Plane

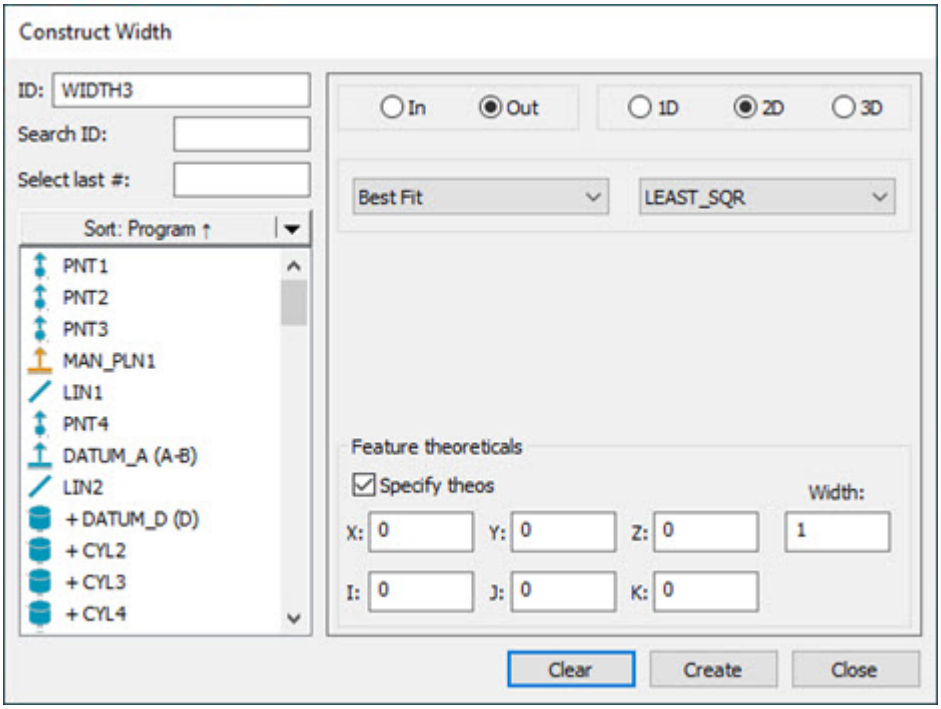
To Adjust Scan Data

To correctly adjust the data in the scan, you need to give enough information to mathematically define the feature.

1. Select the actual sphere, cone, or cylinder feature from the list of features in the **Adjust filter** dialog box.
2. Select a pre-existing scan feature from the list. Generally, your scanned feature is a pre-existing linear closed scan. The scan must be of a sphere, cone, or cylinder.
3. Choose the feature type you scanned with the selected scan from the **Feature Type** list.
4. From the **Filter Type** list, choose the correct filter type for a Cone or Cylinder feature. The filter indicates how to compensate the data.
    - If you are scanning perpendicular to the axis of the Cylinder or Cone, choose the **Radial** filter. This defines the circle that is used to adjust the points.
    - If you are scanning parallel to the feature's axis, choose the **Axial** filter. This defines a line that is used to adjust the points.
5. Define the feature's nominal XYZ location by typing the values in the **XYZ** boxes.
6. Define the feature's normal vector by typing the values in the **IJK** boxes.
7. Define the feature's size by typing a value in the **Radius** or **Angle** box.
8. In the **Cut Plane Info** area, define the scan's cut plane. Once you specify all the nominal information for the scanned feature, you can create the construction.
9. Click **Create**. All the points are projected into the cut plane along the features theoretical definition (taking into account the known geometry) rather than a straight projection along the features normal vector.

Constructing a Width Feature

You can construct a Width feature from two opposing sides of a notch, tab, or other similar surfaces. The Width feature is useful to the GD&T standard which requires the width as a datum or considered feature in Geometric Tolerance dimensions. For more information on Geometric Tolerance dimensions, see "Using Geometric Tolerances".



Construct Width Dialog Box

Width constructions require two input features, one feature for each opposing side of the notch or tab.

PC-DMIS supports the following Width constructions:

- **1D** - This construction is between two opposing points.
- **2D** - This construction is between two opposing lines.
- **3D** - This construction is between two opposing planes.

PC-DMIS considers the Width feature as an outer (OUT) feature when the opposing planes, lines, or points come from a tab-like geometric feature. PC-DMIS considers the feature as an inner (IN) feature when the opposing planes, lines, or points are measured from a notch-like geometric feature.

The vector of the Width feature points from the first selected feature perpendicular to the second feature.

To construct a Width Feature

1. Measure the required two features (two points, two lines, or two planes) on the opposing sides of a notch, tab, or other two opposing surfaces. These are the two features to use when you construct the width.
2. Open the **Construct Width** dialog box in one of these two ways:
   - Select **Insert | Feature | Constructed | Width**.
   - From the **Construction** toolbar, click the **Width** button.
3. Select **1D**, **2D**, or **3D**.
   - If your input features are two points, select **1D**.
   - If your input features are two lines, select **2D**.
   - If your input features are two planes, select **3D**.
4. From the **Feature** list, select the two input features for the construction.
5. Select **In** or **Out**:
   - If the Width feature contains material between the two opposing sides, like a tab, it's an outer feature. Select **Out**.
   - If the Width feature does not contain material between the two opposing sides (like a notch), it's an inner feature. Select **In**.
6. From the **Fitting** list, for **2D** or **3D**, select the type of fitting algorithm to use: **Best Fit** or **Best Fit Recomp**.
7. From the **Math Algorithm** list, for **2D** or **3D**, select the Best Fit math algorithm to use. The available options are:
   - Maximum Inscribed (**MAX_INSC**)
   - Minimum Circumscribed (**MIN_CIRCSC**)
   - Least Square (**LEAST_SQR**)

   For more information, see "Best Fit Type" under "Constructing a Best Fit or Best Fit Recompensate Circle".
8. If you want to define the theoretical width information, mark **Specify theos** and then type the theoretical centroid in the **X**,**Y**, and **Z** boxes. Next, type the theoretical vector in the **I**, **J**, and **K** boxes. Finally, type the width between the two sides in the **Width** box.
9. Click **Create** to insert the Width feature into the measurement routine.

The Edit window command line for the width construction shows:

```
feature_name=FEAT/WIDTH,TOG1,TOG2,TOG3,TOG4
THEO/<x_cord,y_cord,z_cord,i_vec,j_vec,k_vec>,length
ACTL/<x_cord,y_cord,z_cord,i_vec,j_vec,k_vec>,length
CONSTR/TOG5,TOG6,feat1,feat2
```

TOG1 = CARTESIAN or POLAR.

TOG2 = IN or OUT. This is for display only; you cannot use the Edit window to edit this.

TOG3 = MAX_INSC / LEAST_SQR / MIN_CIRCSC. These only apply to **2D** or **3D**.

TOG4 = YES or NO. These are user-defined theoretical values.

TOG5 = The constructed feature type.

TOG6 = BEST FIT / BEST FIT RECOMP. These only apply to **2D** or **3D**.

Notes

You can use a **3D** width as a primary, secondary, or tertiary datum, or as any referenced feature in an alignment. You can use a **2D** width as a secondary or tertiary datum, or as a “rotate to” feature or final “translate to” feature in an alignment. You can use a **1D** width as a tertiary datum or as a final “translate to” feature in an alignment.

If you create a Location dimension of the constructed Width feature, the X, Y, and Z axes represent the midpoint of the Width feature. The L axis represents the length of the Width feature.

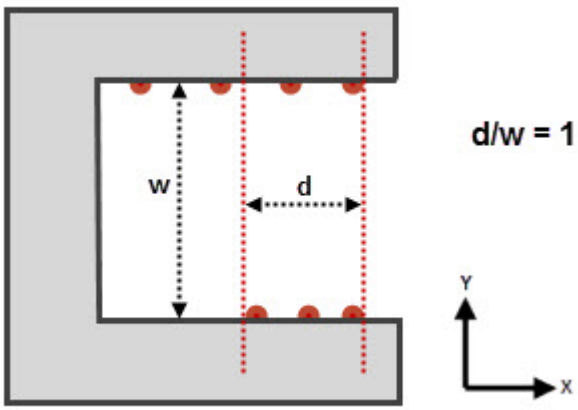
To calculate a Width feature in **2D** or **3D**, the hit locations on the two opposing features must have enough overlap. To calculate a Width feature in **1D**, the hit locations on the two opposing features must be close to opposed.

When PC-DMIS calculates a Minimum Circumscribed or Maximum Inscribed Width feature in **2D** or **3D**, if **d** is the overlap distance and **w** is the width distance (as shown in the figure below), the recommended overlap ratio should be:

d/w = 1 or more

If a large overlap ratio is not feasible due to the physical structure of the part, use the Least Square method for the Width calculation instead.

For example, in this top-down view of a part, **d** represents the amount of distance between the two sets of hits where overlap occurs on the two sides, and **w** represents the Width feature's distance between the two sides. Suppose d = 1.5 and w = 3. The overlap ratio of d/w is .5. The recommended ratio is 1. So, there may not be enough overlap for the Minimum Circumscribed or Maximum Inscribed fitting algorithms. In such a case, use the Least Squares algorithm instead:


w
d
d/w = 1
Y
X

Creating Generic Features

Creating Generic Features: Introduction

This chapter covers the ability to add generic features into your measurement routine and create point features at the probe's current position.

Generic features are often used to control the measured and theoretical values of a specific feature in order to perform calculations that PC-DMIS's constructed features don't currently support. For example, suppose you want to create the shortest line between two lines (3D) that don't intersect. PC-DMIS doesn't have that construction option. However, you can do the math yourself using the PC-DMIS Expression language and then assign the theoretical and measured values of a generic line to the appropriate values. In this way, you can create your own construction.

This section contains these major topics:

- Description of Generic Features
- Purpose of Generic Features

Description of Generic Features

Usually, when PC-DMIS measures features on your part, it guesses the correct feature type depending on the number of hits taken and the inherent feature type available on the imported CAD model itself.

PC-DMIS does not guess generic features. Instead, the user inserts and creates them.

Purpose of Generic Features

Generic features are used to hold and transform values. You can use these values to create new constructed features or modify existing features via the CALCULATION command.

For information on how to create new constructed features, see the "Constructing New Features from Existing Features" chapter.

Generic Feature Command Format

All generic features have XYZ and IJK values. They may have ANGLE, DIAMETER, RADIUS, or DISTANCE values depending on the type assigned to the feature.

The **Nominal / Measured Values** toggle button indicates the values that are used in the generic feature.

The command line in the Edit window (a cone is shown) is as follows:

```
feature_name        =GENERIC/TOG1,TOG2,  TOG3, TOG4
                     NOM/XYZ,x_cord,y_cord,z_cord
                     MEAS/XYZ,x_cord,y_cord,z_cord
                     NOM/IJK,i_cord,j_cord,k_cord
                     MEAS/IJK,i_cord,j_cord,k_cord
                     [RADIUS/radius] | [DIAMETER/diameter]
                     [ANGLE/ang]
                     [DISTANCE/dist]
```

TOG1 =POINT, PLANE, LINE, CIRCLE, SPHERE, CYLINDER, CONE, SLOT, or NONE

TOG2 = This indicates whether the generic feature is alignment dependent or independent.

TOG3 = This indicates whether the feature uses Polar or Rectangular coordinates.

TOG4 = This indicates whether the feature is an IN (hole) or OUT (stud) feature.

[ ] = The options shown in brackets are discretionary and are available based on the type assigned to the feature by TOG1. RADIUS and DIAMETER are mutually exclusive representations for the value represented as the radius or the diameter.

Creating a Generic Feature

While it is entirely possible to type the word `GENERIC` and press Tab in the Edit window in Command Mode to create a generic feature, PC-DMIS provides an easier way to create new generic features. The **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) provides a more intuitive way to create new generic features.

To Add a Generic Feature

To add a generic feature:

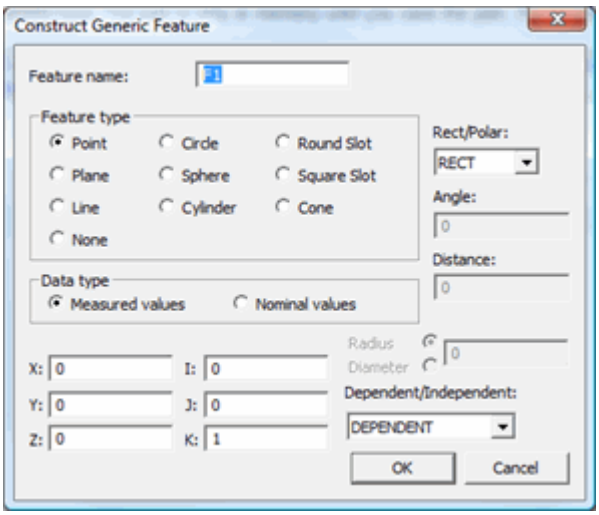
1. Place your cursor in a new line of the Edit window.
2. Access the **Construct Generic Feature** dialog box by one of the following:
    - Select **Insert | Feature | Generic**.
    - Type GENERIC in the Edit window in Command Mode.

- Select Generic Feature from the list that appears in Summary Mode after you select **Add Command**.

3. Make the changes you desire.

4. Click the **OK** button to create the feature. PC-DMIS saves the generic feature and update the information for the feature in the Edit window and the Graphic Display window.

Description of the Construct Generic Feature Dialog Box



Construct Generic Feature Dialog Box

Use this dialog box to create new features. To open this dialog box, select **Insert | Feature | Generic**.

The topics below describe the options that are available on this dialog box.

Limited CAD Selection

With the **Construct Generic Feature** dialog box open, you do have some very limited wireframe CAD selection capabilities. This is limited to selecting points and lines. For more advanced CAD selection requirements, you would need to use PC-DMIS's Auto Feature or Constructed Feature capability.

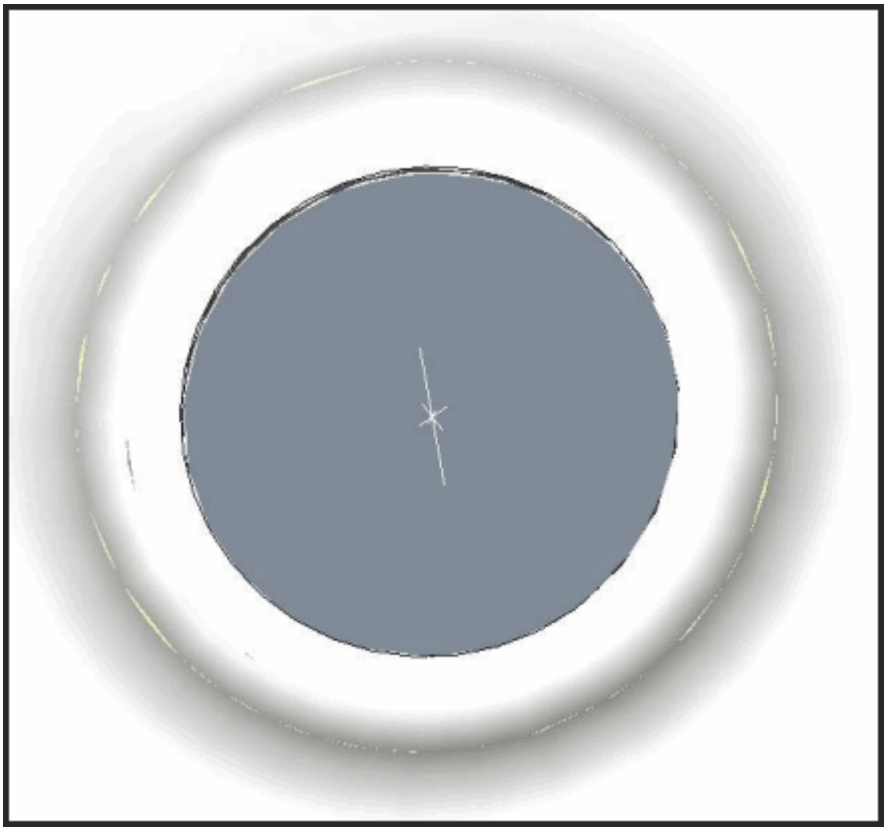
- If you select a point, PC-DMIS fills in both the nominal and measured values into the dialog box for that feature.

- If you select a line, then the IJK information is inserted for both nominal values and measured values.

PC-DMIS ignores any other type of wireframe entity that you select.

For example, in this top-down image of a circle feature seen below, you will notice a white cross and line in the center of the circle. The cross and line are CAD elements of a point and line, respectively, that are created within the CAD system and imported as part of the

CAD file into PC-DMIS. You would select either the line or the cross (not the actual circle feature) to have its data input into the dialog box.



Example showing a wireframe point and line element at the center of a Circle feature.

Feature Type

The **Feature type** area in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to create these generic feature types:

- Point
- Plane
- Line
- Circle
- Sphere
- Cylinder
- Round Slot
- Square Slot
- Cone
- None

PC-DMIS disables or enables other areas and options in this dialog box based on the type of feature that you select.

Data Type

The **Data type** area in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to determine the values of the generic feature that are affected by the changes you make in the dialog box. The choices include **Measured values** and **Nominal values**.

XYZ Boxes

The **X**, **Y**, and **Z** boxes in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enable you to determine the X, Y, and Z positions for the generic feature.

IJK Boxes

The **I**, **J**, and **K** boxes in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enable you to determine the vector for the generic feature.

Alignment Dependent Generic Feature

Alignment dependent generic features (**Insert | Feature | Generic**) have values that depend on the alignment from which they are referenced. They change so that they are always relative to the current alignment. Their location is constant in 3D space.

For an example in 2D, suppose 0,0 is your machine origin. An alignment is set up that locates the origin at 5,5. Now, an alignment dependent feature is defined after the alignment with the values of 2 for x and 2 for y. Relative to the alignment, its values are 2 and 2. Relative to the origin, its values are 7 and 7. No matter what alignment the numbers are given relative to, the point will always be located at 7,7 relative to the true origin.

Alignment Independent Generic Feature

Independent generic features (**Insert | Feature | Generic**) have values that remain the same, regardless of which alignment they are referenced from (therefore considered "independent"). Their location in 3D space changes as the alignment changes.

For an example in 2D, consider the same example of 2,2 (as shown in the Alignment Dependent example above) but this time with an alignment independent generic feature. If you ask for the x and y values relative to the alignment, the object will report 2,2. If you ask for its values relative to the origin, the object will still report 2,2. The set 2,2 relative to the alignment is located at 7,7 relative to the true origin. The set 2,2 relative to the origin is exactly that: 2,2. Thus, the point seems to move in 2D space.

The independent option was added so that a generic feature named XAXIS, for example, could always be used as an input feature to a construction or a datum to a dimension, and it would always have the values of 1,0,0 independent of the current alignment.

Polar/Rect

The **Polar/Rect** list in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to pick the type of coordinate system that is used in the generic feature. You can select either POLAR or RECT.

Feature Name

The **Feature name** box in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to name your feature. This is the ID that appears on the Feature ID label and inside the Edit window.

Radius and Diameter

The **Radius** and **Diameter** options in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enable you to define either a radius or a diameter for a circular feature. Select either **Radius** or **Diameter** and then type a value in the box.

Angle

The **Angle** box in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to define the angle of a Cone. This box also becomes available if you selected **None** in the **Feature type** area.

Distance

The **Distance** box in the **Construct Generic Feature** dialog box (**Insert | Feature | Generic**) enables you to define the height or length of any generic features that have a length or height.

Creating a ReadPoint from the Probe's Position

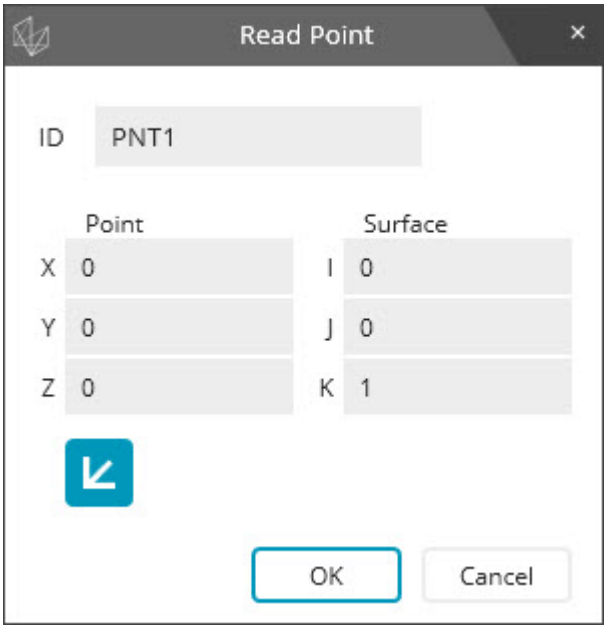
The **Insert | Feature | Read Point** inserts a point in the Edit window at the read in position. This tells PC-DMIS to read the probe's current position.

The following syntax illustrates what's inserted into the Edit window.

```
ID =FEAT/ POINT, TOG1
THEO / x, y, z, i, j, k
ACTL / x, y, z, i, j, k
READPOINT/
```

READPOINT - Identifies this feature as a read point created from the probe's position.

If you press F9 on this command, PC-DMIS displays the **Read Point** dialog box.



Read Point Dialog Box

ID - This box displays the Feature ID.

Point (X, Y, Z) - This box displays the X, Y, Z location for the constructed point.

Surface (I, J, K) - This box displays the I, J, K approach vector of the probe.

- The **Cartesian/Polar** icon switches the display of the X, Y, and Z values between the Cartesian and Polar coordinate systems. The **XYZ** labels display **RAH** when you switch to Polar.

For example:

```
PNT1 =FEAT/POINT,CARTESIAN
THEO/<7.4982,2.0111,0.95>,<0,0,1>
ACTL/<7.4982,2.0111,0.95>,<0,0,1>
READPOINT/
```

If you execute this type of point feature in Offline mode, and the feature follows a `MODE/MANUAL` command, the software copies the theoretical vector and position to the measured vector and position; otherwise, the value comes from the current probe position.

Creating and Using Alignments

Creating and Using Alignments: Introduction

Once you have activated a tip and measured features, you can create a coordinate system (or alignment). PC-DMIS provides you with a variety of tools to create and manage coordinate systems. To access the tools that allow you to work with alignments, select the desired menu options available from the **Insert | Alignment** submenu.

The major topics in this chapter describe how to use these tools to effectively create and manage alignments in your measurement routine. These topics include:

- Alignment Overview
- Alignment Command Format
- Description of the Alignment Utilities Dialog Box
- Creating a 3-2-1 Alignment
- About QuickAlign
- Creating an Iterative Alignment
- Creating a Best Fit Alignment
- Saving an Alignment
- Recalling an Existing Alignment
- Using an Alignment Inside Loops or Branches
- Equating an Alignment
- Equating CAD to Measured Part Data
- Performing a Leapfrog Operation
- Changing Alignment Nominal Values
- Updating Dependent Commands When the Alignment Changes

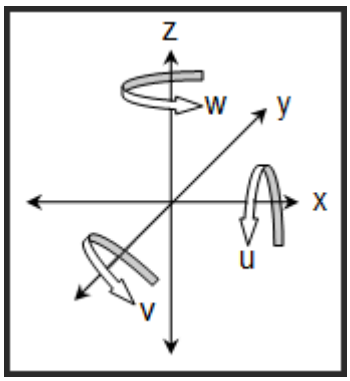
Bundle Alignments

If you have a Leica laser Tracker configuration, you can also perform a Bundle alignment. This special type of alignment is discussed in the PC-DMIS Portable documentation.

Alignment Overview

An alignment allows you to define your part's location and orientation in 3D space. It allows your measuring machine to know where the part is located. A part without any alignment has six degrees of freedom:

- Three degrees of rotation (about the X, Y, and Z axes)
- Three degrees of translation (origin in X, Y, and Z axes)



Example showing the six degrees of freedom in 3D space (X, Y, Z, U, V, and W).

Datum Reference Frame

A Datum Reference Frame (DRF) constrains the six degrees of freedom, fixing the part in 3D space.

A part alignment represents the DRF specified on the drawing. The primary, secondary, and tertiary datums define the DRF and identify the features to measure and use to create the alignment.

- The three degrees of rotation are constrained by the I, J, K vector(s) of the datum feature(s).
- The three degrees of translation are constrained by the X,Y,Z location(s) of the datum feature(s).

LEVEL

Constrains two degrees of rotation such that the leveled axis matches the vector of the selected feature.

This will always be the primary datum and must be a 3D feature with a vector.

Typical Features: Plane, Cylinder, Cone, or a constructed 3D feature.

ROTATE

Constrains one degree of rotation about the leveled axis such that the rotated axis matches the vector of the selected feature.

This will always be the secondary or tertiary datum and must be a 2D or 3D feature with a vector.

Typical Features: Plane, Line, Cylinder, Cone, or a constructed 2D/3D feature.

You can also select any two-point type features to simulate a line that can be used to rotate. These can be two points, two circles, two spheres, or a combination thereof. The direction of the simulated line is based on the order of the selected features.

ORIGIN

Constrains three degrees of translation (origin) in X, Y, and Z-axis.

This sets the origin on primary, secondary, and tertiary datums or as per drawing requirements.

Typical Features: Any feature.

Alignment Tips:

- LEVEL first, ROTATE second, and then set the ORIGIN for the X, Y, and Z axes. **Never Rotate before Leveling!**
- Always LEVEL before measuring 2D features (lines and circles).
- Always LEVEL and ROTATE before measuring points (measured point in X, Y or Z-axis)
- There is no limit on the number of alignments saved in a measurement routine.
- You can use the SAVE ALIGNMENT command to save an alignment to a file. This is typically done to create a fully automated measurement routine dependent on a holding fixture for the part.

1. Create a measurement routine that establishes an alignment on a fixture, and then save the alignment to a file.
2. Create a measurement routine, RECALL the alignment file at the beginning of the measurement routine and set it to run in DCC Mode before measuring the first feature.
3. During measurement routine execution, the CMM pauses, prompts the operator to load part, then automatically measures the part (no manual alignment).

- **Right Hand Rule of Rotation** - Point your right hand's thumb in the positive direction of the axis you are rotating about (+X, +Y, or +Z). The direction your hand naturally

curls is the positive rotation about that axis. Negative rotation is the opposite direction.

Alignment Command Format

All alignments appear in the following format in the Edit window in Command Mode. There will be slight variations that are explained more thoroughly in the following sections.

A typical alignment is shown in this pseudo-code snippet:

```
A1=ALIGNMENT/START, RECALL:,LIST=YES/NO
   ALIGNMENT/LEVEL,'feature_name'
   ALIGNMENT/ROTATE,XPLUS,TO,feat_name,ABOUT,ZPLUS
   ALIGNMENT/TRANS,XAXIS,feature_name
   ALIGNMENT/TRANS,YAXIS,feature_name
   ALIGNMENT/TRANS,ZAXIS,feature_name
ALIGNMENT/END
```

For a list of field rules for alignment commands, see "Conventions"

Descriptions of the alignment commands are available at these locations:

- For the Start command, see "Start Alignment Command".
- For the End command, see "End Alignment Command".
- For the Recall option, see "Recalling an Existing Alignment".
- For the Level command, see "Level".
- For the Rotate command, see "Rotate".
- For the Translate command, see "Origin".

Start Alignment Command

The line that starts the alignment is:

```
ALIGN_ID=ALIGNMENT/START,RECALL:ID, LIST=YES/NO
```

Changeable fields:

Align_ID
This is the ID that the alignment will be stored as. The operator assigns the ID. PC-DMIS assigns a default name if you do not specify an ID. Example: A1.

ID
ID is a previous internal alignment that will be recalled to start a new alignment. Note that you can also use the USE_ACTIVE_ALIGNMENT keyword in this field to cause PC-DMIS to use the active alignment instead of recalling a stored alignment. This is useful when you use alignments with looping and conditional branching. See the "Using an Alignment Inside Loops or Branches" topic.

You can use the USE_PART_SETUP keyword in this field to cause PC-DMIS to use the transformation defined in the **Part setup** area of the **Part/Machine** tab in the **Setup Options** dialog box. This was handled by leaving the ID field blank in versions prior to version 2012. See the "Setup Options: Part/Machine tab" topic in the "Setting Your Preferences" chapter.

LIST
Setting this field to YES or NO lets you determine whether or not PC-DMIS should display the alignment ID in the **Alignments** list on the **Settings** toolbar for later insertion in the measurement routine. The default setting is YES. Setting this field to NO is useful if you have a lot of temporary alignments, and you don't want to list all of them in the **Alignments** list on the **Settings** toolbar.

End Alignment Command

The Edit window command line for this option reads:

```
ALIGNMENT/END
```

There are no changeable fields in this command. You must use this command whenever you use the `ALIGNMENT/START` command.

Adding Lines

To add a line, do the following:

1. Place the cursor in the desired location and press the Enter key.
2. Type the word ALIGNMENT.
3. Press the Tab key.

PC-DMIS adds the new line depending on where the cursor is located:

- If the cursor is in the middle of a command, PC-DMIS creates a new line below the current line.
- If the cursor is placed at the beginning of a command line, PC-DMIS places the new line above the current cursor position.

The first new line always displays the minor command: LEVEL. To easily change this, enter a new command. Additional lines that you create after the initial line display the most recent command.

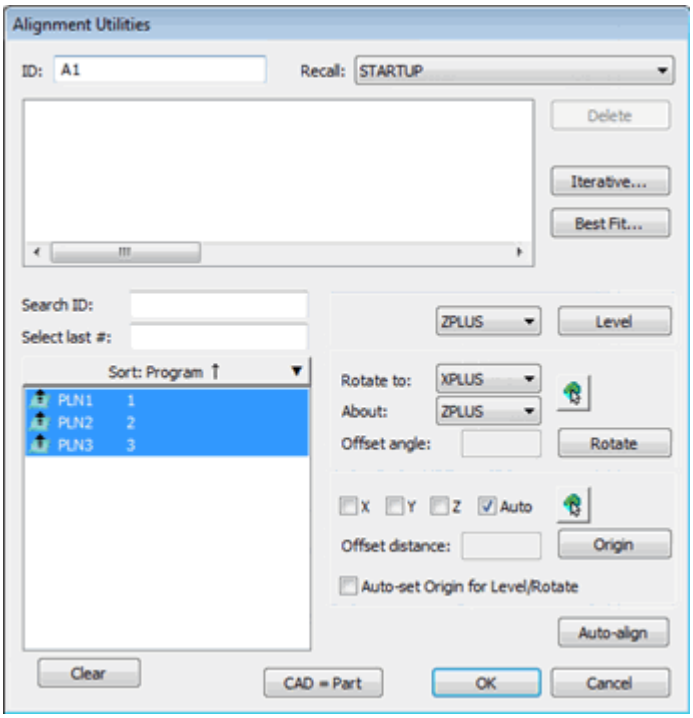
Deleting Lines

To delete a blank line, press the Down Arrow or Enter key. The line may also be highlighted and deleted. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

Conventions

- All alignment text commands are in a macro type format with a start command and an end command.
- The `ALIGNMENT/START` command is always the first line, and the `ALIGNMENT/END` command is the last line of the alignment statement.
- All alignment function sub-commands must be inside the start and end commands. The supported sub-command types are:
    - ALIGNMENT/LEVEL
    - ALIGNMENT/ROTATE
    - ALIGNMENT/ROTATE_CIRCLE
    - ALIGNMENT/ROTATE_OFFSET
    - ALIGNMENT/TRANS
    - ALIGNMENT/TRANS_OFFSET
    - ALIGNMENT/ITERATE
    - ALIGNMENT/BF3D
    - ALIGNMENT/BF2D
    - ALIGNMENT/BFUSER
- In each "ALIGNMENT/" command (except "START" and "END"), the second field allows you to change from one choice to another. All of the other operator-dependent fields then change to represent the field that is currently active.

Description of the Alignment Utilities Dialog Box

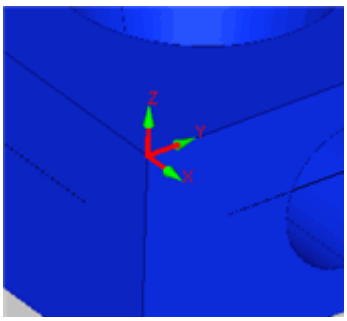


Alignment Utilities Dialog Box

When you select **Insert | Alignment | New** (or press F9 on an existing alignment command), PC-DMIS displays the **Alignment Utilities** dialog box. This dialog box provides a way for you to construct an alignment from the features that have been measured up to that point in the measurement routine.

An alignment is not complete until you select the **OK** button and PC-DMIS updates the **Active Alignment List** box.

As long as the **Alignment Utilities** dialog box remains open, PC-DMIS indicates any remaining unconstrained degrees of freedom by causing the XYZ origin symbol (or alignment trihedron) to continually rotate about and translate in the direction of those unconstrained axes in the Graphic Display window. Once the alignment is completely constrained, PC-DMIS then displays the trihedron in a fixed location and orientation that represent the alignment position:



Example showing an alignment trihedron.

ID box

This box defines the identification for the current alignment. If you are creating a new alignment, the alignment ID will default to a new name. To change the ID, type a new value in this box and press TAB.

Recall list

The **Recall** list contains all the internal alignments that have been defined in the measurement routine prior to this alignment. The alignment selected from this **Recall** list acts as the start condition of the current alignment.

If you are creating a new alignment, the **Recall** list displays the alignment active at the cursor's current location in the Edit window. If you have not defined any alignment, or if the cursor location is before any defined alignments in the Edit window, PC-DMIS selects the STARTUP command as the recalled alignment.

You can recall any available alignment listed in the **Recall** list. Only alignments created prior to the current cursor's position and certain predefined special cases are available for selection from that list.

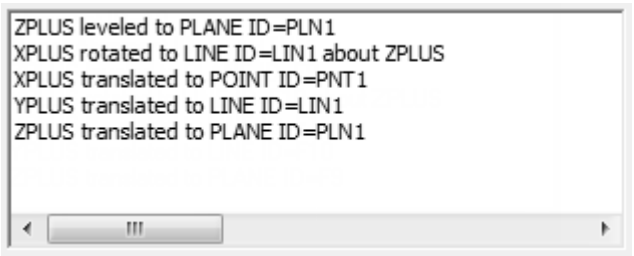
These special cases include:

STARTUP - You can recall the STARTUP alignment automatically defined at the start of the measurement routine.

USE_ACTIVE_ALIGNMENT - See the "ID" description in the "Start Alignment Command" topic for more information.

USE_PART_SETUP - See the "ID" description in the "Start Alignment Command" topic for more information.

Alignment Sub-Command list



This list displays a summary of each of the sub-commands that make up the current alignment block. This summary includes information such as the type of alignment sub-command, axis direction, and feature or features used by the sub-command to perform its rotation and/or translation.

Editing a Sub-Command

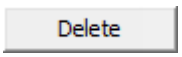
While you can edit any existing Level, Rotate, or Origin alignment sub-command inside the Summary Mode or Command Mode of the Edit window, you can also edit an existing sub-command by clicking on its entry in the sub-command list and making the needed change. Once you select an item, the appropriate area of the **Alignment Utilities** dialog box becomes available to edit. After you've edited the item, click the related sub-command button (**Level**, **Rotate**, or **Origin**) to modify the alignment.

For example, if you wanted to modify the feature to which the alignment was leveled, you would select the "leveled to" item in the list. PC-DMIS will display the current feature used for the leveling portion of the alignment, and then it would make the **Level** button available.

Simply select the new feature and axis, and click **Level**. PC-DMIS updates the **Alignment Sub-Command** list to reflect your changes.

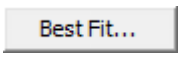
To deselect an item from the **Alignment Sub-Command** list, click on that item a second time, or press the ESC key on the keyboard.

Delete



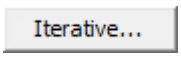
The **Delete** button deletes the currently-selected sub-command from the **Alignment Sub-Command** list.

Best Fit



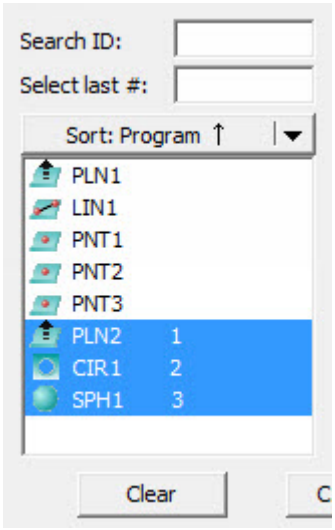
The **Best Fit** button opens the **Best Fit** dialog box. Use this dialog box to create or edit Best Fit alignments. See the "Creating a Best Fit Alignment" topic.

Iterative



The **Iterative** button opens the **Iterative Alignment** dialog box. Use this dialog box to create and edit Iterative alignments. See the "Creating an Iterative Alignment" topic.

Available features list



The **Available features** area displays all the available features in your measurement routine that exist above the current cursor location. These are the features available to your current alignment command.

You can also use the Graphical Hit Selection method to select inputs to your alignment. For details, see "Graphical Hit Selection Method".

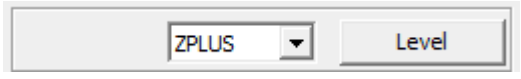
For a full description, see "Feature List box" under "Dialog Box Description" in the "Navigating the User Interface" chapter.

The **Search ID** filters the list of features by the specified ID. Type the ID string, and press the Tab key.

The **Select last #** box selects the final 'N' features in the list, where 'N' is the number you type. Type the number of features to select, and press the Tab key.

The **Clear** button clears the current feature selections from the list.

Level



The **Level** area establishes the orientation of the normal axis of the current workplane.

To choose a feature to "level" to:

1. Select the feature to be used from the **Feature List** box.
2. Choose the axis to level to from the drop-down list.
3. Click the **Level** button.

The **Level** button only becomes active if you first select a feature from the feature list or an existing level sub-command (`ALIGNMENT/LEVEL`). If you select an existing level sub-command, it will be modified to use the new feature selection; otherwise, a new level sub-command will be created using the selected feature.

You may also specify the axis to be used in establishing the orientation by selecting it from the drop-down list. The available options are:

ZPLUS

XPLUS

YPLUS

ZMINUS

XMINUS

YMINUS

After you click the **Level** button, the **Alignment Utilities** dialog box changes the **About** list next to the **Rotate** button to automatically match the **Level** button's axis direction.

The Edit window command line for this option reads:

```
ALIGNMENT/LEVEL,ZPLUS,'feat_id'
```

Changeable fields:

"ZPLUS"

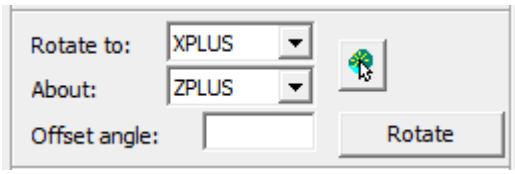
This is a field that allows you to switch between the ZPLUS, XPLUS, YPLUS, ZMINUS, XMINUS, and YMINUS fields in the Edit window. This field represents the direction of the specified axis that the feature is leveled to.

"feat_id"

The alignment levels to the specified feature.

Example: PLANE1.

Rotate



The **Rotate** area rotates the alignment to be parallel to a selected feature by a specific manual offset angle or by an angle defined from a picked CAD surface or edge.

PC-DMIS rotates the **Rotate to** axis around the alignment origin, about the specified alignment axis (**About** axis). The **Rotate** to and **About** axes cannot be the same axis.

The available options are:

ZPLUS

XPLUS

YPLUS

ZMINUS

XMINUS

YMINUS

Rotate to a Feature

To rotate to a measured feature:

1. Select the proper reference feature from the **Feature List** box in the **Alignment Utilities** dialog box.
2. Specify the axis to rotate to in the **Rotate to** list.
3. Specify the axis to rotate about in the **About** list.
4. Click the **Rotate** button.

The **Rotate** button only becomes active if you first select a feature from the feature list or select an existing rotate sub-command (`ALIGNMENT/ROTATE`). If you select an existing rotate sub-command, it is modified to use the new feature selection **Rotate to** axis and **Rotate About** axis; otherwise, PC-DMIS creates a new rotate sub-command using the selected feature and axis settings.

The Edit window command line for this option reads:
`ALIGNMENT/ROTATE,XPLUS,to,'feat_id',about,ZPLUS`

Changeable fields:

"XPLUS","ZPLUS"

This is a toggle field that allows you to toggle between these fields in the Edit window:

ZPLUS

XPLUS

YPLUS

ZMINUS

XMINUS

YMINUS

PC-DMIS sets this axis parallel to the specified input feature. It then rotates about the second axis.

"feat_id"

The alignment rotates parallel to the specified feature. Example: LINE1.

Rotate to a Line between Two Circles

To rotate to the line between two circles, select two circles in the **Available Features** list instead of a single feature and then continue with the "Rotate to a Feature" procedure described above.

The Edit window command line for this option reads:

```
ALIGNMENT/ROTATE_CIRCLE,ID, ID
```

Changeable fields: "ID"

This is one of the two ids to which to rotate.

Rotate by a Manual Offset Angle

The **Offset angle** box allows you to rotate the alignment by a specified angle about a selected axis.

To rotate by an offset:

1. Select the axis to rotate about from the **About** list.
2. Type the desired angle in the **Offset angle** box.
3. Click the **Rotate** button.

The **Rotate** button becomes active once you type a value in the **Offset angle** box or select an existing rotate offset sub-command (`ALIGNMENT/ROTATE_OFFSET`). If you select an existing rotate offset sub-command, it is modified to use the new **Offset angle** value; otherwise, a new rotate offset sub-command is created.

If you click **Rotate**, and you have a feature selected and a value entered in the **Offset angle** box, the software creates these two sub-commands:

- First, it creates a `ROTATE` command to the selected feature.
- Second, it creates a `ROTATE_OFFSET` command with a value from the **Offset angle** box.

The Edit window command line for this option reads:
`ALIGNMENT/ROTATE_OFFSET,'numeric_value',ABOUT,'AXIS'`

Changeable field: "numeric_value"

This is the value PC-DMIS rotates the alignment by in degrees of angle (for example; -14.36). The rotation is about the axis, perpendicular to the active workplane. The rotation is clockwise if the angle is negative and counter clockwise if the angle is positive.

Changeable field: "axis"

This is the axis about which PC-DMIS rotates the alignment.

If you don't select a feature, and you type in a rotation **Offset angle** value, PC-DMIS follows the right-hand rule of rotation. PC-DMIS follows the convention to treat both the negative and positive axes as positive. This means PC-DMIS always rotates about the XPLUS, YPLUS, or YPLUS axis. It does this even if you select **XMINUS**, **YMINUS**, or **ZMINUS**. Positive offset angle values go in the positive direction (counter-clockwise), and negative values go in the negative direction (clockwise). The software enables the **Rotate to** list when you select a feature that controls an axis that matches the vector of the selected feature.

Rotate by a Picked CAD Surface or Edge

To rotate the alignment by an offset angle by means of picking a surface or edge on the CAD model:

1. From the **Rotate to** list, select the axis to which you want the alignment to rotate.
2. From the **About** list, select the axis about which you want the alignment to rotate.
3. Click the **Pick in CAD** icon .

4. PC-DMIS enters a special CAD-selection mode where it disable the other dialog box controls until you either select a surface or edge from the CAD model, or you click the icon again to cancel this mode.

5. Once in this CAD-selection mode, click on the desired CAD model's surface or edge in the Graphic Display window to define the alignment rotation offset angle.

6. If the initial CAD selection does not define a valid rotation direction, PC-DMIS waits for a second CAD model selection and uses that with the first picked item to define the rotation direction.

PC-DMIS supports picking planar features, axial features (cylinders, cones, and lines), and point-type features (spheres, arcs, and ellipses).

For a given CAD feature selection, the following apply:

- If you pick a planar feature, PC-DMIS uses the plane's normal as the rotation direction.

- If you pick an axial feature, PC-DMIS uses the feature axis as the rotation direction. If the feature axis is parallel to the selected **About** axis, you must select a second feature to define the rotation direction perpendicular to the axis.

- If you pick a point-type feature, you must select a second feature to define the rotation direction.

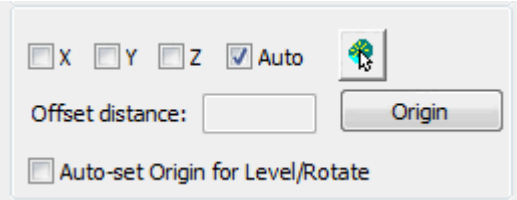
- If you picked two axial features to define the rotation direction, the axes of the two features must be parallel.

- If the picked item cannot be used to define an offset angle, PC-DMIS displays an error message and then lets you pick a different feature or cancel the CAD-selection mode.

- If the resulting angle is 0, PC-DMIS does not create an offset sub-command.

The Edit window command line for this option is identical to the "Rotate by a Manual Offset Angle" case described above.

Origin



The **Origin** area moves the alignment origin to a specific feature location, by a specific manual offset distance, or by a distance defined from a picked CAD surface or Edge.

Move to a Feature

To move the alignment origin to a feature:

1. Select the feature to move to.
2. Select the appropriate check box or check boxes (**X**, **Y**, **Z**, or **Auto**) to choose the alignment axis direction or directions to move the origin.
3. Click the **Origin** button. The **Auto** check box chooses the axes to move based on the feature type, the orientation of the feature, and the workplane.

The Edit window command line for this option reads:

```
ALIGNMENT/TRANS,ZAXIS,'feat_id'
```

Changeable fields:

"ZAXIS"

This is Edit window field that allows you to switch between the fields *ZAXIS, XAXIS, and YAXIS*. This field represents the axis that PC-DMIS moves the origin along to match the input 'feat_id'.

"feat_id"

This is the feature that the alignment sets the origin to, along the given axis. Example: CIRCLE1.

Move by a Manual Offset Distance

To move the origin by a manually-specified offset distance:

1. Select the appropriate check box or check boxes (**X**, **Y**, **Z**, or **Auto**) to choose the alignment axis direction or directions to move the origin.
2. Type the desired value in the **Offset distance** box.
3. Click the **Origin** button.

When you click **Origin**, if you have a feature selected, and you have a value entered in the **Offset distance** box, the software creates two sets of sub-commands:

- First, it creates a set of `TRANS` commands to translate to the selected feature, based on the selection state of the **X**, **Y**, **Z**, and **Auto** check boxes.
- Second, it creates a set of corresponding `TRANS_OFFSET` commands using the value from the **Offset distance** box.

The Edit window command line for this option reads:

```
ALIGNMENT/TRANS_OFFSET,ZAXIS,'numeric_value'
```

Changeable fields:

"ZAXIS"

This Edit window field allows you to switch between the fields ZAXIS, XAXIS, and YAXIS. This field represents the axis along which the origin moves. The distance moved is based on the 'numeric_value'.

"numeric_value"

This is the value by which the alignment is offset (for example, 5.12). A positive value moves along the defined axis in the positive direction. A negative value moves in the negative direction.

Move by a Picked CAD Surface or Edge

To move the origin by an offset by means of picking a surface or edge on the CAD model:

1. Select the appropriate check box or check boxes (**X**, **Y**, **Z**, or **Auto**) to choose the alignment axis direction or directions to move the origin.
2. Click the **Pick in CAD** icon .
3. PC-DMIS enters a special CAD-selection mode where it disables the other dialog box controls until you either select a surface or edge from the CAD model or you click the button again to cancel this mode.
4. Once in this CAD-selection mode, click on the desired CAD model's surface or edge in the Graphic Display window to move the alignment to the picked item.

PC-DMIS supports picking planar features, axial features (cylinders, cones, lines), and point-type features (spheres, arcs, ellipses).

For a given CAD feature and set of alignment axis directions:

- If there is a unique solution point between the CAD feature and the specified alignment axis directions, PC-DMIS moves the alignment to that point as allowed by the selected alignment axes (such as a single alignment axis and a planar feature).

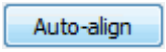
- If there is more than a single solution point, PC-DMIS moves the alignment to the solution point nearest the current alignment location as allowed by the selected alignment axes (such as two alignment axes and a planar feature intersecting at a line).
- Marking the **Auto** check box behaves the same as selecting **X**, **Y** and **Z**.
- If PC-DMIS cannot use the selected CAD entity to define an offset, it displays an error message and then lets you pick a different feature or cancel the CAD-selection mode.
- If the resulting offset distance for a specific alignment direction is 0.0, no corresponding offset sub-command is created for that direction.

The Edit window command line for this option is identical to the "Move by a Manual Offset Distance" case described above.

Auto-Set Origin for Level/Rotate

The **Auto-set Origin for Level/Rotate** check box works together with the **Level** and **Rotate** buttons. If it is selected when you select a feature and click **Level** or **Rotate**, PC-DMIS uses the same feature used to define the level or rotate sub-command to automatically create one or more translate origin commands. If there are two features selected (for a rotate command), PC-DMIS only uses the first feature. The translation commands are based on the current **X**, **Y**, **Z**, or **Auto** settings in the **Origin** area.

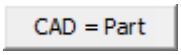
Auto-align



The **Auto-align** button uses the currently-selected features to automatically create the alignment sub-commands. The **Auto-align** button is active only when you select exactly one, two, or three features from the list of available features, and there are no sub-commands currently defined for the alignment. Auto-align uses the same algorithms as QuickAlign. Auto-align supports all valid combinations of features selected.

For more information on QuickAlign, see "About QuickAlign".

CAD = Part



The **CAD = Part** (CAD Equals Part) button moves and orients the part origin defined by the alignment so that it is equal to the CAD origin. You should use this option after a created alignment places the part origin and orientation at the same location as the CAD origin and orientation. This can make it simpler to use the CAD data to help inspect the part by displaying the measured data directly on top of the CAD data.

To set the CAD equal to the part:

1. Measure features on the part or fixture.
2. Use the alignment options to create an alignment.
3. Select the **CAD = Part** button. After the **CAD = Part** button is selected for a part, the **Operation | Graphic Display Window** | **CAD Equals Part** menu item will be selected.

The **CAD = Part** button is only available if the alignment is at the very end of the measurement routine. If there are other commands after the alignment, PC-DMIS hides the button.

Creating a 3-2-1 Alignment

The information presented here describes the steps you'll need to take to create a standard 3-2-1 alignment.

Click this icon from the **Wizards** toolbar to access PC-DMIS's 3-2-1 Alignment Wizard.

Step 1: Measure the Alignment Features

The first thing you will need to do is measure the features used to create the 3-2-1 alignment. A 3-2-1 alignment uses three standard feature types for its creation. The numbers 3, 2, and 1 refer to the minimum number of hits you need to take on in order to measure these features.

- **Measure a Plane.** The first feature is the *leveling* feature and should be a *plane* composed of *three* hits. PC-DMIS will level the part to this feature. This defines the origin and direction of the first axis (usually the Z axis).
- **Measure a Line.** The second feature is the *rotation* feature and should be a *line* composed of *two* hits. PC-DMIS will rotate the part to this feature, orienting the second axis. The second hit of this feature should be in the positive direction of the axis, relative to your first hit. This feature defines the direction of the second axis (usually the X axis) and the origin of the third axis (usually the Y axis).
- **Measure a Point.** The third and final feature is the *origin* feature composed of just *one* hit. Since PC-DMIS creates the origin for this axis from your first two features, the third point merely establishes the origin of the entire alignment. PC-DMIS will translate the part to this feature, making it the X=0, Y=0, and Z=0 location.

Once you've measured the necessary features, you're ready to create your alignment.

Step 2: Level, Rotate, and Translate to the Features

This step uses the **Alignment Utilities** dialog box (**Insert | Alignment | New**) to level, rotate, and translate your part to the features measured in the previous step.

1. Access the **Alignment Utilities** dialog box (**Insert | Alignment | New**). Notice that the **Feature** list shows all the possible features that can be used to level, rotate, or translate the part.

2. From the **Feature** list, click on the *plane feature* you created in the previous step. Once PC-DMIS selects the feature, select the axis to which PC-DMIS should level the part, and then click the **Level** button. PC-DMIS will display a line of text inside the **Alignment Utilities** dialog box informing you of the feature and axis to be used in the leveling process.

3. From the **Feature** list, click on the *line feature* you created in the previous step. Once PC-DMIS selects the feature, select what axis to rotate to and what axis to rotate about, and then click the **Rotate** button. Again, PC-DMIS displays the feature and axis to be used for the rotation process.

4. From the **Feature** list, click on the *point feature* you created in the previous step. Once PC-DMIS selects the feature, select the appropriate axis check box to determine which axis (or axes) you want to move to this origin feature, and click the **Origin** button.

Alternately, you can select all three of the features from the **Feature** list and click the **Auto Align** button to have PC-DMIS automatically level to the first feature selected, rotate to the second feature selected, and translate the axes to the third feature selected.

Now you're ready to complete the alignment creation process.

Step 3: Completing the Alignment

To complete the alignment:

1. Ensure that the information in the **Alignment Utilities** dialog box (**Insert | Alignment | New**) is correct.

2. When you're ready, click the **OK** button. PC-DMIS closes the dialog box. If this new alignment differs from the existing alignment, PC-DMIS displays a prompt asking if you want to update affected commands in the Edit window to use the new alignment (see "Updating Commands in Learn Mode" in the "Updating Dependent Commands When the Alignment Changes" topic). If the alignment doesn't change (or the change

is too small to matter), PC-DMIS simply inserts the alignment without displaying the prompt or updating any commands.

3. PC-DMIS inserts the commands for the alignment into the Edit window, and graphically shows the alignment on the CAD model inside the Graphic Display window.

4. You can edit the alignment's code at any time by using the techniques described in the "Using the Edit Window" chapter.

About QuickAlign

QuickAlign is designed so you can create alignments from the Edit window with a single click of a toolbar icon. You can select valid combinations of one, two, or three features to create an automatic alignment. QuickAlign is based on datum precedence principles as described in the GD&T ASME Y14.5.1M standard.

The alignment is based on certain rules described in the topics below. There is no dialog box for this functionality. The alignment is automatically created directly in the Edit window in both the Summary Mode and Command Mode.

Using QuickAlign

Measure the alignment features you want to use in the QuickAlign.

From the **QuickMeasure** or **Alignment** toolbar, click the **QuickAlign** icon , or select the **Insert | Alignment | QuickAlign** menu item.

If no other user-defined alignment command exists, PC-DMIS automatically selects the features and creates the alignment depending on whether the measurement routine is in Startup mode or Standard mode.

QuickAlign creates the alignment based on:

- Feature type
- Selection order
- Feature positions relative to each other

QuickAlign constrains the degrees of freedom (DOF) as explained below:

- The first feature constrains all available DOF possible for that feature type.
- If two or more features are selected, the second feature constrains all available unconstrained DOF possible for that feature type.

- If three features are selected, the third feature constrains as many remaining unconstrained DOF possible for that feature type.
- QuickAlign aligns the machine axes closest to the theoretical axis of the features that determine the LEVEL and ROTATE.
- QuickAlign supports all valid cases of feature combinations for creating an alignment.

PC-DMIS displays the DOF that are controlled by the alignment on the Status bar.

For more information on constraining degrees of freedom, see the "Alignment Overview" topic.

QuickAlign in Startup Mode

When you first create a measurement routine, it contains a `STARTUP` command with a default alignment. For QuickAlign, the measurement routine is considered to be in "Startup mode" if it does not have any user-defined alignment commands.

If it is in Startup mode, the following apply:

- You do not need to select any features to create the alignment but you can if you desire.
- If you don't select features, QuickAlign uses up to the last three features in the measurement routine to automate the initial alignment creation.
- If you do select features, QuickAlign uses the features in the selected order. For more information on feature selection, see the "QuickAlign Feature Selection" topic.
- It recalls the STARTUP alignment automatically if the alignment constrains all six DOF. The STARTUP alignment is the default empty alignment at the start of every measurement routine. This means that the software adds no extra features to the dependency list.
- QuickAlign creates the initial manual alignment by "backfilling" partial alignments for level and rotation where applicable. For information on backfilling, see the "Alignment Backfilling for QuickAlign" topic.
- QuickAlign creates DCC features and a DCC alignment if the following conditions are met:
    - The Machine type supports DCC Mode.
    - The alignment constrains all six degrees of freedom (DOF).
    - The measurement routine is in Manual Mode.

  For more information, on DCC alignment creation, see the "DCC Alignment Creation for QuickAlign" topic.

You can have non-feature commands, like dimensions, assignments, and so on, in-between the features used for the alignment.

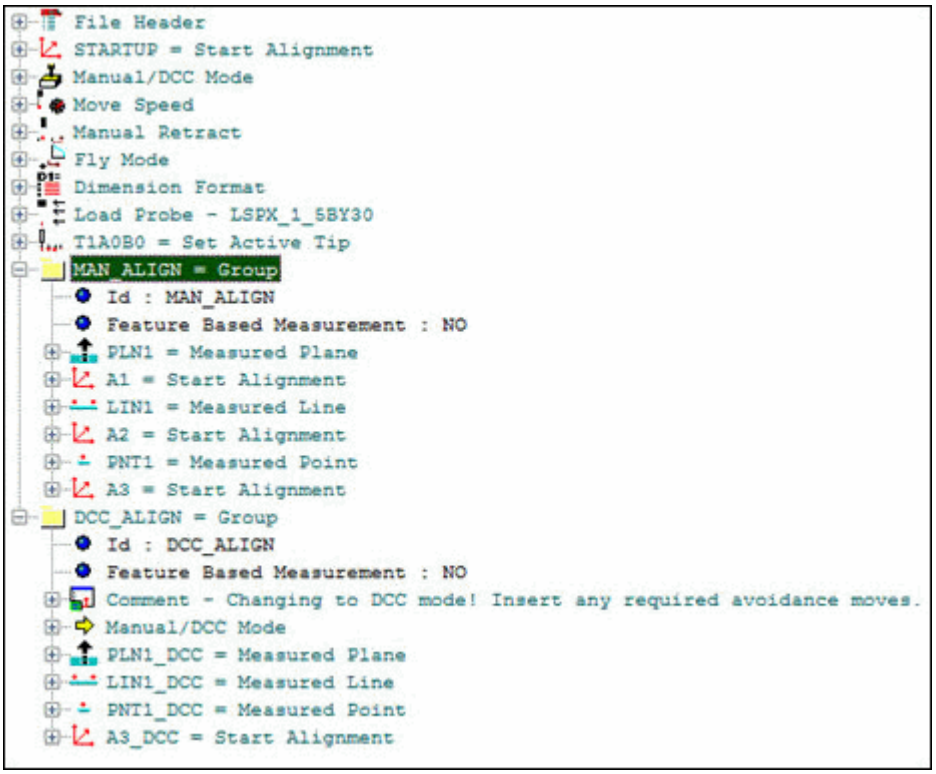
An Example

Measure a plane, line, and a point.

From the QuickMeasure or Alignment toolbar, click the **QuickAlign** icon , or select the **Insert | Alignment | QuickAlign** menu item.

QuickAlign searches up from the end of the measurement routine and collects the three newest features to use in the alignment.

It then uses the collected features to automatically generate fully constrained manual and DCC alignments.



Clearance moves or other move points are not generated. To avoid collision during probe motion, be sure to activate the ClearanceCube or insert `MOVE/POINT` commands as needed. QuickAlign does insert an Operator comment (`COMMENT/OPER`) to remind the operator to add all needed clearance avoidance commands.

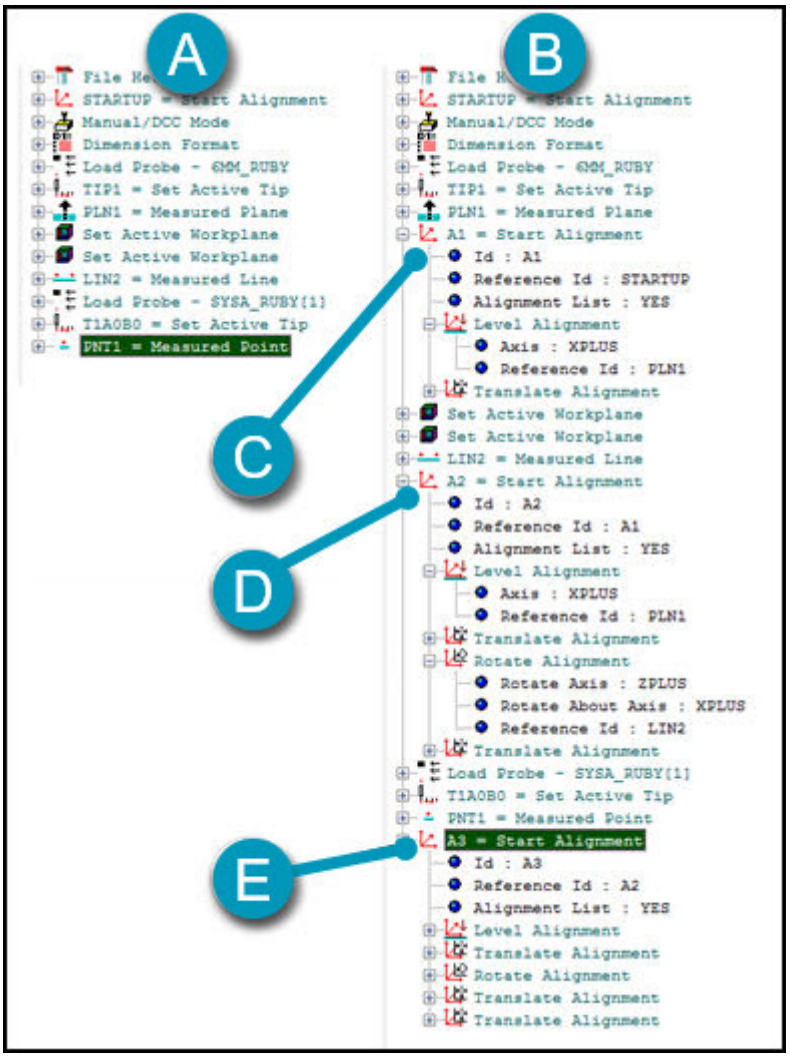
Alignment Backfilling for QuickAlign

"Backfilling" means QuickAlign looks back in the measurement routine and creates partial alignments as required in order to accurately handle probe compensation of any non-3D feature.

Back-filling occurs if these conditions are met:

- The measurement routine is in startup mode and therefore doesn't have a user-defined alignment.
- One or more selected features for the QuickAlign operation is a 2D feature, such as a line, circle, or point.
- A feature before the selected 2D feature exists that controls two degrees of rotation (LEVEL).

An Example



A. Measurement Routine before QuickAlign

B. Measurement Routine after QuickAlign

C. Back-filled alignment after PLN1

D. Back-filled alignment after LIN2

E. Final, fully constrained alignment

DCC Alignment Creation for QuickAlign

For measuring machines capable of DCC operation, QuickAlign can automatically define a DCC alignment by duplicating the features and rough manual alignment commands in DCC Mode in order to get a more accurate alignment.

DCC alignment creation can occur if these conditions are all met:

- PC-DMIS is in startup mode.
- PC-DMIS is connected to a DCC-enabled machine.
- The measurement routine is in Manual Mode.
- The features you choose for the alignment fully constrain the six degrees of freedom.

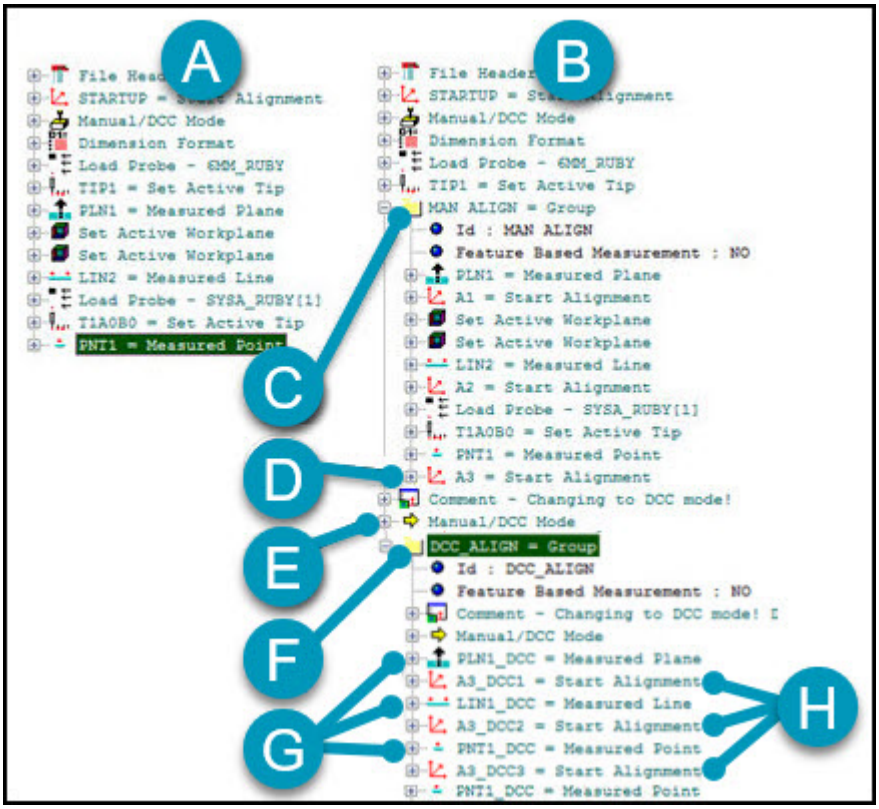
How the DCC QuickAlign Functionality Works

The QuickAlign DCC functionality does the following:

- Adds an operator comment command (`COMMENT/OPER`), to remind the operator to add all needed collision avoidance commands.
- Adds a mode command to change to DCC Mode (`MODE/DCC`).
- Copies all the feature commands used for the manual alignment, along with any feature commands they depend on, and pastes them using the same name as their original feature names, appended with "_DCC".
- If the manual features are Auto features and the probe type used is a scanning probe and the **Use Scanning Strategies for Quick Align** check box, from the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) is marked then the DCC Mode features are created using the scanning strategy defined in the Measurement Strategy Editor (MSE). Make sure that a measurement strategy is defined and selected in the Measurement Strategy Editor.
- Generates a DCC alignment command after each DCC feature that is a copy of one of the original manual features used to define the final manual alignment. The original manual feature is replaced with its DCC copy in the new DCC alignment. The resulting DCC alignment commands constrain all six DOF. Each generated DCC alignment command is named based on the last manual alignment name appended with _DCC1, _DCC2, and so on (for example, if A3 is the name of your manual alignment, and two DCC commands are generated from that alignment, they appear as A3_DCC1 and A3_DCC2).

- Recalls the STARTUP alignment automatically if the alignment constrains all six DOF. The STARTUP alignment is the default empty alignment at the start of every measurement routine. This means no extra features are added to the dependency list.
- Ensures that all constructed features in DCC Mode are constructed using features measured in DCC Mode.
- Creates an alignment using QuickAlign with features similar to a manual alignment but is measured and constructed in DCC Mode.
- Combines the manual alignments and their associated feature commands into a `GROUP` command named "MAN ALIGN".
- Combines DCC alignment and its associated features into a `GROUP` command named "DCC ALIGN".

An Example



A. Measurement Routine before QuickAlign

B. Measurement Routine after QuickAlign

C. Manual alignment group

D. Manual alignment

E. Change to DCC Mode

F. DCC alignment group

G. Copied feature commands

H. Final DCC alignments

QuickAlign in Standard Mode

If one or more user-defined alignments already exist in the measurement routine, the measurement routine is considered to be in "Standard mode".

In standard mode, the following apply:

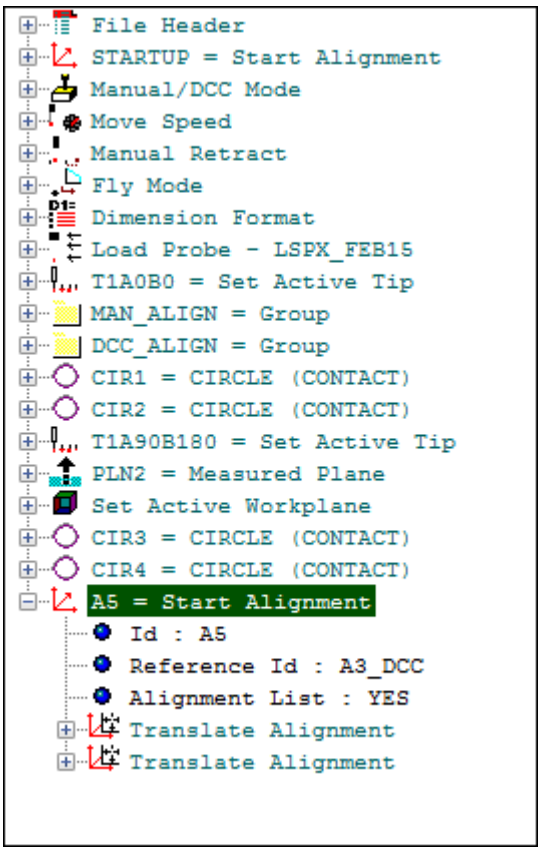
- QuickAlign does not backfill partial alignments.
- QuickAlign does not automatically create a DCC alignment.
- If you don't select features, only the last feature is used to create the alignment.
- To create an alignment with more than just one feature, you must select the features from the Edit window.
- The alignment is added at the end of the measurement routine.
- The alignment is created based on the same principles as described in the "QuickAlign in Startup Mode" topic.

An Example

Measure some features.

From the **QuickMeasure** or **Alignment** toolbar, click the **QuickAlign** icon , or select the **Insert | Alignment | QuickAlign** menu item.

QuickAlign automatically uses the last feature in the measurement routine to constrain all possible degrees of freedom for that feature type.



QuickAlign Feature Selection

You can tell QuickAlign to use features by selecting them from the Edit window. QuickAlign uses these features in the order you select them and creates the alignment.

To select multiple features in the Edit window:

1. Press and hold Ctrl.
2. From the Edit window, click on each feature to use in the alignment. The selected features are highlighted as you select them.
3. From the QuickMeasure or Alignment toolbar, click the **QuickAlign** icon , or select the **Insert | Alignment | QuickAlign** menu item.
4. QuickAlign uses the selected features to create an alignment at the *end of the measurement routine*.

Be sure to select the features in the correct precedence order. If you select more than three features, PC-DMIS disables QuickAlign.

An Example

```
STARTUP    =ALIGNMENT/START,RECALL:USE_PART_SETUP,LIST=YES
            ALIGNMENT/END
            MODE/MANUAL
            MOVESPEED/ 100
            MANRETRACT/20
            FLY/ON,3
            FORMAT/TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;NOM,TOL,MEAS,DEV,OUTTOL, ,
            LOADPROBE/LSPX_1_5BY30
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
PLN1       =FEAT/PLANE,CARTESIAN,TRIANGLE
            THEO/<143.01173,44.01082,0>,<0,0,1>
            ACTL/<143.01173,44.01082,0>,<0,0,1>
            MEAS/PLANE,4
            HIT/BASIC,NORMAL,<91.87123,101.06849,0>,<0,0,1>,<91.87123,101.06849,0>,USE THEO=YES
            HIT/BASIC,NORMAL,<227.24958,65.06429,0>,<0,0,1>,<227.24958,65.06429,0>,USE THEO=YES
            HIT/BASIC,NORMAL,<164.84462,3.97514,0>,<0,0,1>,<164.84462,3.97514,0>,USE THEO=YES
            HIT/BASIC,NORMAL,<88.08148,5.93537,0>,<0,0,1>,<88.08148,5.93537,0>,USE THEO=YES
            ENDMEAS/
LIN1       =FEAT/LINE,CARTESIAN,UNBOUNDED
            THEO/<46.81872,0,-7.89217>,<1,0,0>
            ACTL/<46.81872,0,-7.89217>,<1,0,0>
            MEAS/LINE,2,ZPLUS
            HIT/BASIC,NORMAL,<46.81872,0,-6.24759>,<0,-1,0>,<46.81872,0,-6.24759>,USE THEO=YES
            HIT/BASIC,NORMAL,<172.59529,0,-9.53674>,<0,-1,0>,<172.59529,0,-9.53674>,USE THEO=YES
            ENDMEAS/
LIN2       =FEAT/LINE,CARTESIAN,UNBOUNDED
            THEO/<0,5.05021,-27.07443>,<0,1,0>
            ACTL/<0,5.05021,-27.07443>,<0,1,0>
            MEAS/LINE,2,ZPLUS
            HIT/BASIC,NORMAL,<0,5.05021,-26.28887>,<-1,0,0>,<0,5.05021,-26.28887>,USE THEO=YES
            HIT/BASIC,NORMAL,<0,24.27247,-27.85998>,<-1,0,0>,<0,24.27247,-27.85998>,USE THEO=YES
            ENDMEAS/
PNT1       =FEAT/POINT,CARTESIAN,NO
            THEO/<0,0,-17.4833>,<1,0,0>
            ACTL/<0,0,-17.4833>,<1,0,0>
            CONSTR/POINT,INT,LIN1,LIN2
                        END OF MEASUREMENT FOR
```

QuickAlign Error Messages

If the set of feature commands does not form a valid alignment, the alignment is not created. In addition, PC-DMIS shows one or more of these error messages:

- "Unsupported feature combination!" - This message means the selected combination of features does not define a well-formed alignment.
- "<FEATURE> is invalid!" - This message means the feature represented by <FEATURE> has errors in its definition; this usually occurs with a line measured on a surface where the line's line direction vector and the surface's normal vector are not perpendicular.
- "<FEATURE> is not valid as the alignment second feature!" - This message means the feature represented by <FEATURE> cannot be used as the second feature in the alignment, due to its vector direction or its origin location.
- "<FEATURE> is not valid as the alignment third feature!" - This message means the feature represented by <FEATURE> cannot be used as the third feature in the alignment, due to its vector direction or its origin location.
- "Alignment error!" - This message indicates a general QuickAlign or Auto-align error.

If PC-DMIS creates the alignment, but it has problems with its degrees of freedom, PC-DMIS shows one of the following warning messages:

- "<FEATURE> is not used. All DOFs constrained by first two features." - This message means that all six degrees of freedom (DOF) are already constrained by first two features used to define the alignment; so the third feature isn't used to constrain any of the DOF.
- "Not all 6 DOFs constrained" - This message means that the three features used to define the alignment, do not constrain the six DOF, even though PC-DMIS creates a valid alignment.

- QuickAlign determines if an axial feature is in a given plane by checking if the vector of the feature (such as a circle, cylinder, cone, or line) is within +/- 5 degrees of the axis or vector of the primary feature.
- QuickAlign determines if an axial feature can be used for rotation by checking if it is more than 45 degrees from the primary feature vector.

Degrees of Freedom for Features in QuickAlign

The Auto-align algorithms are based on GD&T datum precedence principles and the native degrees of freedom (DOF) of each feature type.

DOF of Supported Features

The supported features are grouped into the following six DOF-based cases, detailed in this table:

DOF Case	Feature Types	DOF Constraints
Planar	Planes, 3D Widths	R1 R2 T3
Axial	Cylinder, 3D lines, Cones*	R1 R2 T1 T2
2D Line	Lines (on a surface), 2D Widths	R1 T2
1D Point	Vector Points, Round Slots*, Square Slots*	T1
2D Points	Circles, Ellipses*, Edge Points, Angle Points	T1 T2
3D Points	Spheres, Corner Points	T1, T2, T3

* These features are treated differently in the QuickAlign algorithm:

- Cone features are treated as cylinders.
- Ellipses are treated as circles.
- Slot features (both round and square) are treated as one-dimensional vector points with their surface normal vector pointing in the direction of the slot width.

Table Key:

R1 - Rotation DOF about the first coordinate axis

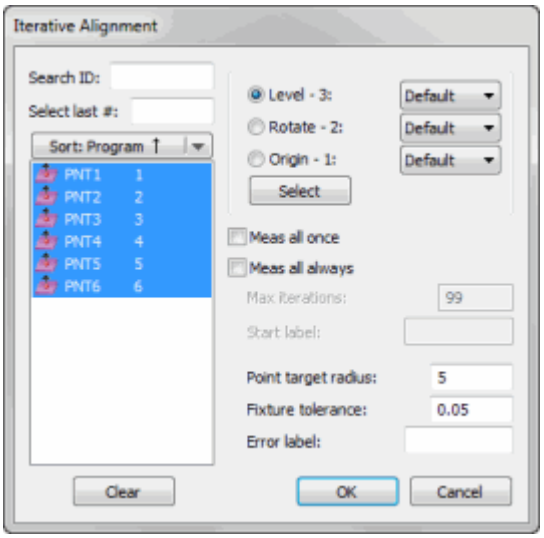
R2 - Rotation DOF about the second coordinate axis

T1 - Translation DOF in first coordinate axis direction

T2 - Translation DOF in the second coordinate axis direction

T3 - Translation DOF in the third coordinate axis direction

Creating an Iterative Alignment



Iterative Alignment Dialog Box

When you click the **Iterative** button from the **Alignment Utilities** dialog box (**Insert | Alignment | New**), PC-DMIS displays the **Iterative Alignment** dialog box. This dialog box provides a way for you to three-dimensionally "best fit" the measured data to the nominal points (or surfaces, if available).

The resulting iterative alignment coordinate system origin is approximately located at and oriented with the CAD coordinate system, even if the set of features used is located and oriented somewhere very different. This is much like an automotive system, in the sense that all the features on all the parts of the automobile's body are defined in terms of a single global coordinate system, even if they are physically located or oriented far away from that coordinate system. The CAD coordinate system serves the same function for the iterative alignment as the car body's global coordinate system.

An iterative alignment requires at least three features to be measured. Certain feature types, such as points and lines, have poor three dimensional locations. If one of these types of features is selected, additional features are needed to provide accurate measured data.

- The first set of features establishes the orientation of the normal axis of the current workplane by fitting a plane through the feature centroids. At least three features must be used in this section (LEVEL - 3 +).
- The next set of features rotates the defined axis of the workplane to the features, fitting a line through the features. At least two features need to be used in this section (ROTATE - 2 +).
  If no features have been marked, the alignment will use features from the LEVEL section. (The two features used from the LEVEL section are the second and third from last feature.)
- The last set of features translates the part origin to a specific location (SET ORIGIN - 1).

  If no features have been marked, the alignment will use the last feature from the LEVEL section.

Understanding Iterative Alignments

To properly create your iterative alignment, first consider the information presented here; these topics will help you to understand important aspects of iterative alignments.

Iterative Alignment Command Format

The Edit window command line for this option reads:

```
ALIGNMENT/ITERATE,'feat_id'
, PNT TARGET RAD=n, START LABEL=label, FIXTURE TOL = n,
ERROR LABEL=label
MEAS ALL FEAT=NO/ALWAYS/ONCE,
MAX ITERATIONS = n
LEVEL AXIS =axis, ROTATE AXIS=axis, ORIGIN AXIS=axis
LEVEL = id, id, id,...
ROTATE = id, id,...
ORIGIN = id,...
```

Changeable field: "feat_id"
These are the features that are used to do the iterative alignment. Currently, you must select at least three different features for the calibration. If features support a datum axis in more than one direction, such as circles or slots, you can specify them in more than one datum axis. For example, you can use a circle to both establish the

level axis as well as the rotation axis. Typically, you can only use measured points (including vector and surface points) to establish one datum axis.

PNT TARGET RAD = This option specifies the target radius value for the measured point features used in the alignment. See "Point Target Radius" for complete information.

START LABEL = PC-DMIS starts at the label specified here when remeasuring alignment features. You must set `MEAS ALL FEAT` to `ALWAYS` for this to work. See "Start Label" for complete information.

FIXTURE TOL = This is the tolerance that PC-DMIS uses to compare measured alignment features to their theoretical values. See "Fixture Tolerance" for complete information.

ERROR LABEL = PC-DMIS goes to the label specified here when the fixture tolerance level is exceeded. If you don't define a label, PC-DMIS generates an error message showing the amount of error on each of the input features. See "Error Label" for complete information.

LEVEL AXIS = PC-DMIS uses the **LEVEL** input features to set the orientation and origin of the axis specified here. See "Level" for complete information.

ROTATE AXIS = PC-DMIS uses the **ROTATE** input features to set the rotation of the axis specified here about the level axis. The origin of the axis specified here is also set by PC-DMIS using the **ROTATE** input features. See "Rotate" for complete information.

ORIGIN AXIS = PC-DMIS uses the **ORIGIN** input features to set the origin of the axis specified here. See "Origin" for complete information.

MEASURE ALL FEATURES = This option either re-measures the input features or automatically re-executes a portion of the measurement routine in DCC Mode. There are three possible settings for this option:

- `NO` - See "Point Target Radius" for complete information.
- `ONCE` - See "Meas All Once" for complete information.
- `ALWAYS` - See "Meas All Always" for complete information.

MAX ITERATIONS = This option determines the maximum number of iterations that PC-DMIS performs for this iterative alignment. PC-DMIS only uses this value if the **Meas All Always** check box is selected.

Iterative Alignment Rules

There are some general rules when doing an iterative alignment:

PC-DMIS needs both the measured values and theoretical values for each of the elements in the sets. The normal vectors for the first set of elements must be approximately parallel. The one exception to this rule is if only three features are being used in the set.

If measured points (VECTOR, EDGE, or SURFACE) are being used, then all three sets of elements are needed (three features to Level, two features to Rotate, and one feature to Set Origin) to define the alignment. Any feature type can be used, but three-dimensional elements are better defined elements and therefore improve the accuracy. Some of the possible 3D elements are sheet metal circle, slot, cylinder, sphere, or a corner point.

The sheet metal circle, slot, or cylinder needs at least three sample hits.

The difficulty of using measured points lies in not knowing where to take the measurement until after the alignment. This poses a problem as the points must be measured before the alignment. Three dimensional elements, by definition for this use, are elements that can be measured precisely the first time.

Further, if measured points (VECTORS, EDGE, or SURFACE) are being used, the features in the ROTATE set must have normal vectors that are approximately perpendicular to the vectors of the features in the LEVEL set. The features in the ORIGIN set must have a normal vector that is approximately perpendicular to both the vectors from the LEVEL set and the ROTATE set.

If measured points (VECTORS, EDGE or SURFACE) are being used as part of the set, PC-DMIS may ask that they be re-measured if they were taken too far from the nominal location. PC-DMIS first "best fits" the measured data to the nominal data. Next, PC-DMIS checks to see how far off each measured point is. If the distance is greater than the amount specified in the **Point Target Radius** box, PC-DMIS will request that the point be re-measured. PC-DMIS in effect puts a cylindrical tolerance zone around the theoretical location of each vector, surface, or edge point. The radius of this tolerance zone is the point tolerance specified in the dialog. PC-DMIS will continue to re-measure point features until all the measured points fall into 'tolerance'. The tolerance zone only affects measured points.

A special capability of PC-DMIS allows the center point of a slot to slide up or down the axis as needed. For this reason, an iterative alignment cannot converge when a slot is used as part of the ORIGIN set. It is possible to use a slot as part of the ORIGIN set by first constructing a point from the slot and then using that constructed point in the ORIGIN set.

A slot is not recommended as part of the ORIGIN set of an iterative alignment.

Type of Feature used:	Minimum # of Features needed:	
Circle	3 circles:	This method uses 3 DCC circles for the alignment.
Line	This feature type is not recommended.	
Point	6 points:	The points are used as a 3-2-1 alignment.
Slot	This feature type is not recommended as part of the ORIGIN set.	
Sphere	3 spheres:	This method uses 3 spheres for the alignment.

To Create an Iterative Alignment

Click this icon from the **Wizards** toolbar to access PC-DMIS's Iterative Alignment Wizard.

To create an iterative alignment:

1. Access the **Alignment Utilities** dialog box (**Insert | Alignment | New**).
2. Click the **Iterative** button. The **Iterative Alignment** dialog box appears. You will use this dialog box to create the iterative alignment. See "Description of the Iterative Alignment Dialog Box" if you need information about the dialog box.
3. From the **Feature List** box, select the first set of features (at least three features) to be used in establishing the orientation of the normal axis on the current workplane.
4. Verify that the **Level** option is selected.
5. Click the **Select** button.
6. Select with the mouse the second set of features (at least two features) to be used in the rotation process.
7. Verify that the **Rotate** option is selected.

8. Click the **Select** button.
9. Select the final set of features (at least one feature) that indicate the desired location of the part origin. (The same features may be used in more than one process.)
10. Verify that the **Origin** option is selected.
11. Click the **Select** button.
12. Click the **OK** button. The **Iterative Alignment** dialog box closes.
13. Click the **OK** button on the **Alignment Utilities** dialog box to complete the alignment. The dialog box closes. If this new alignment differs from the existing alignment, PC-DMIS will display a prompt asking if you want to update affected commands in the Edit window to use the new alignment (see "Updating Commands in Learn Mode" in the "Updating Dependent Commands When the Alignment Changes" topic). If the alignment doesn't change (or the change is too small to matter), PC-DMIS simply inserts the alignment without displaying the prompt or updating any commands.

Selecting the **Level**, **Rotation**, or **Origin** options after they've already been assigned features will display the indicated input features for that option.

After this process is complete, PC-DMIS will three-dimensionally "best fit" the measured data and display the new alignment in the Graphic Display window and in the Edit window. See the "Iterative Alignment Command Format".

Description of the Iterative Alignment Dialog Box

The following describes the items included in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button).

Level

The **Level - 3** option in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) is used in conjunction with at least three features selected from the **Feature List** box. This set of features establishes the orientation of the normal axis of the current workplane by fitting a plane through the feature centroids.

At least three features must be used to level.

Rotate

The **Rotate - 2** option in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) is used in conjunction with at least two features selected from the **Feature List** box. This set of features rotates the defined axis of the workplane to the features, fitting a line through the features.

At least two features need to be used to rotate.

If no features have been marked, the alignment uses features from the LEVEL section. (The two features used from the LEVEL section are the second and third from last feature.)

Origin

The **Origin - 1** option in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) is used in conjunction with one feature selected from the **Feature List** box. This feature set translates (or moves) the part origin to a specific location

One feature must be used to set the origin.

If no features have been marked, the alignment uses the last feature from the LEVEL section.

Select

The **Select** button in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) enables you to use selected features from the **Feature List** box to perform the Leveling, Rotating, and translating (or moving) to the Origin operations for an iterative alignment.

Meas All Once

If you select the **Meas All Once** check box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button):

- PC-DMIS remeasures all the input features at least once in DCC Mode.

- The input features are measured in the order specified by the iterative alignment command in the Edit window.
- PC-DMIS displays which feature is about to be measured in a message box.
- Before accepting the move, ensure that the probe can reach the indicated feature(s) without colliding with the part.
- Stored moves found before or after each feature are *not* executed.
- After all the features have been measured at least once, PC-DMIS continues to remeasure the features for measured point types of features and those points that missed their **Point Target Radius** target (see "Point Target Radius").

Since their location never changes, PC-DMIS only measures circles once while in this mode.

Meas All Always

If you select the **Meas All Always** check box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button), PC-DMIS remeasures (or re-executes) a portion of the current measurement routine at least once in DCC Mode. The portion that is re-executed depends on the Start Label (for information, see "Start Label"):

With a Start Label

If you provide a start label, PC-DMIS re-executes in DCC Mode from that defined label to the `ALIGNMENT/START` command that contains the currently executing iterative alignment command.

Without a Start Label

If you don't provide a start label, the following occurs:

- PC-DMIS begins the DCC re-execution from the first feature measured in the measurement routine that is used by the iterative alignment command.
- If the first feature has any stored move points preceding it then PC-DMIS also executes those move points.
- PC-DMIS continues the re-execution of the measurement routine commands until the last measured feature used by the iterative alignment command.
- PC-DMIS does not execute any stored moves after this command.
- Once the re-execution finishes, PC-DMIS recalculates the alignment and tests any measured input points to see if they are all within the tolerance radius specified in the **Point Target Radius** value.

- If they are all in the target radius, the re-execution does not need to continue, and PC-DMIS considers the iterative alignment command complete.
- If any points missed the target area, the same portion of the measurement routine is re-executed as described above.

Move Points and Meas All Always During First Execution

A move point contains an XYZ value to which the probe moves during execution. If you mark **Meas All Always** but PC-DMIS is in Manual Mode then during the measurement routine execution of the iterative alignment, PC-DMIS redefines all the move point locations to be relative to the iterative alignment's coordinate system in which they are executed. This only happens once, on the very first execution of the iterative alignment. If you later add additional move points and then re-execute the iterative alignment, the new move points are also redefined to be relative to the iterative alignment, similar to the other move points.

Point Target Radius

Use the **Point Target Radius** box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) to specify the target radius tolerance for measured point features used as inputs in the alignment. Measured input points include the following:

- Measured Points (MEAS/POINT)
- Auto Vector Point (AUTO/VECTOR)
- Auto Edge Point (AUTO/EDGE)
- Auto Surface Point (AUTO/SURFACE)
- Auto Angle Point (AUTO/ANGLE)

While you can usually see the location needed to measure a circle on a part easily enough, determining the exact location to measure a point on the surface isn't easily done. Without any visual indicators to tell you where to measure the point, it is difficult to manually measure the point in an exact spot. The **Point Target Radius** specifies an imaginary tolerance zone (or target) the size of the radius, around each point. This allows you to take a manual hit anywhere within the indicated tolerance when you execute the measurement routine. If the measured point does not fall within this zone, PC-DMIS re-measures the point in DCC Mode.

PC-DMIS attempts to re-measure the input features based on the selected check boxes in the **Iterative Alignment** dialog box (see "Meas All Once" and "Meas All Always").

If you don’t select either the **Meas All Always** or **Meas All Once** check box (or if you manually set `MEASURE ALL FEAT=NO` in the Edit window),

- PC-DMIS tries to calculate an alignment transform so that the measured datums match the theoretical datums, and then it checks to see if any measured input points missed their targets. If so, only these features are re-measured in DCC Mode.
- PC-DMIS displays a dialog box indicating the feature that is about to be measured. This lets you ensure that the probe can reach the desired feature without colliding with the part.
- Once all point features are in target, PC-DMIS considers the iterative alignment command complete.
- If there are any measured point features that missed their targets, PC-DMIS continues to re-measure these features until they are in target.

Be careful not to set the **Point Target Radius** value too small (for example 50 microns). Many CMMs are unable to accurately position the probe to touch each measured point on a minuscule target. A better choice is a tolerance of about .5 millimeters. If the re-measurement continues indefinitely, increase the value.

Fixture Tolerance

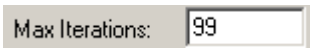
The **Fixture Tolerance** box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) enables you to type a fitting tolerance value against which PC-DMIS compares feature elements making up the iterative alignment to their theoretical values.

If after fitting the measurement values to the theoretical values, one or more of the input features have an error along their assigned datum axis that exceeds this tolerance value, PC-DMIS automatically goes to the error label (if one exists). See "Error Label".

If you don't provide an error label, PC-DMIS displays an error message showing the errors along each of the datums. You will have the choice to accept the datum as it is and continue with the rest of the measurement routine, or to cancel the measurement routine execution.

PC-DMIS can only use the fixture tolerance value if you used more than the minimum number of points needed to create the feature. For example, if you are measuring a plane, the minimum number of points needed for that plane is usually three points. However, if you wanted to use the fixture tolerance value, you would need to measure at least four points. If you use only three points, then only one solution exists, and PC-DMIS cannot adjust or re-iterate.

Max Iterations



This box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) determines the maximum number of repetitions that PC-DMIS performs when creating the iterative alignment. PC-DMIS only uses this value if the Meas All Always check box is selected.

Start Label

The **Start Label** box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) allows you to define a label that PC-DMIS execution moves to when remeasuring iterative alignment features. It remains disabled until you mark the **Meas All Always** check box.

For information on how the **Start Label** works with **Meas All Always** during execution, see the "Meas All Always" topic.

For more information on labels, see "Using Labels" in the "Branching by Using Flow Control" chapter.

Error Label

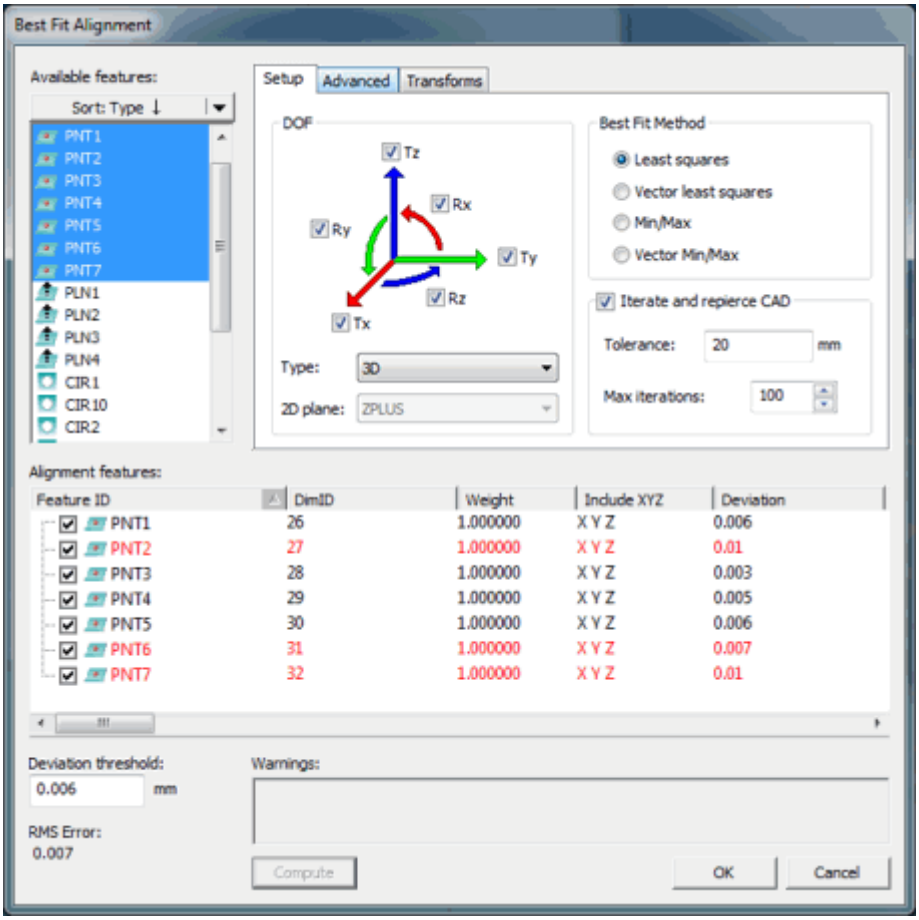
Use the **Error Label** box in the **Iterative Alignment** dialog box (**Insert | Alignment | New | Iterative** button) to define a label that the measurement routine flow will go to if the error in matching the measured level, rotate, and origin datum features to their corresponding theoretical features exceeds the fixture tolerance level in the **Fixture Tolerance** box.

If you supply the minimum number of inputs for each of the datum axes (three for the **Level** datum, two for the **Rotate** datum, and one for the **Origin** datum), PC-DMIS can fit the input feature measurement value to its theoretical values without errors. In this case PC-DMIS doesn't really need the fixture tolerance. If you supply more than the minimum number of inputs for any of the defined datums, then either part or fixture errors may make it impossible to fit the measurement values to the theoretical values with less error then the supplied fixture tolerance.

If you don't define an error label, PC-DMIS will generate an error message displaying the amount of error for each of the datum features, giving you the option to cancel or continue execution with the datums as they are.

To create a label, see "Using Labels" in the "Branching by Using Flow Control" chapter.

Creating a Best Fit Alignment



Best Fit Alignment Dialog Box

When you click the **Best Fit** button in the **Alignment Utilities** dialog box (**Insert | Alignment | New**), PC-DMIS displays the **Best Fit Alignment** dialog box. This dialog box provides a way for you to "best fit" the measured data to the nominal points. With the exception of the **Vector** method, which requires at least two points, you need at least one point feature to create a best fit alignment.

To Create a Best Fit Alignment

To create a Best Fit alignment:

1. Access the **Alignment Utilities** dialog box (**Insert | Alignment | New**).
2. Click the **Best Fit** button to open the **Best Fit Alignment** dialog box. You can use the dialog box to create the Best Fit alignment. For details on the **Best Fit Alignment** dialog box, see the "Description of the Best Fit Alignment Dialog Box" topic in the PC-DMIS Core documentation.
3. From the **Available features** list, select the features that you want to use. PC-DMIS shows the selected features in the **Alignment features** list box.

4. To define the orientation and degrees of freedom of the alignment, select either the **2D**, **3D** or **User Defined** options from the **Type** list. For a 2D alignment, select the correct active plane from the **2D Plane** list.

5. From the **Best Fit Method** area, select the best fit type.

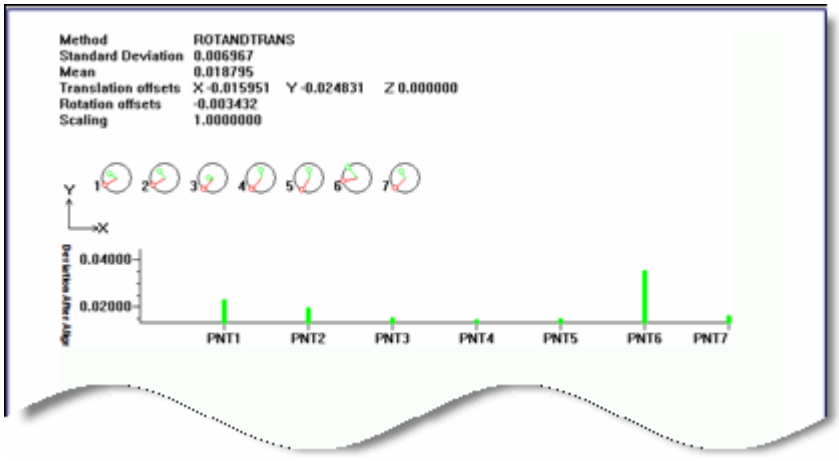
6. To edit the weights of the features, double-click its weight value in the **Alignment features** list. The weight value cell turns into an inline edit field. Edit the value, and then return or click outside the cell to finalize the change.

7. To set the point of rotation about a given feature, select the feature from the **Input List**, and then click the **Set** button in the **Rotate About** area of the **Advanced** tab. Alternately, you can enter a value into the **Theoretical** and **Measured** boxes in the **Rotate About** area of the dialog box.

8. Click the **OK** button to close the **Best Fit Alignment** dialog box.

9. Click the **OK** button on the **Alignment Utilities** dialog box. The dialog box closes. If this new alignment differs from the existing alignment, PC-DMIS displays a prompt asking if you want to update affected commands in the Edit window to use the new alignment (see "Updating Commands in Learn Mode" in the "Updating Dependent Commands When the Alignment Changes" topic). If the alignment doesn't change (or the change is too small to matter), PC-DMIS simply inserts the alignment without displaying the prompt or updating any commands. Following the execution of the measurement routine, PC-DMIS displays a 3D Alignment Best Fit Graphical Analysis in the Report window.

Example of a Best Fit Alignment's Graphical Analysis in the Report Window



Example of a Best Fit alignment graphical analysis in the Report Window.

This graphical analysis of the 3D Best Fit alignment displays this information in the Report window:

- **Header** - This contains various values used in the Best Fit alignment: Method, Standard Deviation, Mean, Translation offsets, Rotation offsets, Max iterations, Iterations.

- **Vertical Axis** - This shows the amount of deviation before and after the alignment.

  Red bars on bar graphs or red dots on circle graphs represent the deviation, the 3D distance between the actual and the theoretical, *before* the Best Fit alignment.

  Green bars on bar graphs or green dots on circle graphs represent the deviation, the 3D distance between the actual and the theoretical, *after* the Best Fit alignment.

- **Horizontal Axis** - This displays the IDs of the points used in the alignment.

Note that 3D alignments only use the bar graph.

Understanding Best Fit Alignments

A Best Fit alignment is an alignment that causes a set of measured points or a set of actual feature centroids to match, as closely as possible to its nominal location or theoretical counterpart. In certain cases, it is also possible for a Best Fit alignment to optimally match a set of points to a CAD curve or surface.

To understand the different Best Fit Alignment methods, see "The Best Fit Alignment Methods".

About Exclusions

For 2D Vector Best Fit Alignments - Features with vector directions that are perpendicular to the 2D best-fit plane are excluded from the alignment calculation.

For All Best Fit Alignments - Features that are skipped during measurement routine execution are excluded from the alignment calculation.

If all features of the alignment are excluded, PC-DMIS displays an error message. If this occurs during execution, PC-DMIS pauses the execution and gives you the option of canceling execution. You can check the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button) to see if any of the alignment's features were excluded due to invalid vectors or because they were skipped during execution.

The Best Fit Alignment Methods

This topic covers the available Best Fit Alignment solution methods when you define a Best Fit Alignment.

The Best Fit Alignment minimizes deviations between measured point data and nominal point data. The features you use in the alignment require correct nominal values in their THEO fields.

The alignment methods below minimize the deviations in different ways:

Method 1: Least Squares Fit

Command Mode: LEAST_SQR

Description: The Least Squares algorithm aligns the two point sets. It does this by transforming the measured points in such a way that it minimizes the sum of the squared distances (between matching measured and nominal points). This is the same as minimizing the average squared error. To understand how the Least Squares alignment works, visualize a spring (of initial length zero) between each measured point and its nominal counterpart. As the distance between the two points increases, the springs stretch. The point set's final position when you release it and let the springs act is the solution to the Least Squares alignment problem.

When to Use It: Use Least Squares to reverse engineer parts and to troubleshoot manufacturing processes, when aligning to feature centers. These feature centers come from regular features such as circles or cylinders.

Least Squares is the most common best fit type because it produces very repeatable results. Least Squares is also used to troubleshoot the manufacturing process, because it gives a better representation of all the point data, rather than just the most extreme points (see the Min Max description below). Least Squares is not ideal for complex surfaces where data is a set of points with unique vectors. In that case, a Vector Least Squares fit is a better option.

Other Info: This is the default. Least Squares works with 2D, 3D, and **User Defined** best fit alignments.

Algorithm: The least squares algorithm minimizes the sum of the squared lengths of the deviations, meaning it minimizes the following mathematical function:

$$\mathcal{L}_{\text{least squares}} = \sum_i w_i \|\mathbf{d}_i\|^2,$$

where w_i are the weights.

Method 2: Vector Least Squares Fit

Command Mode: VECTOR_LST_SQR

Description: The Vector Least Squares fit is a kind of Least Squares fit, except that it projects the error vectors onto the nominal vectors. It then uses these projected distances in the Least Squares fit. A vector fit type allows points to "slide" along the surface but discourages them from pulling away from the surface. All errors are along the nominal vectors.

When to Use It: Use Vector Least Squares to reverse engineer parts and to troubleshoot manufacturing processes, when aligning to surfaces. These surfaces might be regular features, such as circles or cylinders, or they might be irregular complex surfaces. The surface may be represented as a set of points with unique vectors.

For instance, suppose the points are on the surface of a car hood. In this case, motion along the surface is not as important as motion perpendicular to the surface. Therefore, we use a vector fit to only measure deviations perpendicular to the surface.

Other Info: This is also known as Projected Least Squares. Vector Least Squares works with standard 2D and 3D best fit alignments, not **User Defined** best fit alignments.

Suppose you have a nominal point of 1,1,1 with a vector of 0,0,1, with measured values of 4,2,0.95. With this fit, the software adjusts the measured data to 1,1,0.95, and snaps it to the vector of 0,0,1.

Algorithm: The vector least squares algorithm minimizes the sum of the squared projected deviations, where the deviations are projected onto the nominal vectors $\mathbf{v}_i$, meaning it minimizes the following function:

$$\mathcal{L}_{\text{vector least squares}} = \sum_i w_i (\mathbf{d}_i \cdot \mathbf{v}_i)^2.$$

Method 3: Min Max Fit

Command Mode: MIN_MAX

Description: A Min Max fit minimizes the maximum error (the largest distance) among the features being fit.

When to Use It: Use Min Max to evaluate Position tolerances according to the ASME and ISO standards, when tolerancing feature centers. These feature centers come from regular features, such as circles or cylinders.

The Min Max algorithm applies simultaneous diametric tolerance zones to the feature centers. Min Max fit types comply with ASME Y14.5 and ISO 1101.

Min Max fits are very sensitive to measurement uncertainty. A single bad feature can affect the alignment results drastically.

The recommended input feature types for Min Max fits to evaluate position tolerances are circles, spheres, cylinders, and cones. Min Max is not ideal for complex surfaces where data is a set of points with unique vectors. In that case, a Vector Min Max fit is a better option.

Other Info: Min Max works with 2D and 3D best fit alignments, not **User Defined** best fit alignments.

Algorithm: The Min Max algorithm minimizes the maximum deviation length, meaning it minimizes the following function:

$$\mathcal{L}_{\text{min-max}} = \max_i \, w_i \, \|\mathbf{d}_i\| \, .$$

Method 4: Vector Min Max Fit

Command Mode: VECTOR_MIN_MAX

Description: The Vector Min Max fit is a kind of Min Max fit, except that it projects the error vectors onto the nominal vectors. It then uses these projected distances in the Min Max fit.

A vector fit type allows points to "slide" along the surface but discourages them from pulling away from the surface. All errors are along the nominal vectors. A Min Max fit minimizes the maximum deviation (the largest distance) among the features being fit.

When to Use it: Use Vector Min Max to evaluate Profile tolerances according to the ASME and ISO standards, when tolerancing surfaces. These surfaces might be regular features such as circles or cylinders or they might be irregular complex surfaces. Such surfaces include parts like blades and airfoils. The surface may be represented as a set of points with unique vectors.

For instance, suppose the points are on the surface of a car hood. In this case, motion along the surface is not as important as motion perpendicular to the surface. Therefore, we use a vector fit to only measure deviations perpendicular to the surface.

Make sure there is a good initial alignment of the machine to the part. Min Max fit types comply with ASME Y14.5 and ISO 1101.

Other Info: Vector Min Max works with 2D and 3D best fit alignments, not **User Defined** best fit alignments.

Algorithm: The Vector Min Max algorithm minimizes the maximum projected deviation, so it minimizes the function:

$$\mathcal{L}_{\text{vector min-max}} = \max_i \, w_i \, |\mathbf{d}_i \cdot \mathbf{v}_i| \, .$$

2D Best Fit Command Line Format

The Edit window command line for the Best Fit 2D option reads:

```
ALIGNMENT/BF2D,TOG1,TOG2,CREATE WEIGHTS=NO,TOG3,USE
SCALING=YES,n1,n2,n3,n4,n5

ITERATEANDREPIERCECAD=YES,TOLERANCE=n6,MAX ITERATIONS=n7,

CENTER OF ROTATION,MEAS_X,MEAS_Y,MEAS_Z,THEO_X,THEO_Y,THEO_Z

SHOWALLINPUTS=YES,SHOWALLPARAMS=YES

ID=
```

Available Fields:

"**TOG1**" This field allows you to switch between the available workplanes. The current workplane should be displayed.

"**TOG2**" This field allows you to switch between the available types of best fits: **LEAST_SQR**, **VECTOR_LST_SQR**, **MIN_MAX**, **VECTOR_MIN_MAX**. For information on Best Fit alignment methods, see the "The Best Fit Alignment Methods" topic in the PC-DMIS Core documentation.

CREATE WEIGHTS= This option lets you determine whether or not PC-DMIS creates weights for the features used in the best fit alignment. Available options are **YES** or **NO**. The created weights correspond to tolerances on the features used in the alignment. For information on feature weight, see the "Feature Weights" topic in the PC-DMIS Core documentation.

"**TOG3**" This field determines the degrees of freedom for the 2D alignment. Available options are: **ROTONLY** (rotation only), **ROTANDTRANS** (rotation and translation), and **TRANSONLY** (translation only).

USE SCALING = This option is available if TOG2 is set to LEAST_SQR. If set to **YES**, PC-DMIS computes a transformation (rotation and translation) and a scale factor to optimally match nominal data to the scaled measured data. For information on the **Use Scaling** check box, see the "Description of the Best Fit Alignment Dialog Box" topic in the PC-DMIS Core documentation.

n1,n2,n3= Translation in X,Y,Z

n4= The angle offset value.

n5= This is the scale factor. It only appears if **USE SCALING** is set to **YES**.

ITERATEANDREPIERCECAD= If set to **YES**, PC-DMIS will perform an Iterative Best Fit alignment by piercing the CAD geometry and adjusting the feature nominals with each iteration, using TOLERANCE= and ITERATIONS= values to control the result. If set to **NO**, PC-DMIS does a single best fit alignment. For information on the **Use Scaling** check box,

see the "Description of the Best Fit Alignment Dialog Box" topic in PC-DMIS Core documentation.

TOLERANCE=**n6**. This is the tolerance value for performing an iterate and repierce cad operation. This only appears if **ITERATEANDREPIERCECAD** is set to **YES**.

ITERATIONS=n7. This is the maximum number of times the best fit alignment algorithm iterates when performing an iterate and repierce cad operation. This only appears if **ITERATEANDREPIERCECAD** is set to **YES**.

CENTER OF ROTATION This field and the measured and theoretical XYZ values associated with it, represent the center of rotation. They only appear if ROTONLY or ROTANDTRANS is used for the TOG2 field. MEAS_X, MEAS_Y, and MEAS_Z are the X,Y,Z values for the measured center of rotation. THEO_X, THEO_Y, and THEO_Z are the theoretical X,Y,Z values for the theoretical center of rotation.

SHOWALLINPUTS= This option lets you determine whether or not the alignment code block displays the feature inputs used to create the alignment. Available options are **YES** or **NO**.

SHOWALLPARAMS= This option lets you determine whether or not the alignment code block displays all the parameters for the feature inputs. Available options are **YES** or **NO**.

If set to **YES**, then PC-DMIS displays this information for each input feature: Feature ID, Feature Type, Dimension ID, Feature Weight, Feature Use.

For example, it might look something like this:

```
ID = CIR2,Circles,LOC12,2.000000,YES
```

If set to **NO**, then PC-DMIS displays only the ID of the input feature:

```
ID = CIR2
```

ID= Each line began with "ID=" represents an input feature used in the alignment.

3D Best Fit Command Line Format

The Edit window command line for the Best Fit 3D option reads:

```
ALIGNMENT/BF3D,TOG1,CREATE WEIGHTS=YES,TOG2,USE
SCALING=YES,n1,n2,n3,n4,n5,n6,n7

ITERATEANDREPIERCECAD=YES,TOLERANCE=n8,MAX ITERATIONS=n9,

CENTER OF ROTATION,MEAS_X,MEAS_Y,MEAS_Z,THEO_X,THEO_Y,THEO_Z

SHOWALLINPUTS=YES,SHOWALLPARAMS=YES

ID=
```

Available Fields:

"**TOG1**" This field allows you to switch between the available types of best fits.

"**TOG2**" This field allows you to switch between the available types of constraints for the 3D alignment. Available options are: **ROTONLY** (rotation only), **ROTANDTRANS** (rotation and translation), and **TRANSONLY** (translation only).

n1,n2,n3 - Translation in X,Y,Z

n4 - Rotation in the XY plane

n5 - Rotation in YZ plane

n6 - Rotation in ZX plane

Note that the values for translation and rotation are relative to the current active alignment, and the angles are always in degrees.

USE SCALING= This option is available if TOG2 is set to LEAST_SQR. If set to **YES**, PC-DMIS computes a transformation (rotation and translation) and a scale factor to optimally match nominal data to the scaled measured data. See "Description of the Best Fit Alignment Dialog Box" where the **Use Scaling** check box is discussed.

n7= This is the scale factor. It only appears if USE SCALING is set to YES.

If seven number values are visible then the seventh number is the scale factor.

CENTER OF ROTATION This field and the measured and theoretical XYZ values associated with it, represent the center of rotation. They only appear if ROTONLY or ROTANDTRANS is used for the TOG2 field. MEAS_X, MEAS_Y, and MEAS_Z are the X,Y,Z values for the measured center of rotation. THEO_X, THEO_Y, and THEO_Z are the theoretical X,Y,Z values for the theoretical center of rotation.

ITERATEANDREPIERCECAD= If set to **YES**, PC-DMIS will perform an Iterative Best Fit alignment by piercing the CAD geometry and adjusting the feature nominals with each iteration, using TOLERANCE= and ITERATIONS= values to control the result. If set to **NO**, PC-DMIS does a single best fit alignment. See "Description of the Best Fit Alignment Dialog Box" where the **Use Scaling** check box is discussed.

TOLERANCE=n8. This is the tolerance value for performing an iterate and repierce cad operation. This only appears if **ITERATEANDREPIERCECAD** is set to **YES**.

ITERATIONS=n9. This is the maximum number of times the best fit alignment algorithm iterates when performing an iterate and repierce cad operation. This only appears if **ITERATEANDREPIERCECAD** is set to **YES**.

SHOWALLINPUTS= This option lets you determine whether or not the alignment code block displays the feature inputs used to create the alignment. Available options are **YES** or **NO**.

SHOWALLPARAMS= This option lets you determine whether or not the alignment code block displays all the parameters for the feature inputs. Available options are **YES** or **NO**.

If set to **YES**, then PC-DMIS displays this information for each input feature: Feature ID, Feature Type, Dimension ID, Feature Weight, Feature Use.

For example, it might look something like this:

```
ID = CIR2,Circles,LOC12,2.000000,YES
```

If set to **NO**, then PC-DMIS displays only the ID of the input feature, like this:

```
ID = CIR2
```

ID= Each line that starts with "**ID=**" represents an input feature used in the alignment.

Best Fit Alignment Types

You can have Best Fit Alignments solve 2D, 3D, or custom user-defined degrees of freedom.

(For a user-defined approach, in the **Type** list of the **Best Fit Alignment** dialog box, select **User Defined**).

Some important differences between these best fit alignment types are:

- A 2D Best Fit alignment requires an initial alignment to establish the 2D plane. The alignment is created in the workplane defined by the current active alignment.
- A 3D Best Fit alignment uses the raw (machine) data and correlates it to the theoretical values. It does not require a previous alignment, but if the degrees of freedom are set to rotation-only and a rotation center is not explicitly defined, it uses the origin of the current active alignment as the rotation center.
- In most cases, a user-defined Best Fit alignment requires a previous alignment in order to reliably establish the XYZ directions for the custom degrees of freedom.

Constraints for Best Fit Alignments

There are also constraints that can be applied to the Best Fit alignments.

Constraints for 2D and 3D Best Fit Alignments

1. **Rotate and Translate** (default) - This allows the alignment full flexibility while correlating the machine data to the theoretical values.

2. **Rotate Only** - This option restricts the alignment to rotation only, without applying any translations of the rotation center. If no rotation center is defined, the part coordinate system origin (0,0,0) is used as the rotation center.

3. **Translate Only** - This option restricts the alignment to translation only, without applying any rotations.

Constraints for User-Defined Best Fit Alignments

The user-defined Best Fit alignment allows any degree-of-freedom combination other than cases with two rotations (zero, one, and three rotations are allowed). It does not support defining a rotation center. For relevant degree-of-freedom combinations, it uses the part coordinate system origin (0,0,0) as the rotation center.

Feature Weights

The purpose of weights is to aid in simultaneous evaluation of Position or Profile tolerances.

You can specify feature weights, or you can have PC-DMIS automatically generate feature weights based on the tolerances' values.

Specified Weights

Each feature used as an input has an associated weight. The default value for these weights is 1. You can modify the weights in the Edit window (**View | Edit Window**) or in the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button). The values of these weights affect the resulting alignment. The larger the weight of a particular feature, the more the resulting alignment will attempt to match the measured value of that feature to its theoretical value. This association allows you to assign priorities to the features in the alignment. If the weights of all the input features are equal, then the features will be treated equally *regardless* of the value of the weight.

You can edit any of the weights by selecting the feature in the **Alignment features** list and double-clicking the specific weight value. Edit the value as needed and then click outside the field to set the new value. The new value will then be assigned to that feature and used during the calculations.

Weights may also be assigned to each feature based on its associated dimension. If there is no associated dimension, it will be assigned a default tolerance. Assign the weights by clicking the **Set** button. A composite weight for the feature is then computed by multiplying the user-defined weights with the tolerance weights.

Tolerance-Generated Weights

For tolerance-based weights, a Min_Max fit reduces the percentage of the available tolerance that each feature uses. The Least Squares fit reduces the "average" amount of tolerance that all features use.

Because the generated weights are reciprocals of the tolerances, a feature with a relatively small weight (or lower priority) corresponds to a large tolerance zone. This gives it more freedom to move without affecting the other features. A feature with a relatively large weight (or small tolerance zone) gets a high priority in the alignment process.

Using Weights for Simultaneous Evaluation

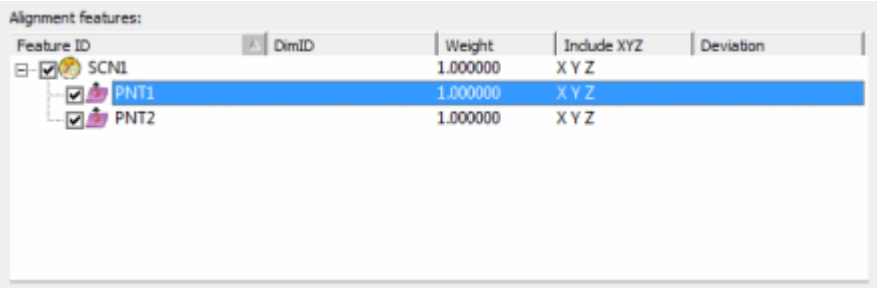
1. Create some dimensions with the appropriate tolerances. Choose "no fit" for these dimensions. Do not include these dimensions in the report. These dimensions are for-reference-only. These are only present to inform the best-fit alignment what the tolerances are.

2. Create a best-fit alignment that automatically creates weights on the toleranced features. The best-fit alignment takes into account the tolerance sizes and any applicable bonus as it creates the weights.

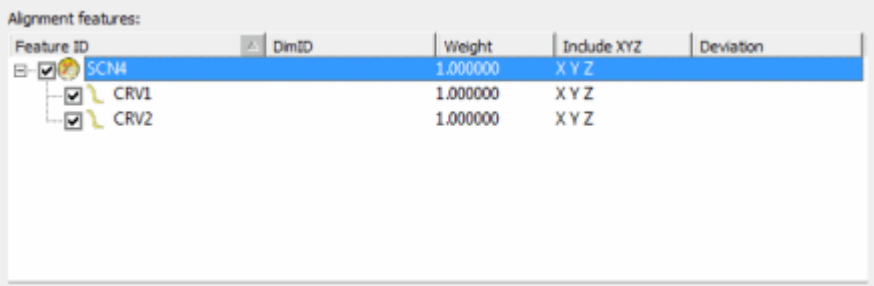
3. Create a second set of dimensions with the appropriate tolerances, using the weighted best-fit alignment. Include these dimensions in the report.

Constructed Feature Sets and Curves as Best Fit Alignment Inputs

When using a constructed feature set or a constructed curve as inputs to the Best Fit alignment, a plus (+) sign will appear next to the feature ID in the **Alignment features** list box in the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button). When you click on the plus (+) sign once, the features that make up that set or curve will be displayed. Initially the weights for each feature will be the same as its parent feature (the original set or curve).



Example of the Alignment features list box showing features in a set.



Example of the Alignment features list box showing features in a curve.

You can edit the weight for each child feature of the set or curve by double-clicking the current weight value and entering the new value as necessary. To change the weight of all of the features in the set or curve, edit the weight on the feature set or the curve itself. That weight will be transferred to all of its children.

Scans as Best Fit Alignment Inputs

When working with a scan, there is one extra component than when working with a feature set. Scans are composed of basic scans. Each basic scan is composed of individual points. When clicking a plus (+) sign next to a scan, it will expose all of the basic scans associated with it. A plus (+) sign will appear next to each basic scan. By clicking the plus (+) sign next to each basic scan, it will expose all of the individual points associated with the basic scan. The weight for each point can be edited, the weight of the basic scan (and all of its points) can be edited, or the weight of the scan itself can be edited.

Tagged Features in a Feature Set or Scan

Also, you can tag individual members within a feature set so that they are not used. To tag a member of a set to not be used in the best fit alignment, clear the check box to the left of the feature in the **Alignment features** list box in the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button). If a check box for a basic scan is cleared, it and all of its associated points will *not* be used in the calculations.

Alignment Center of Rotation

For 2D and 3D Best Fit alignments, you can specify a center of rotation in one of two ways:

- Method 1: Select a feature from the **Alignment features** list in the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button). Next, click the **Set** button. The theoretical and measured values are automatically entered into the appropriate boxes in the **Rotate About** area.
- Method 2: If a specific coordinate is desired, manually provide the values by typing the XYZ coordinate into the **Theoretical** and **Measured** boxes in the **Best Fit Alignment** dialog box. The number values must be in an X,Y,Z comma-delimited format in order to be valid. The values must be entered in terms of part coordinates with regards to the active alignment trihedron.

If you do not specify a center of rotation, PC-DMIS uses the part coordinate system origin (0,0,0) as the rotation center.

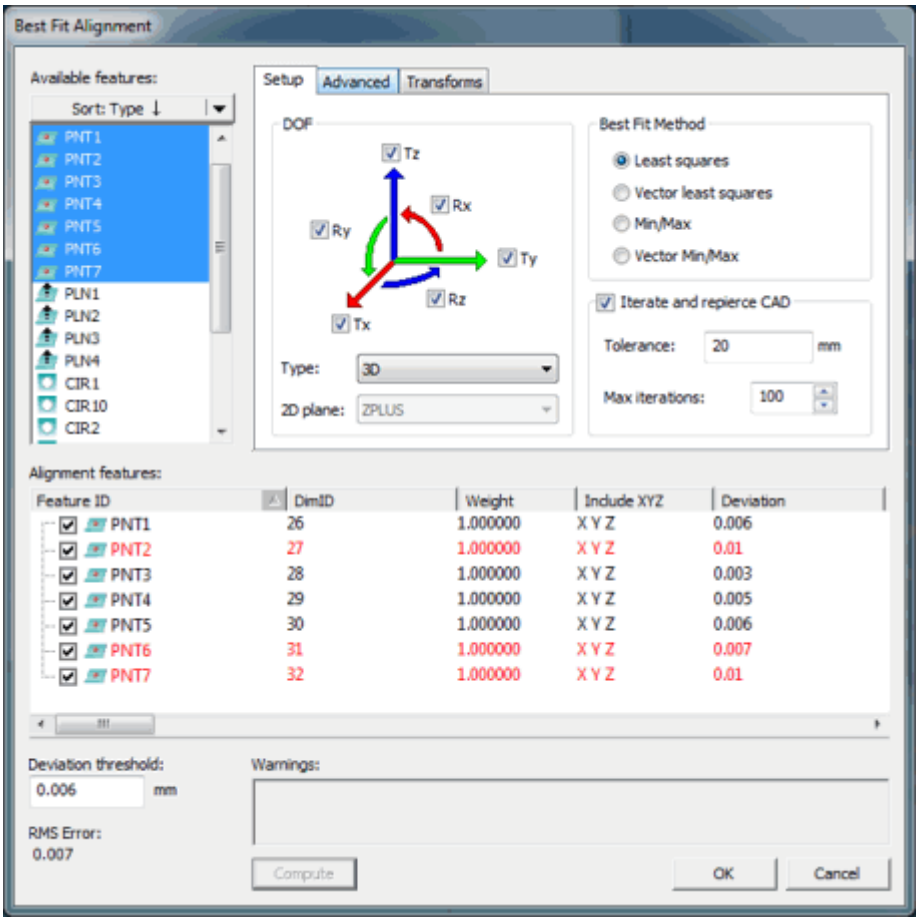
For user-defined Best Fit alignments, you cannot specify the rotation center. In this case, the part coordinate system origin (0,0,0) is always used for degree-of-freedom combinations where a rotation center is relevant.

Point Set to CAD Alignments

By default, PC-DMIS matches the measured points in your best fit alignment to the original nominal point set. However, by enabling **Iterate and Repierce CAD** (see the **Iterate and Repierce CAD** area discussed in "Description of the Best Fit Alignment Dialog Box"), you can have the best fit alignment match the measured points to CAD curves or surfaces instead. In this case, after the first best fit alignment is computed, updated nominal points on the CAD curve or surface, corresponding to the transformed measured points, are calculated. This process is repeated until convergence takes place. This alignment method changes the theoretical values of the points.

If the re-pierce operation fails for all of the measured points included in the Best Fit alignment, PC-DMIS will display an error message in the Warnings area and abort the alignment calculation.

Description of the Best Fit Alignment Dialog Box



Best Fit Alignment Dialog Box

The following features appear on all of the tabs in the **Best Fit Alignment** dialog box (**Insert | Alignment | New | Best Fit** button):

Available features list
The **Available features** list contains a list of all the features in your measurement routine before the alignment. For a full description, see "Dialog Box Description" and "Feature List box".

Alignment features list
The **Alignment features** area lists the features selected for the best fit calculation from the **Available features** list. You can sort each column by clicking on the column header. The check box to the left of each feature ID enables or disables the feature in the best fit calculation. If marked, the feature is used in the calculation; otherwise, it isn't.

If you select the **Pause during execution** check box (discussed below), the alignment is automatically reset to include all features in the best fit calculation during measurement routine execution.

If a calculated deviation is above the **Deviation threshold** value, the text for that feature is displayed in red.

You can modify weights by double-clicking the specific **Weight** factor for the feature in the **Alignment Features** list.

You can determine which axes are included in the calculation by double-clicking the specific item in the **Include XYZ** column for the feature. This displays the **Include XYZ** dialog box. Marking an axis includes the axis in the calculation. Clearing an axis excludes it.

> You cannot change the **Weight** and **Include XYZ** settings of a scan's hits independent of their parent scan feature. Each hit uses the **Weight** and **Include XYZ** settings of its parent scan.

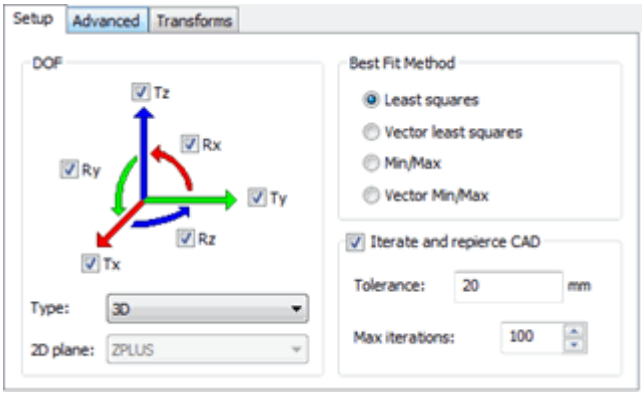
Deviation threshold box
The **Deviation threshold** sets the maximum acceptable value for the calculated deviation value. If a calculated deviation is above this threshold value, the text for that feature is displayed in red in the **Alignment features** area. The measurement units are the same as those defined in your system settings.

RMS Error
This displays the **RMS Error** value from the most recent alignment calculation.

Warnings area
The **Warnings** area displays any error messages from the most recent alignment calculation.

Compute button
The **Compute** button updates the best fit alignment calculation using the current features and settings.

Setup Tab



Setup Tab

The **Setup** tab contains the most common settings for defining a best fit alignment. The following features are specific to this tab:

DOF (Degree of Freedom) area

DOF Check Boxes

The **Tx**, **Ty**, **Tz**, **Rx**, **Ry**, and **Rz** check boxes define which of the six degrees of freedom (rotation about X, Y, or Z axis, and translation in the X, Y, or Z direction) to include in the alignment. As the degrees of freedom are modified, the value of the **Type** list (**2D**, **3D**, **3D No Rotation**, and so on) is automatically updated according to the current settings. If the current combination of degrees of freedom does not match any of the pre-defined cases, the type is set to **User Defined**. Conversely, as the type is changed, the check boxes are updated to match.

Type

The **3D** option causes the resulting alignment to be a 3D alignment that is both Translated and Rotated.

The **3D No Translation** option causes the resulting alignment to be a 3D alignment that is Rotated but not Translated.

The **3D No Rotation** option causes the resulting alignment to be a 3D alignment that is Translated but not Rotated.

The **2D** option causes the resulting alignment to be a 2D alignment that is both Translated and Rotated in the selected 2D plane.

The **2D No Translation** option causes the resulting alignment to be a 2D alignment that is Rotated but not Translated in the selected 2D plane.

The **2D No Rotation** option causes the resulting alignment to be a 2D alignment that is Translated but not Rotated in the selected 2D plane.

2D Plane

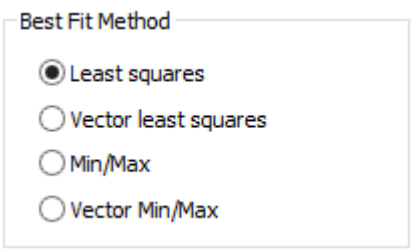
The **2D Plane** list sets the plane in which to calculate a 2D alignment.

If **Type** is set to **User Defined**, PC-DMIS automatically selects the Least squares option in the **Best Fit Method** area. The **2D plane** list on this tab and the **Rotate About** options on the **Advanced** tab are then unavailable.

The **2D Plane** list is available only if you choose one of the 2D options from the **Type** list. Available options are ZPLUS, XPLUS, YPLUS, ZMINUS, XMINUS, and YMINUS.

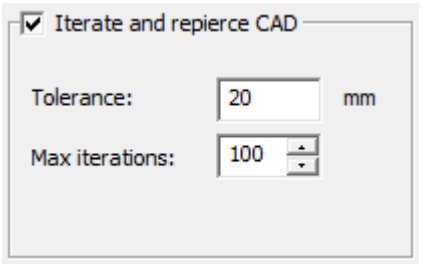
The coordinate image dynamically updates based on the marked vector or coordinate check boxes.

Best Fit Method area



The **Best Fit Method** area contains several methods that you can use to compute the best fit alignment. For information on these methods, see "The Best Fit Alignment Methods".

Iterate and repierce CAD area



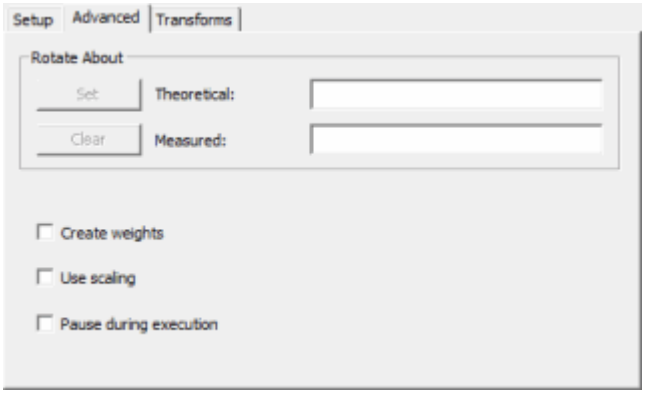
Iterate and repierce CAD - If you select this check box, PC-DMIS performs an Iterative Best Fit alignment after you click **OK**. This pierces the CAD geometry and adjusts the feature nominals with each iteration using Tolerance and Max iterations to control the result. If this check box is cleared, PC-DMIS does a single Best Fit alignment.

Tolerance - Type a tolerance value. PC-DMIS uses this tolerance when searching for CAD surfaces to pierce. The measurement units for the tolerance is the same as those used by the measurement routine. The new nominal point becomes the CAD point closest to the actual feature, provided it is within this tolerance. If no CAD surface is found within this distance from the actual feature, then that feature is ignored in subsequent iterations.

Max iterations - This determines the maximum number of times the best fit alignment algorithm iterates.

If the repierce operation fails for all of the features included in the Best Fit alignment, PC-DMIS displays an error message in the **Warnings** area and aborts the alignment calculation.

Advanced Tab



Advanced Tab

The **Advanced** tab contains specialized settings for several less-common conditions. The following are features specific to this tab:

Rotate About area
This area has two boxes that define the theoretical and measured center of rotation. You can click **Set** to automatically populate these fields with X, Y, and Z values from the feature selected in the **Alignment features** list, or you can manually type your own X, Y, and Z values.

The **Theoretical** box contains the *theoretical* center of rotation for 2D and 3D best fit alignments.

This **Measured** box contains the *measured* center of rotation for 2D and 3D best fit alignments.

The **Clear** button clears both fields.

Use scaling check box

The **Use scaling** check box becomes available for 2D or 3D alignments when you select the **Least squares** method. It is not available for alignments with specified constraints.

When you use scaling, PC-DMIS computes a transformation (rotation and translation) and a scale factor that optimally matches the nominal data to the scaled measured data. The scaled alignment also scales all of the measured data and subsequent measured features in the measurement routine, multiplying it by the computed scale factor.

Once scaling has been applied to the measured data and features in a measurement routine, it cannot be undone. Only mark **Use scaling** on in a single alignment in the measurement routine.

You may find this useful, for example, in compensating for expansion or contraction of a part due to temperature.

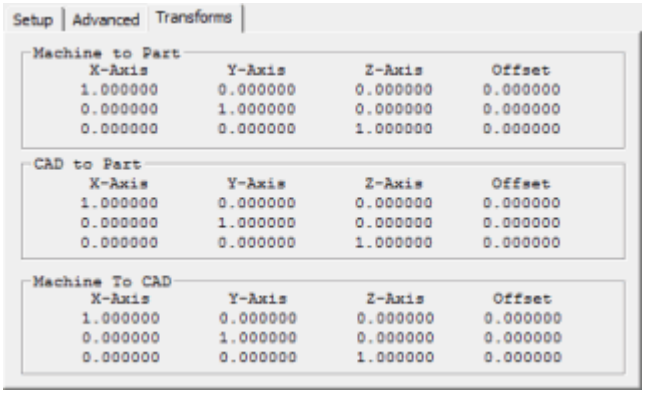
Create weights check box
If you select this check box, the software scales the values in the **Weights** column located in the **Alignment features** list by the tolerance values of the dimensions that control each feature.

If you clear this check box, the dimension tolerances are not factored into the calculation. Only the weights you defined are included.

Pause during execution check box
If you select this check box, the software causes the measurement routine to pause execution and display the **Best Fit Alignment** dialog box. This allows you to examine the magnitude of the error in the alignment solution, identify any features or scan hits that are bad, eliminate them from the alignment, recalculate, and then repeat that process until you are satisfied with the alignment. When you click **OK**, measurement routine execution resumes.

> This option is only available for portable measurement devices. For non-portable machine types, it is hidden and ignored during measurement routine execution.

Transforms Tab



Transforms Tab

The **Transforms** tab displays the alignment transforms from the most recent best fit alignment calculations. These transforms represent the cumulative state of the measurement routine up to and including this best fit alignment. The following sections are specific to this tab:

Machine to Part area - Displays the calculated Machine to Part transform values.

CAD to Part area - Displays the calculated CAD to Part transform values.

Machine to CAD area - Displays the calculated Machine to CAD transform values.

Saving an Alignment

To save the current alignment in an external file that can be recalled by a different measurement routine, select **Insert | Alignment | Save**. It also inserts a `SAVE/ALIGNMENT` command into the Edit window.

The topics presented here describe the **Save Alignment As** dialog box and how to save the alignment so that you can use it in other measurement routines.

Description

- The **Save in** list and other familiar folder controls let you navigate to the directory to which you will be saving the alignment.
- **File name** - Enables you to name the file of the alignment you are saving. By default, the file name is based off of the **Active Alignment** box when the dialog box opens.
- **Save as type** - Shows all the .aln files in the current directory. Only those files with the alignment extension (*.aln) will be shown.
- **Active Alignment** - Defines the alignment name that will be saved to the external alignment file. The alignment saved is the current (or active) alignment in use based on the current insertion point in the Edit window.
    - If the active alignment is from an `ALIGNMENT/START` command, it will display the alignment's name.
    - If it is from a `RECALL/ALIGNMENT, EXTERNAL` command, it will display the external filename without the ".aln" extension.
    - If it is from a `RECALL/ALIGNMENT, INTERNAL` command, it will display the name of the recalled internal alignment plus ':INTERNAL'.
- **Inches** or **Millimeters** - Sets the alignment to be saved with this type of measurement unit.
- **Machine to Parts** - Stores only the machines to parts transformation matrix.
- **Both** - Stores both the CAD to parts transformation matrix and the machine to parts transformation matrix.

To Save an Alignment

Note that you only need to save the alignment using the following procedure if the alignment is going to be recalled into a *different* measurement routine. All alignments are automatically saved when used within a measurement routine.

To save an alignment:

1. Select **Insert | Alignment | Save**. The **Save Alignment As** dialog appears.
2. Type an alignment name (maximum ten characters) in the **File name** box.
3. Select either the **Inches** or **Millimeters** option to save the alignment as either inches or millimeters, respectively. The default unit of measurement for any alignment will be the same unit of measurement used by the measurement routine for which the alignment was created. To use an alignment in a different measurement routine, it is not necessary that the alignment's units of measurement be saved in the units type of the new measurement routine. The alignment will automatically convert to the same units as the new measurement routine (For information on recalling an alignment, see "Recalling an Existing Alignment").
4. Select either the **Machine to Parts** or **Both** option. Select **Both** to store both the machine-to-parts and the CAD-to-parts transformation matrices. Select **Machine to Parts** to store only the machine-to-parts transformation.
5. Click the **Save** button.

You must provide a file name in order to save. The only file name extension permitted is ".aln". If you don't type a valid name for the alignment file, clicking **Save** will not close the dialog box.

The alignment can be saved to any directory. However, to use the saved alignment file in a measurement routine, it must be saved the same directory as the measurement routine or in the user-specified **Recall Directory**.

Whether you choose the **Machine to Parts** option or the **Both** option depends on the following:

- If the part alignment origin matches the CAD origin, then it is not necessary to include the CAD to Part transform. Either option will work.
- If the part alignment origin differs from the CAD origin in either location or orientation, it is necessary to include the CAD to Part transform. Choose **Both**.
- If the measurement routine does not include a CAD model, there is no need to include the CAD

The Edit window command line for this option reads:

```
SAVE /ALIGNMENT,filename, TOG1
```

TOG1

This toggle field switches between `BOTH` and `MACHINETOPARTS`. Select `BOTH` to store both the machine to parts and the CAD to parts transformation matrices. Select `MACHINETOPARTS` to store only the machine to parts transformation.

Editing the Save/Alignment Command

You can edit the `SAVE/ALIGNMENT` command by placing the mouse on the command line and pressing F9. The **Save Alignment As** dialog box opens. You can use this dialog box to edit all the settings for the alignment (file name, units, saved transformation matrix) and then save the changes as a new or existing alignment file. Once you make your changes, and click **Save**, PC-DMIS saves the alignment file and applies the changes to the Edit window's `SAVE/ALIGNMENT` command.

See the "Saving an Alignment" topic for a description of the dialog box.

Recalling an Existing Alignment

The **Insert | Alignment | Recall** menu option recalls an alignment that was previously created in the current measurement routine (Internal Alignment) or saved from another measurement routine (External Alignment). You can also recall an alignment using the **Alignments** list on the **Settings** toolbar. For information, see the "Settings Toolbar" topic.

> **!** You must save alignments (.aln) in the same folder as your measurement routine (.prg), or in a sub-folder where you saved your measurement routine. If not, you must use the **Search Path** dialog box (**Edit | Preferences | Set Search Path**) to specify where to recall your alignments.

For example, if you save your measurement routine in the "C:\Users\Public\Documents\Hexagon\PC-DMIS\2021.2" folder, and you save your alignment in "C:\Users\Public\Documents\Hexagon\PC-DMIS\2021.2\folder1\folder2", you can use this command in the Edit window to recall the alignment:

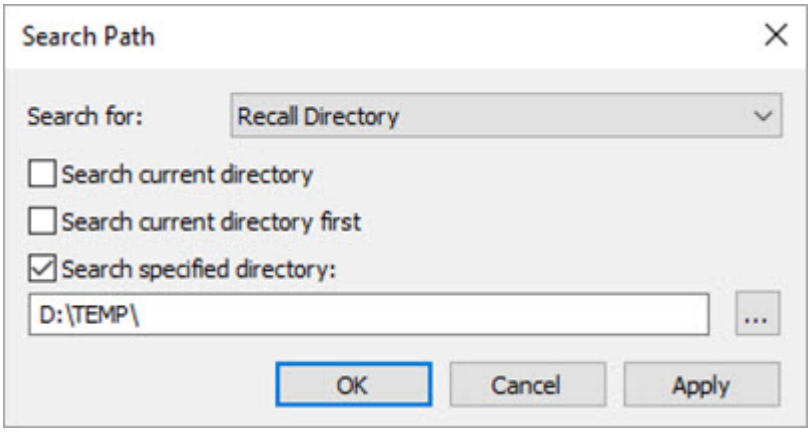
```
RECALL/ALIGNMENT,EXTERNAL,FOLDER1/FOLDER2/A1
```

However, if you save your measurement routine in a different drive, such as "D:\TEMP\folder1", you must define the recall path in the **Search Path** dialog box (**Edit | Preferences | Set Search Path**).

To do this:

1. Open the **Search Path** dialog box.
2. From the **Search for** list, select **Recall Directory**.
3. Select the **Search specified directory** check box.
4. Enter the folder location you want PC-DMIS to search in to recall your files.
5. Click **Apply** and then **OK** to set the recall location and close the **Search Path** dialog box.

For this example, the **Search Path** dialog box will look like this:



Search Path Dialog Box

You can now use this command in the Edit window to recall the alignment:

```
RECALL/ALIGNMENT,EXTERNAL,A1
```

PC-DMIS uses the path you set for the **Recall Directory** option in the **Search Path** dialog box for an A1 alignment.

Even if you save your alignment in a new sub-folder to the location you defined, PC-DMIS can locate the alignment. For example, you can create the folder "D:\TEMP\folder1". Because this is a sub-folder of the path defined in the **Search Path** dialog box, you can use this command successfully without making any further changes in the dialog box:

```
RECALL/ALIGNMENT,EXTERNAL,FOLDER1/A1
```

You can only insert the command outside an alignment block.

*The Alignment Block is the block of text in the Edit window that defines the alignment. It consists of the **ALIGNMENT/START** command and ends with the **ALIGNMENT/END** command.*

The **Recall** list in the **Alignment Utilities** dialog box (**Insert | Alignment | New**) also enables you to recall an existing alignment, but it only enables you to recall alignments previously created in that measurement routine (internal alignments).

Before an alignment can be recalled to another measurement routine, you must save it to a folder with the **Insert | Alignment | Save** menu option. For information on how to do this, see "Saving an Alignment".

If the alignment you want to recall has different units of measurement than the current measurement routine, its measurement units are automatically converted to the units of measurement of the current measurement routine.

All recall external alignment commands re-load their external alignment file when the measurement routine is first opened. During this process, if the new alignment transforms differ from the existing alignment, PC-DMIS displays a prompt asking if you want to update affected commands in the Edit window to use the new alignment. For more information, see "Updating Commands When Loading a Measurement Routine" in the "Updating Dependent Commands When the Alignment Changes" topic.

To Recall an Alignment

To recall an alignment using the **Recall** menu or **Alignment Utilities** dialog box:

1. To access the **Select Alignment** dialog box, select **Insert | Alignment | Recall**; or access the **Alignment Utilities** dialog box (**Insert | Alignment | New**) and select an alignment from the **Recall** list.
2. Type the 15 character (or less) saved alignment ID or use the list to select the desired alignment.
3. Click **OK** to insert the recall command into the Edit window (`RECALL/ALIGNMENT`).

To recall an alignment using the **Settings** toolbar (**View | Toolbar | Settings**):

1. Use the **Alignments** list in the **Settings** toolbar and select the desired alignment.
2. PC-DMIS inserts the recall command into the Edit window (`RECALL/ALIGNMENT`).

Recall Alignment Command Line Format

The Edit window command line for this option reads:

```
RECALL/ALIGNMENT, INTERNAL, 'align_id'

RECALL/ALIGNMENT, EXTERNAL, FILE_NAME
```

This command does not need to be bounded by the `ALIGNMENT/START` or `ALIGNMENT/END` commands.

Code Used to Recall an Internal Alignment

```
RECALL/ALIGNMENT, INTERNAL, 'align_id'
```

align_id

This is the internal alignment that will be recalled from within the current measurement routine.

Example:

```
RECALL/ALIGNMENT, INTERNAL, A1
```

Code Used to Recall an External Alignment

```
RECALL/ALIGNMENT, EXTERNAL, FILE_NAME
```

`FILE_NAME`

This is the file name used for the saved external alignment, minus the .aln extension. If you type a non-existent filename, PC-DMIS will display a warning message letting you know that it could not find the alignment.

Example:

```
RECALL/ALIGNMENT, EXTERNAL, FIXTURE1
```

Using an Alignment Inside Loops or Branches

PC-DMIS facilitates changing alignments inside a measurement routine that uses looping or conditional branching by using the `USE_ACTIVE_ALIGNMENT` keyword following the `RECALL:` text within your `ALIGNMENT/START` command. This keyword lets you recall the active alignment.

If you have an alignment command inside a loop, and the loop uses offsets, you *must* define all axes for that alignment. Additionally, the alignment inside the loop must use features measured inside the loop.

The `USE_ACTIVE_ALIGNMENT` keyword also displays the name of the active alignment within parentheses. So, if the active alignment was A3 during the last execution, the keyword shows the following after execution:

```
USE_ACTIVE_ALIGNMENT(A3)
```

PC-DMIS uses the active alignment name for different purposes depending on whether PC-DMIS is in learn or in execute mode.

- *In learn mode*, changing the alignment name in parentheses does not have any bearing on the actual active alignment used or displayed during execution. In learn mode this name is merely used as a tool for you to visualize different scenarios that may occur during execution. You can visualize what may occur by changing the

active alignment in parentheses and watching the trihedron move to that alignment's coordinate system in the Graphic Display window.

- *In execute mode*, PC-DMIS determines which alignment is active based on the last executed alignment. This is dependent on any branching and looping that occurred during execution. After execution, the actual alignment which was active during the last execution is displayed in parentheses.

Also, the alignment name in parentheses is now saved with the measurement routine. When opening a measurement routine created prior to PC-DMIS 2010 MR2, the information that appears in parentheses is dynamically populated by PC-DMIS as it searched for alignments above the alignment with the `USE_ACTIVE_ALIGNMENT` setting. This happens regardless of commands that may affect measurement routine flow during execution, such as looping or branching commands.

Measurement Routines that have been imported from a DMIS file into PC-DMIS do not support the functionality of showing the alignment name in parentheses, even if you save the file as a new measurement routine.

USE_ACTIVE_ALIGNMENT Example

If you look at the `ALIGNMENT/START` command line in Command Mode, you'll see that the field immediately following the `RECALL:` text tells the measurement routine to use a stored starting alignment first. In the following example, alignment D_1 starts with the alignment from D_0 and then executes a 45 degree rotation about ZPLUS:

```
D_1=ALIGNMENT/START,RECALL:D_0, LIST= YES
ALIGNMENT/ROTATE_OFFSET,45.0,ABOUT,ZPLUS
ALIGNMENT/END
```

However, if you use the `USE_ACTIVE_ALIGNMENT` keyword instead, you can cause PC-DMIS to rotate 45 degrees from the active alignment:

```
D_1=ALIGNMENT/START,RECALL:
USE_ACTIVE_ALIGNMENT(D_0), LIST= YES
ALIGNMENT/ROTATE_OFFSET,45.0,ABOUT,ZPLUS
ALIGNMENT/END
```

When you use this keyword for an alignment inside a loop:

- The *first time* through the loop the active alignment will be the last executed alignment preceding the loop.
- Additional times through the loop the active alignment will be itself and will rotate a further 45 degrees from the previous time through the loop.

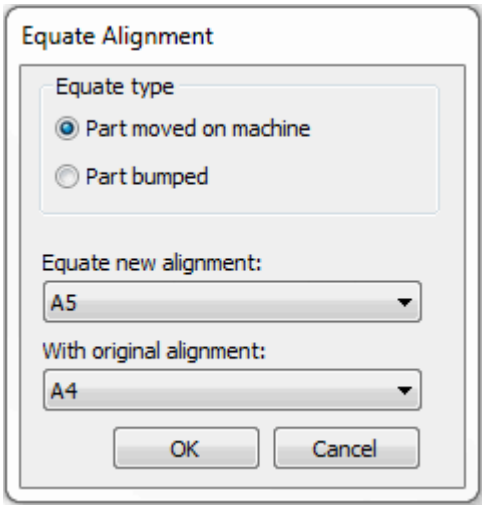
For information on looping, see the "Creating Generic Loops" topic in the "Branching by Using Flow Control" chapter.

Preferences that Affect USE_ACTIVE_ALIGNMENT

When you use the `USE_ACTIVE_ALIGNMENT` keyword, you should generally deselect the **Reset global settings when branching** check box and select the **Treat theo values as if stored in part coordinates** check box on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**).

See the "Setting Your Preferences" chapter for information on specifying setup preferences.

Equating an Alignment

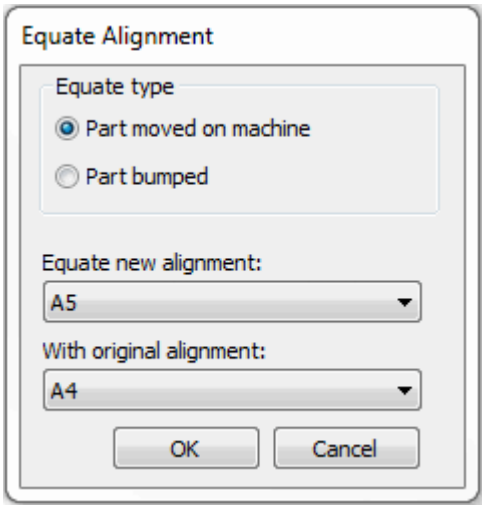


Equate Alignment Dialog Box

The **Insert | Alignment | Equate** option provides the means to:

- Change the position or orientation of a part while retaining previous dimensional information.
- Re-align the part and save previously measured data if the part is accidentally bumped or moved during the inspection process.

To Change a Part's Position and Orientation



Equate Alignment Dialog Box - Part moved on machine Option

For Equate Alignment to function properly:

- The features referenced in your new alignment must be measured after you move your part.
- Those features must reside in the measurement routine below the commands that are executed in the part's original position and orientation (including the original alignment).
- You should reference all of the features for your new alignment in a single alignment block.
- The new alignment should be fully constrained.

This command works with both regular alignment blocks (`START_ALIGN/ END_ALIGN` pairs) and external recalled alignments. The external alignment must be recalled using the `RECALL/ALIGNMENT, EXTERNAL` command before it becomes available for use in the dialog box.

Equate new alignment:

This list allows you to select the *new alignment* you are equating with your original created alignment.

With original alignment

This list allows you to select the *previously created original alignment* to which you are equating the new alignment.

For example, to measure a dimension that references features on two sides of the part that are not accessible from a single part orientation:

1. Measure the alignment features on the first side of the part.

2. Create the original (fully-constrained) alignment.
3. Measure all required features that can be reached from the first orientation of the part.
4. Move the part to its new position.
5. Switch to Manual Mode.
6. Recall the startup alignment (Machine Coordinate System).
7. From CAD or the print values, program the new alignment features in Manual Mode and in DCC Mode.

   > The origin must be the same, and the axis must be in the same direction as the axis of the alignment to which you are equating. It is easiest to understand this if you imagine that the original origin and axes' arrows are glued onto the part before moving it. The new alignment places the origin and axes' arrows in the same position with respect to the part.

8. Select the **Insert | Alignment | Equate** menu option. The **Equate Alignment** dialog box appears.
9. In the **Equate Type** area, select **Part moved on machine**. Note that when editing an existing Equate alignment command, the **Equate Type** area is disabled.
10. In the **Equate new alignment** list, select the *new alignment*.
11. In the **With original alignment** list, select the *original alignment*.
12. Click the **OK** button to insert the new `EQUATE` alignment command into the measurement routine. The CAD model won't move relative to the alignment axes, but the measured values will move once PC-DMIS executes the equate alignment.

The Edit window command line for this option reads:

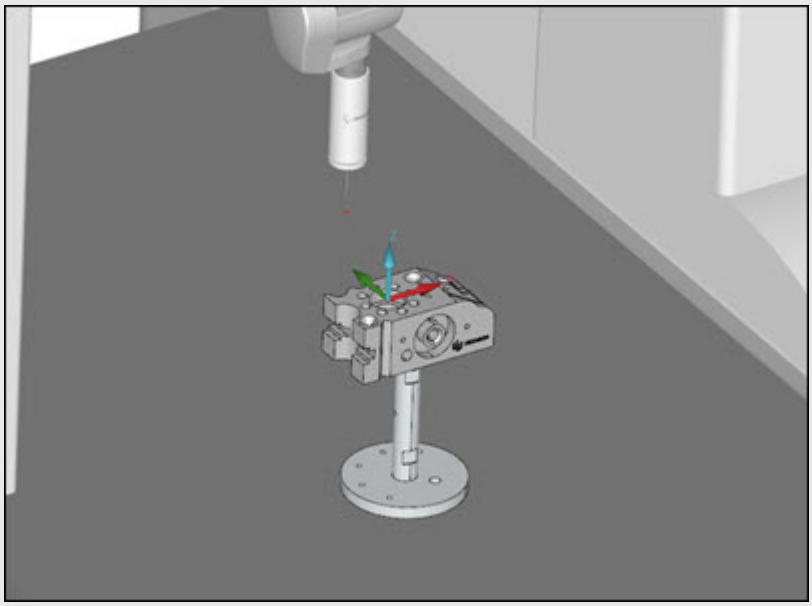
```
EQUATE/"name1"TO ALIGNMENT,"name2"
```

Note about Clearance Planes

After equating the new alignment, any existing clearance planes continue to use the same "relative" plane from the previous alignment. This means after you have moved your part, you should define new clearance planes to avoid incorrect clearance plane moves.

Example:

In this setup, we want to measure all features in the X-, X+, Y-, Y+, and Z+ planes. The features in the Z- direction are not accessible which requires a second setup.

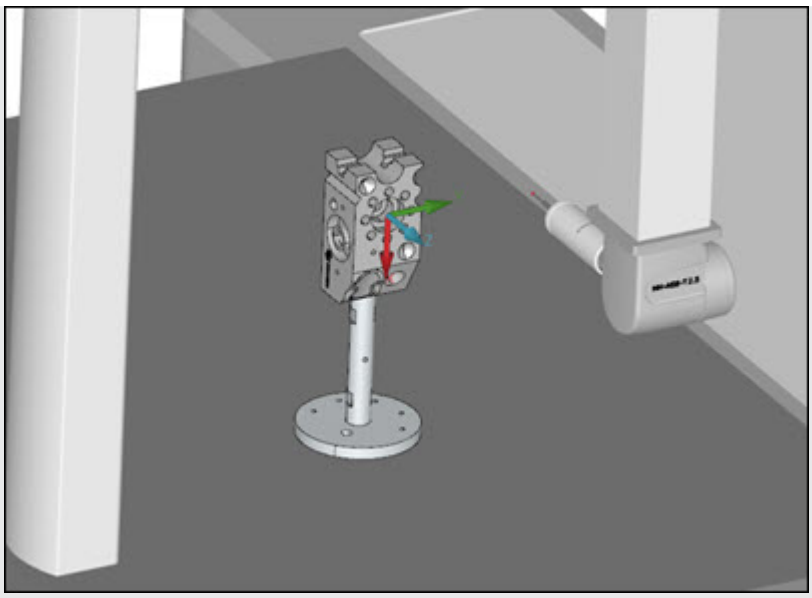


Example setup #1

1. To begin, create the measurement routine with a manual and DCC alignment, and then measure all the accessible features.
2. When you need to relocate the part, make note of the coordinate system and the orientation of the axes. You must replicate this in the second setup.

   Before you switch to the second setup, remember to switch to Manual Mode and recall the startup alignment (step 5 above).

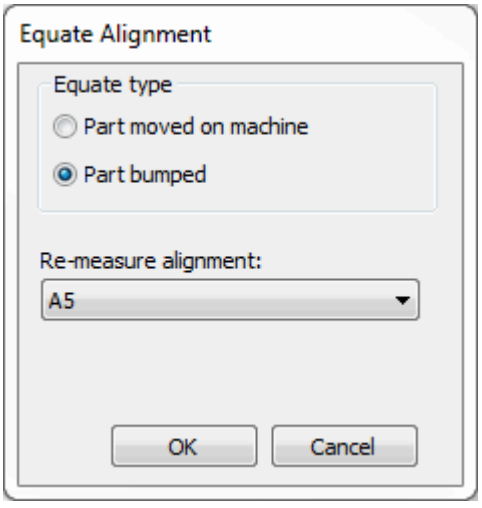
3. Add your manual and DCC alignments. For the best accuracy, take the measurements in the same location as in the first setup.
4. Measure all the features in the Z- plane.



Example setup #2

5. To relate the Z- features with the other planes, equate the alignments. For details on how to equate an alignment, see "Equating an Alignment" in the PC-DMIS Core documentation.

To Recover after Accidentally Moving a Part



Equate Alignment Dialog Box - Part bumped Option

For this operation to function properly, the alignment should be fully constrained.

This command only works with regular alignment blocks (`START_ALIGN/ END_ALIGN` pairs).

Re-measure alignment

This list allows you to select the *existing alignment* to remeasure to update all the machine coordinates of the commands in the measurement routine.

If a part has been moved accidentally, do the following:

1. Select the **Insert | Alignment | Equate** menu option to open the **Equate Alignment** dialog box.
2. In the **Equate Type** area, select **Part bumped**.
3. In the **Re-measure alignment** list, select the ID of the alignment you need to remeasure.
4. Click **OK** to begin the remeasure process.
5. Re-measure the alignment features. When this finishes, all dimensional and feature information are translated to the new position of the part. The CAD model does not move relative to the alignment axes, but the measured values are updated.

If you use this option, PC-DMIS does not insert a new command in the Edit window.

Equating CAD to Measured Part Data

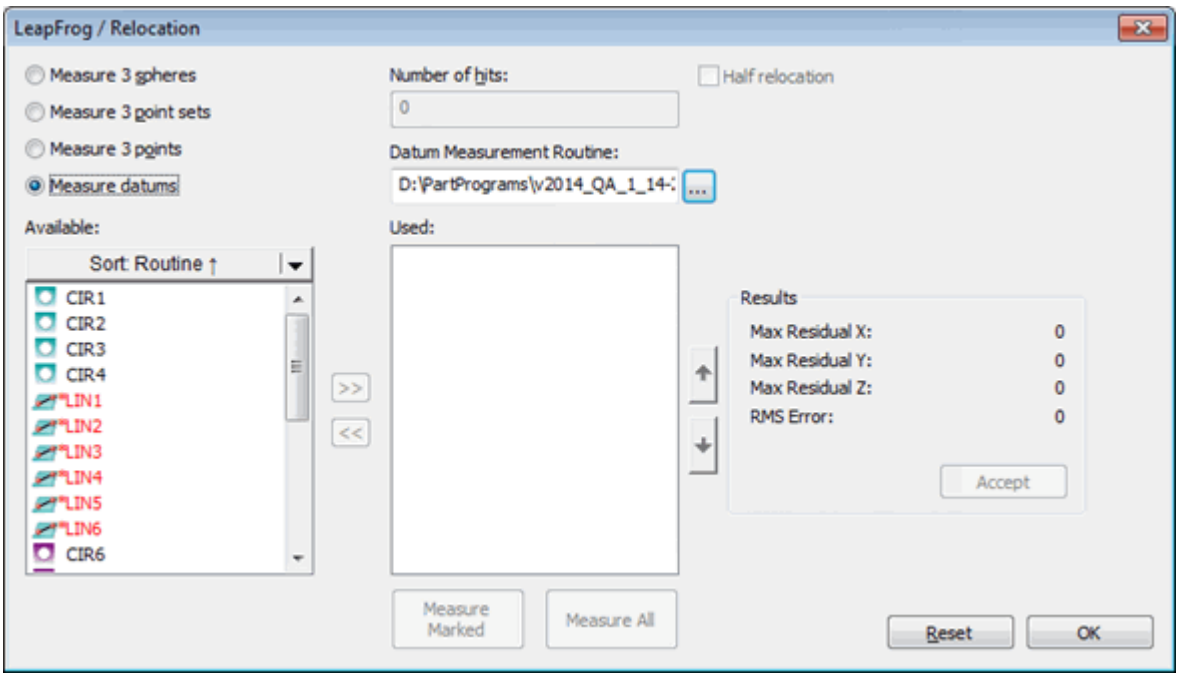
The **Operation | Graphic Display window | CAD Equals Part** menu option (or the **CAD = Part** button in the **Alignment Utilities** dialog box) links CAD data to the measured data. This option is only available after a created alignment places the part origin/orientation at the same location as the CAD origin/orientation. PC-DMIS offers the CAD EQUALS PART option in two areas (also see "CAD Equals Part" within the **Alignment** option). Select this option and PC-DMIS will display the measured data on top of the CAD data. It will use the CAD data to help inspect the part.

Once the CAD Equals Part option has been used on a measurement routine, the CAD Equals Part menu option will be selected.

Performing a Leapfrog Operation

PC-DMIS does not support Leapfrog and Bundle Alignment commands used in the same measurement routine.

Select **Insert | Alignment | Leapfrog** to display the **LeapFrog / Relocation** dialog box. This menu option is disabled in Offline mode. You must be connected to a physical and supported portable machine.



LeapFrog / Relocation Dialog Box

The **LeapFrog / Relocation** dialog box allows you to move supported portable machines in order to measure parts that are too large for your CMM. You should be aware of machine accuracy limitations before using this method.

Currently, supported machines include Faro, Romer, and Garda.

Your PC-DMIS license needs to be programmed to support your portable machine.

The basis for Leapfrog is to measure a series of features, and then after moving the machine, re-measure the same features in the same order. This creates a transformation and makes the machine behave as if it were in the same coordinate system before the move.

Prior to PC-DMIS version 4.2, Leapfrog transformation information was stored in a separate file and was thereby independent of all measurement routines. This meant the Leapfrog was still active in newly-created measurement routines and you had to remove it by clicking the **Reset** button on the **Leapfrog / Relocation** dialog box. In version 4.2 and later, however, this has changed. Leapfrog transformation information is now stored with the measurement routine that used the Leapfrog operation; you no longer have to remove the Leapfrog from new measurement routines.

PC-DMIS enters a Leapfrog command into the Edit window when you click the **Accept** button.

The command line in the Edit window is:

```
LEAPFROG/TOG1, NUM, TOG2
```

TOG1: This first parameter in the Leapfrog command is a toggle field that relates to the different types of measurement options that are available in the dialog box. These include:

1. SPHERES (**Measure 3 Spheres** option)
2. PSETS (**Measure 3 Point Sets** option)
3. POINTS (**Measure 3 Points** option)
4. DATUMS (**Measure Datums** option)

There is also an OFF value for this parameter, in which case the other two parameters do not appear. The OFF value turns off Leapfrog translation.

NUM: This second parameter in the Leapfrog command is the number of hits you want to take. This corresponds to the **Hits** box in the **LeapFrog / Relocation** dialog box.

TOG2: This last parameter in the Leapfrog command is a toggle field that you can use to switch between either a FULL or a PARTIAL Leapfrog. This parameter corresponds to the **Half relocation** option in the dialog box.

When this command executes, PC-DMIS prompts you to take your hits. After all the hits are taken, a Leapfrog translation is in effect.

Measure Options

The **Measure** options in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) enable you to select the method that PC-DMIS uses to perform the translation comparison.

- The **Measure 3 Spheres** option tells PC-DMIS to use spheres as the features for translation comparison. This method uses the center of each measured sphere.
- The **Measure 3 Point Sets** option tells PC-DMIS to use the centroid of a set of points. It is recommended that you use the bottom of an inverted cone with a hard probe. This method is slightly more accurate than the spheres method and much quicker for the operator. For information on how to create point sets, see the "Creating Measured Feature Sets" topic in the "Creating Measured Features" chapter.
- The **Measure 3 Points** option tells PC-DMIS to use only three points and is the least accurate of the three methods.
- The **Measure Datums** option tells PC-DMIS to use existing datum features from a measurement routine of your choice. Because the datum features are already assumed to have been measured in your existing measurement routine, you only need to measure them after relocating your machine.

Number of Hits

The **Number of Hits** box in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) enables you to specify the number of hits you wish to use for each feature. Of course, this will not be used in the point method.

Half Relocation

The **Half Relocation** check box in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) lets you determine whether or not PC-DMIS performs a FULL RELOCATION (FULL LEAPFROG) operation (if not selected) or performs a PARTIAL RELOCATION (PARTIAL LEAPFROG) operation (if selected).

Relocation refers to moving the portable measuring machine to a new location.

- Doing a full relocation (clearing this check box) means you would need to measure something before you move the portable machine and then re-measure some or all of those items after moving the machine. Remeasuring allows PC-DMIS to determine the new location of the machine.
- A half relocation (selecting this check box) means you move the portable machine first and then measure the datum features.

Datum Measurement Routine

This area in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) enables you to specify the measurement routine file to use as the datum program file. This box becomes enabled when you click the **Measure Datum Features** option. You can type the full pathway to the measurement routine (.PRG) file, or you can use the **Browse** button to navigate through your directory structure and select one that way.

Once you select a file, the features available for use in the Leapfrog operation appear in the **Available** list.

Available and Used Lists

The **Available** and **Used** lists in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) display, respectively, datum features that are available for use or datum features that you have chosen to use in the Leapfrog operation.

Available List

When you select a measurement routine file to use in the **Datum Program File** area, the available features from that file appear in the sortable **Available** list. You can then assign features to the current Leapfrog operation by selecting them and then clicking the **>>>** button. More information on the Feature list section can be found in the

Used List

Assigned features that appear in the **Used** list will be measured when you click the **Measure Marked** or **Measure All** buttons in the order that they appear in the **Used** list. You can remove them from the **Used** list by clicking the **<<<** button. You can change a feature's order of execution by selecting a feature clicking the up or down arrow buttons.

Measure Marked

The **Measure Marked** button in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) only functions when you use the **Measure datums** option. It allows you to measure any selected features from the **Used** list. PC-DMIS will use these features in the Leapfrog operation. When you click this button, PC-DMIS displays the **Execution** dialog box, prompting you to measure the selected features once you have moved the CMM, not before.

The results box will display the 3D distance between the features taken before the move and after the move of the CMM. If you find the results unsatisfactory then you may re-measure the last set of features again as the button will now read **Re-measure**.

After you move the CMM, there is not a way to revert to the previous alignment. If the Leapfrog results are unsatisfactory after the **Re-measure** process, you must reset the Leapfrog and restart the entire part inspection process by executing the measurement routine from the initial alignment. The physical limitation of using a single CMM device makes this condition possible for all relocation methods. *You should take great care during the execution of any relocation procedure.*

Measure All

Similar to Measure Marked, the **Measure All** button in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) also opens the **Execution** dialog box.

- If you are using **Measure 3 Spheres**, **Measure 3 Point Sets**, or **Measure 3 Points**, then this dialog box will first prompt you to measure the three features before prompting you to move the CMM. After moving the machine, you will be prompted to re-measure the same features in the same order.
- If you are using **Measure Datums**, then the **Execution** dialog box prompts you to measure all the datum features once you have moved the CMM, not before.

The results box will display the 3D distance between the features taken before the move and after the move of the CMM. If you find the results unsatisfactory then you may re-measure the last set of features again as the button will now read **Re-measure**.

If the re-measure process proves unsatisfactory, then you must reset the leapfrog and start again from the beginning. This is a problem with all leapfrog systems and should be remembered.

Results area

The **Results** area in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) shows the deviations between the machine's first position and its subsequent position(s) by displaying the 3D distance between the features taken before the move and after the move of the CMM.

Accept

Once you have a filled out the **Leapfrog / Relocation** dialog box, you must click the **Accept** button from the **Results** area before PC-DMIS uses the leapfrog transformation. Click **Accept** to add the `LEAPFROG` command to the measurement routine. If you don't click the **Accept** button but click the X in the upper-right corner or click **OK** first, the constructed leapfrog translation is lost.

Reset

The **Reset** button in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) removes any translation by adding a `LEAPFROG/OFF` command into the Edit window.

OK

Clicking **OK** in the **LeapFrog / Relocation** dialog box (**Insert | Alignment | Leapfrog**) closes the dialog box. If you click this button before you click the **Accept** button, the dialog box will close without inserting the LEAPFROG command.

Changing Alignment Nominal Values

If you modify an alignment feature's theoretical values during execute mode, PC-DMIS changes the CAD to Parts alignment. This means that features in your measurement routine that come after the alignment, and are measured relative to the alignment features, shift by the amount of the changed theoretical values.

If you select the **Ignore CAD to Part** check box on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**), the CAD to Parts alignment does not change when your alignment feature theoretical values change. PC-DMIS measures the features

below the alignment in their same position. See "Ignore CAD to Part" in the "Setting Your Preferences" chapter.

You cannot use the **Ignore CAD to Part** functionality if you use Position dimensions in your measurement routine.

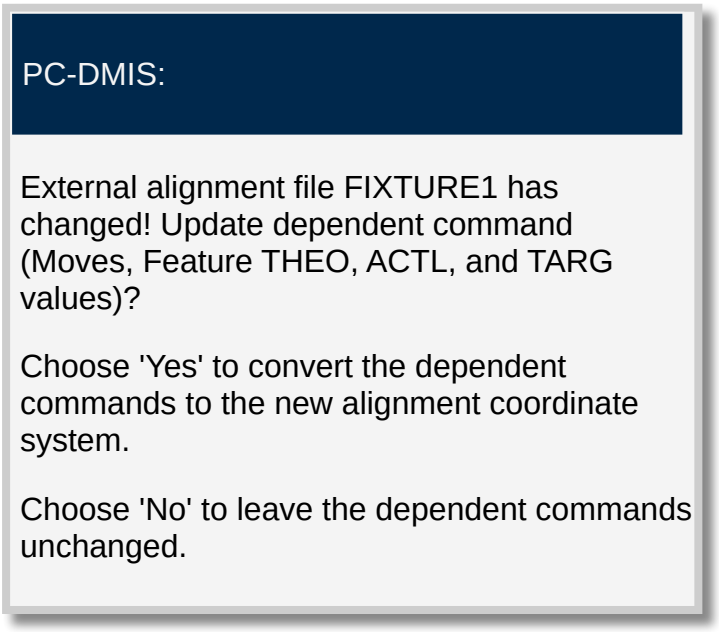
You can also control how PC-DMIS handles your features' nominal values when you update an alignment's theoretical values with the `UpdateBelowChangedAlignmentDuringExecution` entry in the PC-DMIS Settings Editor. For additional information, see the "Modifying Setting Entries" appendix.

Updating Dependent Commands When the Alignment Changes

An alignment consists of two transforms:

1. Machine origin to Part origin (termed "Machine-to-Part")
2. CAD origin to Part origin (termed "CAD-to-Part")

If you modify either of the transforms, PC-DMIS displays a Yes/No message similar to the following, letting you choose how the commands dependent on the alignment are updated:



You can choose to convert the dependent commands to the new alignment coordinate system or to leave the dependent commands unchanged.

The text in the message indicates what types of commands and command values the software updates for each alignment change.

Different commands and command values may change depending on the transform change. This table lists the commands and command values that are updated after a transform change:

	Transform Change		
	Machine-to-Part	**Cad-to-Part**	**Both**
Commands and command values that may update after a changed transform:	• Move commands • Feature command ACTL	• Feature command THEO • Feature command TARG	• Move commands • Feature command THEO • Feature command ACTL • Feature command TARG

In the following two general scenarios, you need to decide how to update the commands in the measurement routine in response to an alignment change:

1. When you load a measurement routine that recalls an external alignment file. See "Updating Commands When Loading a Measurement Routine " below.
2. When you add a new alignment or modify or delete an existing alignment while in Learn mode. See "Updating Commands While in Learn Mode" below.

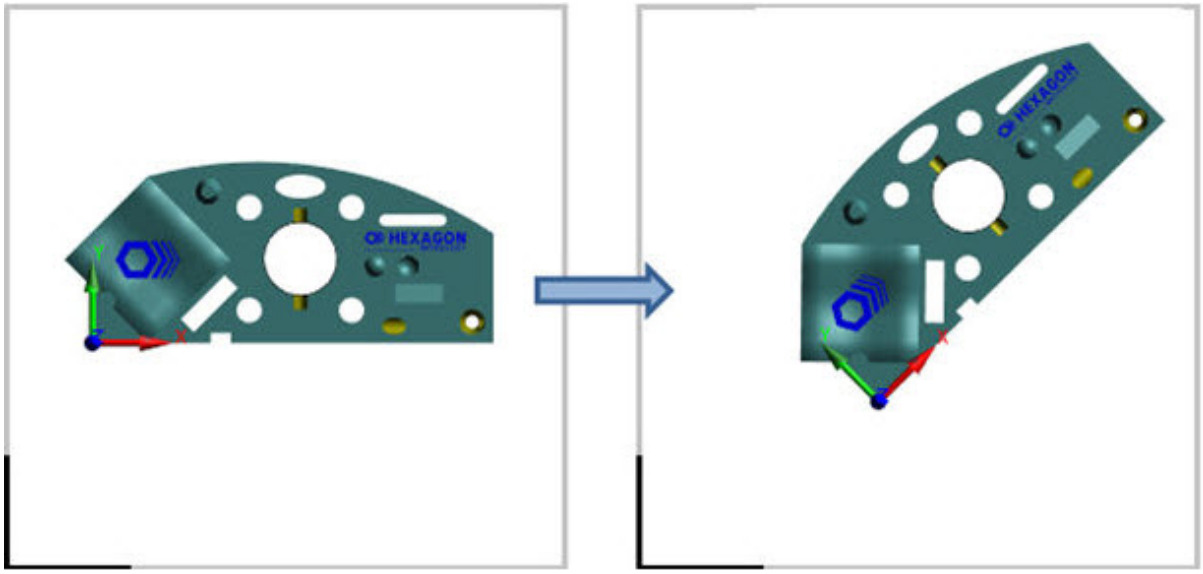
Updating Commands When Loading a Measurement Routine

A measurement routine that includes `RECALL/ALIGNMENT, EXTERNAL` commands attempts to re-load the referenced external alignment files each time the measurement routine opens.

The correct answer to the question "Update dependent commands?" depends on *why* the external alignment file has changed. Consider these reasons:

- **The Alignment file changed because you moved the part on the CMM** - In this case, the theoretical and actual values of the commands dependent on that alignment need to remain fixed, relative to the part (see the figure below). The Machine-to-Part transform is modified. It's possible that the Cad-to-Part transform is also modified, but that is not very common. Choose **No** at the Yes/No message so

the values of the dependent commands do not change. The position and orientation of the part can then change while retaining the previous dimensional information, similar to equating an alignment. *This is the most common scenario*.



Example of part movement on the CMM.

- **The Alignment file changed because you made small adjustments to the alignment to see how it affects the measured results, without actually re-measuring the part** - In this case, the values of the dependent commands must be updated to the new alignment coordinate system. The Machine-to-Part transform is modified, but the Cad-to-Part transform has not changed.

  This usually requires the `SAVE/ALIGNMENT` command be configured to only include the Machine-to-Part transform, not both transforms.

  Choose **Yes** at the Yes/No message.

- **The Alignment file changed because you moved the alignment origin to a new location on the part, but you did not move the part itself** - For example, you changed the level, rotate, and/or locate features to different features. In this case, theoretical and actual values of the commands dependent on that alignment must be updated to the new alignment coordinate system. The Cad-to-Part transform has been modified, but the Machine-to-Part transform has not changed.

  This requires the `SAVE/ALIGNMENT` command to be configured to include both transforms.

  Choose **Yes** at the Yes/No message so that the measured data does not shift away from the CAD geometry. *This is not a common scenario.*

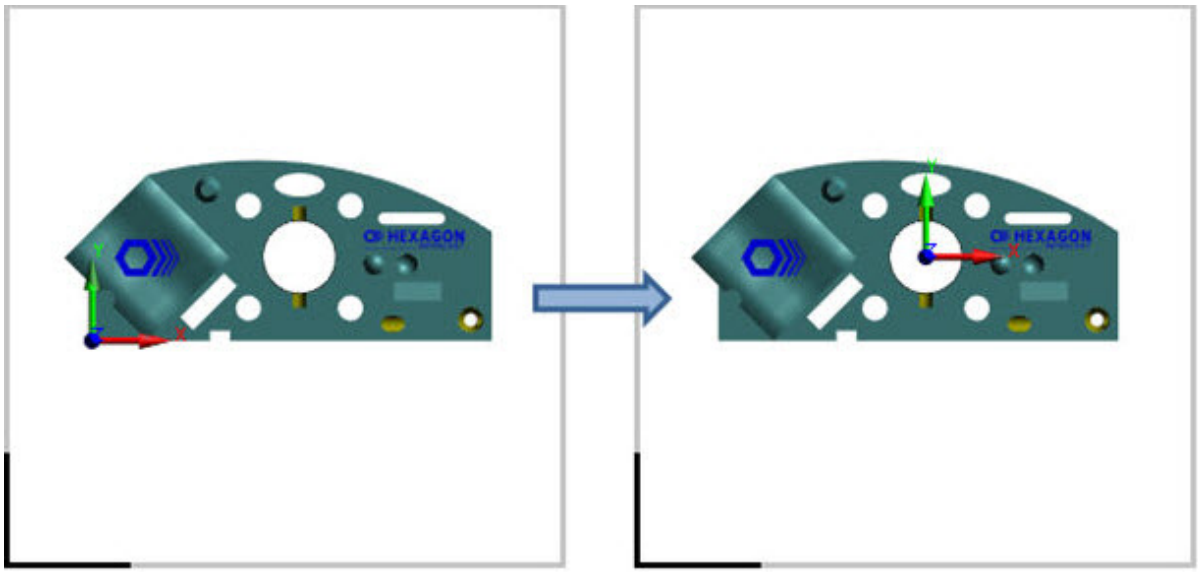
When loading a measurement routine that recalls an external alignment, if the external alignment file transforms have been modified since the measurement routine was last saved, by default, PC-DMIS does not display the dialog box, asking to update the dependent commands. Instead, it automatically answers No to that question. You can control this behavior with the `UpdateExtAlignmentDepCommandsDuringFileOpen` entry. For information on how to modify values for entries, see "Modifying Setting Entries".

Updating Commands While in Learn Mode

When you edit a measurement routine in Learn mode, if changes are made that modify or delete an existing alignment command or create a new one, PC-DMIS asks if you want to update the commands dependent on that alignment.

The correct answer to the question "Update dependent commands?" depends on why the alignment definition has changed. Consider these reasons:

- **The alignment definition changed because you moved the alignment command origin to a new location on the part** - For example, you changed the level, rotate, and/or locate features to different features. In this case, the theoretical and actual values of the commands dependent on that alignment command need to be updated to the new alignment coordinate system (see the figure below). Both transforms have been modified.



Example showing the alignment movement on the part.

Choose **Yes** at the Yes/No message so that the measured data does not shift away from the CAD geometry. *This is a common scenario.*

- **The alignment is from a command that recalls an external alignment (RECALL/ALIGNMENT, EXTERNAL), and you changed the file reference to a different external alignment file that represents the part at a different location on the CMM** - In this case, the theoretical and actual values of the commands dependent on the alignment remain fixed relative to the part. The Machine-to-Part transform has been modified, but the Cad-to-Part transform needs to remain the same.

Choose **No** at the Yes/No message so that the values of the dependent commands are not changed. *This is a less-common scenario*. It allows the position and orientation of the part to change while retaining the previous dimensional information, similar to equating an alignment.

Using Gages

This section of the documentation provides information on the PC-DMIS gages.

PC-DMIS gages are quick-check tools designed to measure various system and feature characteristics.

2D Radius Gage

The 2D Radius Gage function is a quick-check tool that you can use to measure the radii on a pointcloud or mesh cross section.

You can create a 2D Radius Gage graphically on a cross section in the 2D Slide Show view.

For details, see "2D Radius Gage Overview" in the PC-DMIS Laser documentation.

Caliper Gage

The Caliper is a quick-check tool that works similar to a physical caliper. It provides a local two-point size check on the Pointcloud (COP), Mesh, or COPOPER (such as the COPSELECT, COPCLEAN or COPFILTER) object. The Caliper shows the measured length along the selected axis or direction.

For details, see "Caliper Overview" in the PC-DMIS Laser documentation.

Temperature Gage

For the temperature gage, see "Temperature Gage" under Setting Your Preferences.

Thickness Gage

The Thickness Gage command calculates and reports a 2D thickness and gage position.

The Thickness Gage command requires a primary and a secondary feature since PC-DMIS determines the thickness by piercing the secondary feature normal to a primary feature gage point. When you create a Thickness Gage command, the software only shows the valid features to select from in the **Thickness Gage** dialog box.

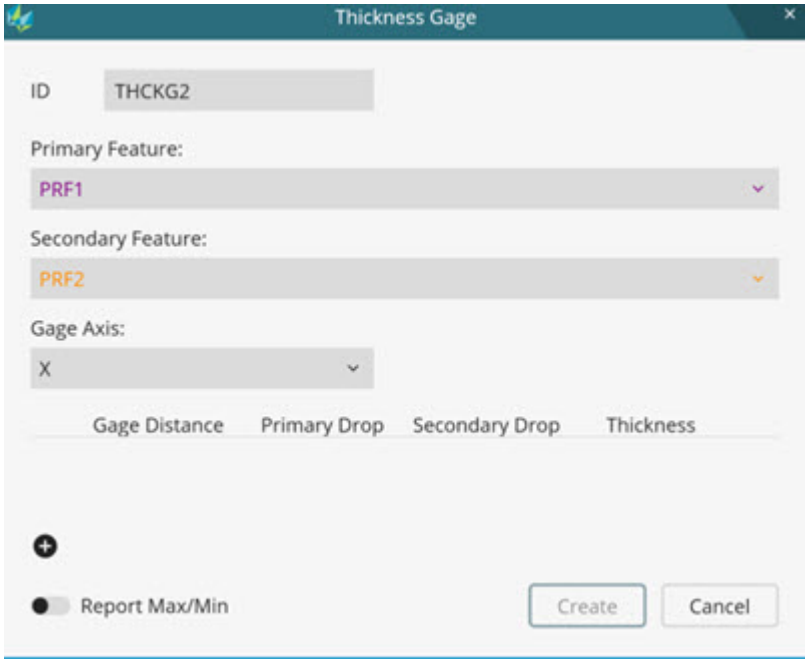
The Thickness Gage command only supports these feature types:

- Thickness Scan
- Linear Scan
- Profile 2D

You can use the Linear scan and the Profile 2D in any combination. However, you can only use the Thickness scan by itself since it contains data for two surfaces.

To create a Thickness Gage command:

1. Create your primary and secondary features.
2. Select **Insert | Gage | Thickness** to open the **Thickness Gage** dialog box.



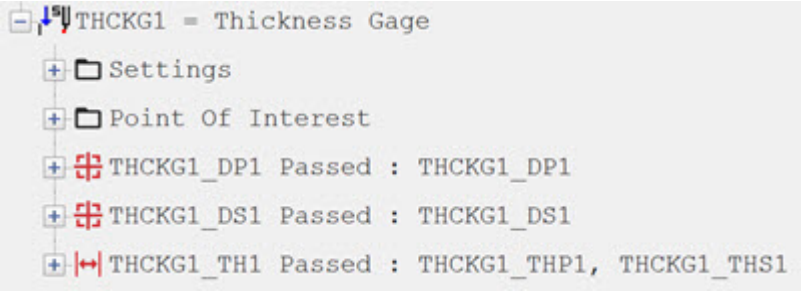
Thickness Gage Dialog Box

You can also access the **Thickness Gage** dialog box from the **QuickMeasure** toolbar (**View | Toolbars | QuickMeasure**). Click the **Gage** drop-down arrow, and then click the **Thickness Gage** button.

3. Select the primary and secondary features from the respective **Primary Feature** and **Secondary Feature** lists. The software populates the list items based on the surface or cut vector of the current alignment.
4. Select the **Gage Axis** and define your Point of Interest (POI). For details, see "Point of Interest (POI) Definition and Reporting".

5. To add the Thickness Gage to your report, click the **Report Max/Min** button.
6. Click **Create** to create the Thickness Gage.

Once you create the Thickness Gage command, the software adds the selected dimensions for the command in the Edit window:



Example of the Thickness Gage command in the Edit window

PC-DMIS considers the dimensions created by the Thickness Gage command to be internal dimensions to the command. Because of this, you cannot delete or cut and paste them in the Edit window.

You can press the F9 key while your cursor is over the internal dimension to display the **Thickness Gage** dialog box.

PC-DMIS considers the features generated by the Thickness Gage command to be internal commands. Because of this, you can use them in Alignments, Constructions, Dimensions and Assignments.

Point of Interest (POI) Definition and Reporting

Once you select the Gage Axis, you can define the Point of Interest (POI).

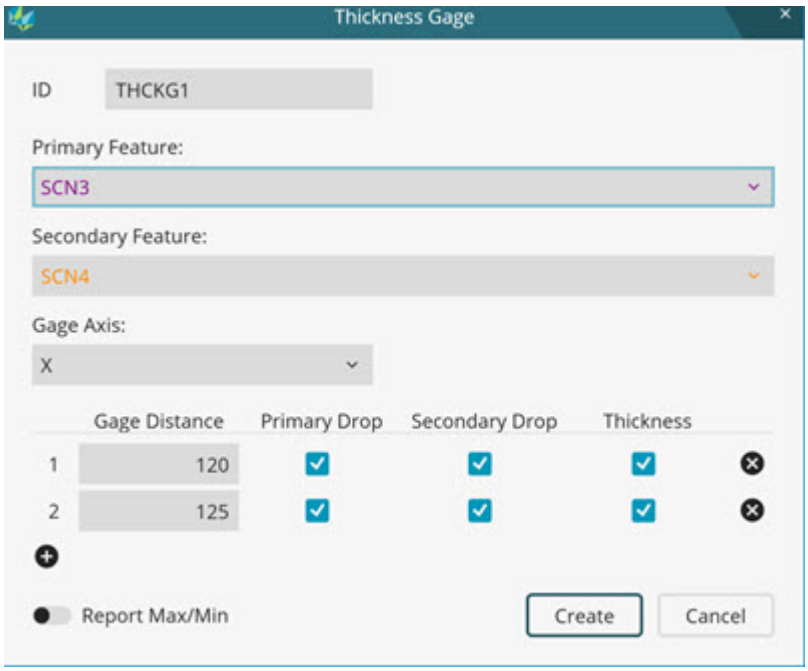
There are two methods to define the POI.

Method 1:

In the **Thickness Gage** dialog box, from the CAD View in the Graphic Display window, click the primary feature. The software completes the Gage Distance value based on the mouse-click position on the CAD.

Method 2:

In the **Thickness Gage** dialog box, click the **Add** button and then enter a value.



Once you define a POI, PC-DMIS creates these commands:

THCKG1_DP1 (Drop Point Primary) - This is the nominal pierce value and the actual primary curve value at the Gage Distance.

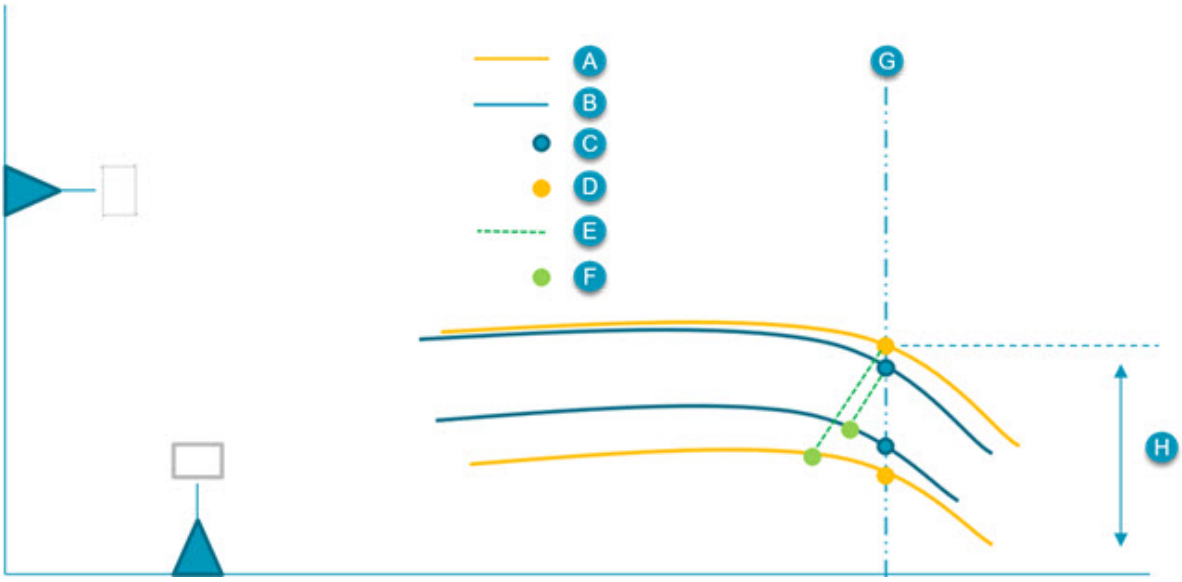
THCKG1_DS1 (Drop Point Secondary) - This is the nominal pierce value and actual secondary curve value at the Gage Distance.

THCKG1_THP1 (Thickness Point of THCKG1_DP1) - This is a duplicate point of THCKG1_DP1.

THCKG1_THS1 (Thickness Point Secondary) - This is the nominal pierce value and actual secondary curve value normal to THCKG1_THP1.

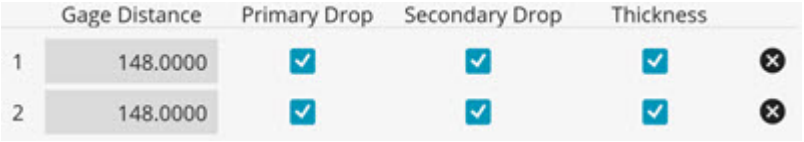
To report the thickness and gage position, the Thickness Gage command uses a Gage Axis method:

- If the input features are in the XY plane (Z is constant), the Gage Axis option is XY.
- If the input features are in the YZ plane (X is constant), the Gage Axis option is YZ.
- If the input features are in the XZ plane (Y is constant), the Gage Axis option is XZ.

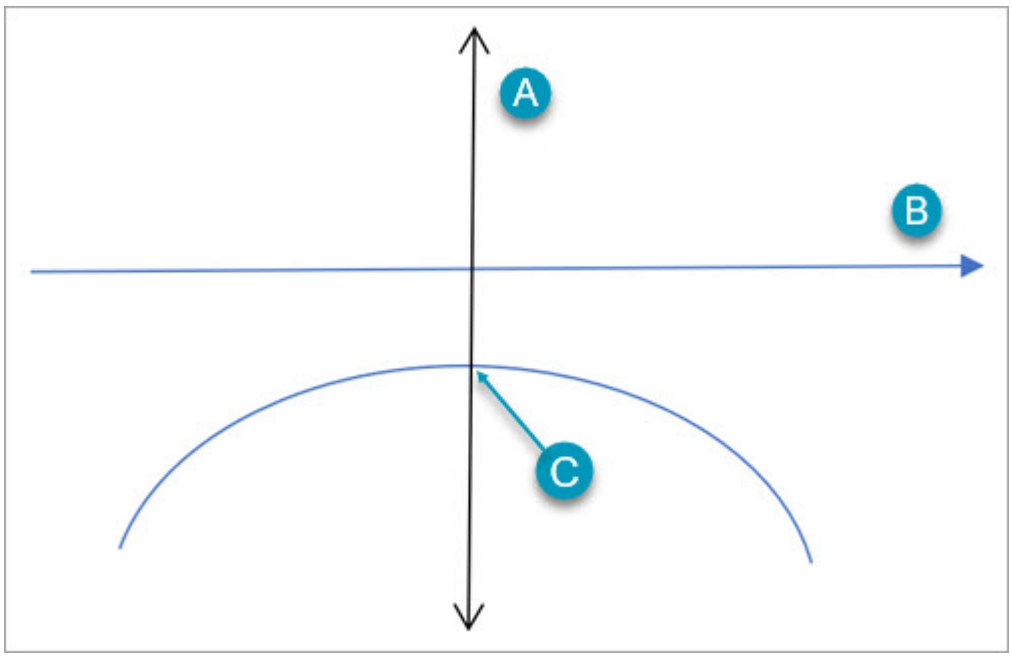


A - Actual
B - Nominal
C - Nominal Drop Point XYZIJK
D - Actual Drop Point XYZIJK
E - Normal to Top Curve
F - Thickness Point XYZIJK
G - Gage Distance
H - Drop

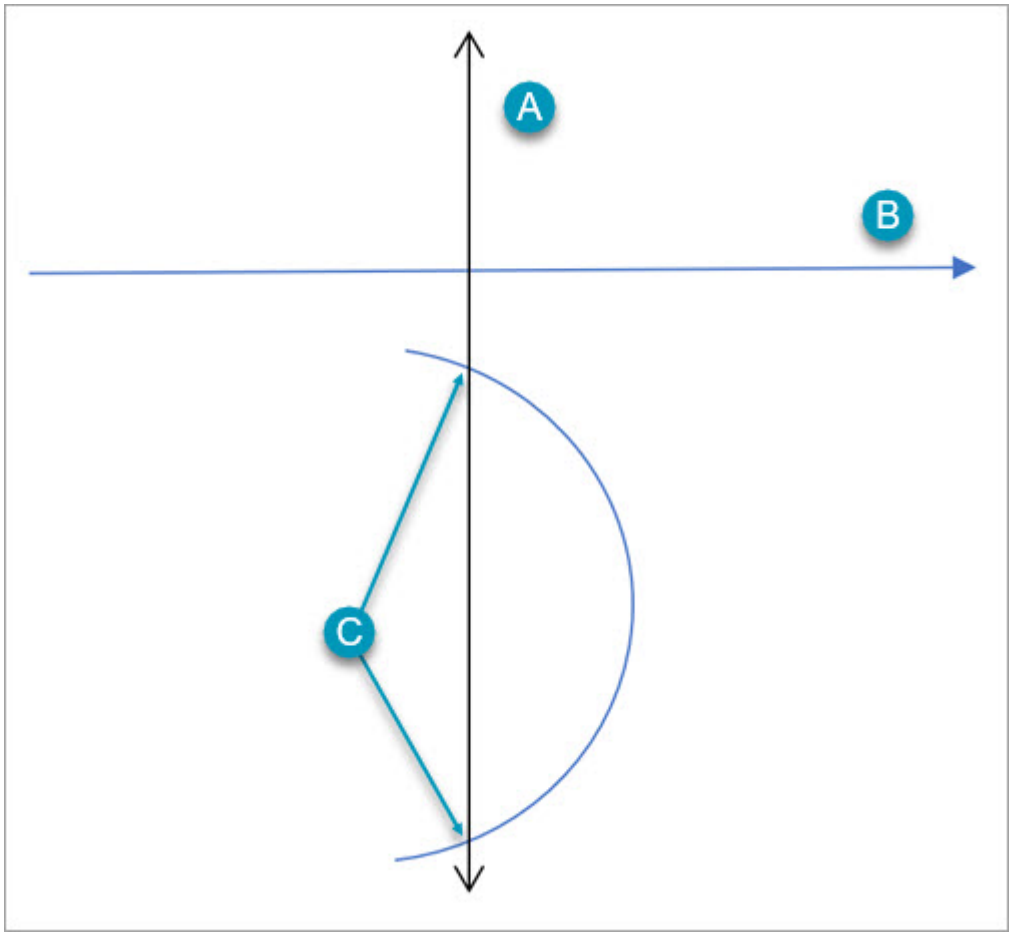
You can determine which dimensions to report with the **Primary Drop**, **Secondary Drop**, and **Thickness** check boxes.



You must use simple curves to create a POI. A simple curve is a curve that has only one intersection along any line normal to the gage axis as shown here:



A - Vector normal to the gage axis
B - Gage Axis
C - Single intersection point along the line normal to the gage axis
Example of a simple curve



A - Vector normal to the gage axis
B - Gage Axis
C - Multiple intersection points along the line normal to the gage axis

Example of a non-simple curve

When you manually enter the gage distance value or click the CAD to create a POI, if the primary or secondary curve is not simple, PC-DMIS displays an error message and does not create a POI.

PC-DMIS

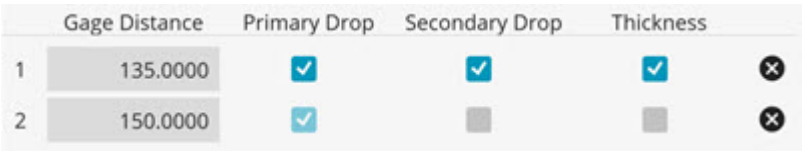
The features and gage axis represents an unsupported shape. Features with curves which have multiple intersections along any line normal to the gage axis are not supported.

The primary curve nominal data must also be smooth relative to the thickness. When you enter the **Gage Distance** value, or click the CAD to create a POI, if the primary curve is not smooth enough, PC-DMIS displays an error message and does not create a POI.

PC-DMIS

The nominals are not smooth enough to calculate thickness curve.

If a Drop Point or Thickness Point cannot be found on the secondary curve, the software disables the relevant options in the **Thickness Gage** dialog box.



If PC-DMIS cannot detect a POI, it displays this message:

PC-DMIS

No point of interest found at this gage distance.

You can add the maximum and minimum thickness values between the primary and secondary features into your report.

Using Legacy Dimensions

Using Legacy Dimensions: Introduction

Once you measure or construct your features, you can generate the inspection report. The first step in preparing a report is to calculate dimensions according to specific requirements. While this chapter covers PC-DMIS's legacy dimensions, understanding the information here may help you to create the newer Geometric Tolerance dimensions discussed in the "Using Geometric Tolerances" chapter.

You can calculate dimensions immediately after you complete a measurement or later in the measurement routine. PC-DMIS displays the result of each dimensioning operation in the Report window.

See the "Printing Dimensions to the Inspection Report" section of the PC-DMIS Core documentation for general information on how to modify the Edit window report.

PC-DMIS also allows you to specify nominal values, modify the output format, and/or print out the results of the calculations.

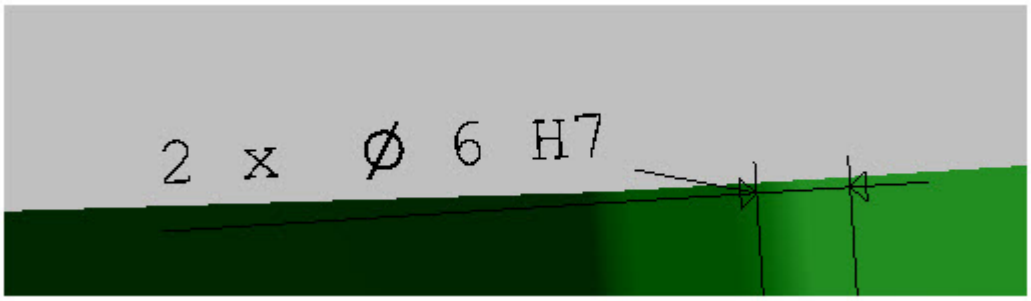
- To change the nominal or tolerance values displayed in the Edit window, click on the value to be changed and type in a new one.
- To show or hide nominals or tolerance fields in the Edit window, change the selected check boxes in the **Dimension Output Format** area of the **Dimension** tab in the **Parameters** dialog box (**Edit | Preferences | Parameters**). See the "Parameter Settings: Dimension tab" topic in the "Setting your Preferences" chapter.
- To change the measured values of a feature (for example, to get the radius of a circle to print out), see the "Modifying Report and Motion Parameters" topic in the "Setting your Preferences" chapter.

To dimension a feature, select the **Insert | Dimension** submenu from the menu bar, or access the **Dimension** toolbar and then select the desired dimension. This chapter covers the legacy dimensions found in the **Dimension** submenu.

You can toggle between using legacy dimensions and the newer Geometric Tolerance commands from the **Insert | Dimension | Use Legacy Dimensions** menu.

ISO Dimension Tolerances

ISO tolerances have an alphanumeric label instead of normal plus and minus tolerance values. The labels appear as "H7", "h7", or "G9", for example, as shown below.



Example of ISO tolerances in a PC-DMIS graphic

The following is based on the ISO Standard 286-1, ISO system of limits and fits.

The letter ("H", "h", "g", etc.) indicates the tolerance class. It is case-sensitive: capital letters indicate holes or other internal shapes, while lower-case letters indicate stems or other external shapes.

The number indicates the grade of the tolerance.

For more detailed information on ISO dimension and tolerance standards, visit the ISO website.

Reporting Dimensions for Measured Points

PC-DMIS has a flag in the basic hit now that stores whether the probe has a shank vector. If it does not, then all three axes are displayed rather than just one. This value is also serialized because when a measurement routine is read in, global_active_probe is not valid and the flag using that mechanism cannot be checked or set.

Dimension Command Format

All dimensioned features appear in the following format. There will be slight variations that are explained more thoroughly in the following sections.

PC-DMIS displays dimensions that aren't linked to any features with the color you have defined for out-of-tolerance text.

For example, the Edit window displays dimensions as:

```
 dimension_name = TYPE OF DIMENSION,feat_1 UNITS=IN,$
GRAPH=OFF TEXT=OFF MULT=1.00 OUTPUT=BOTH
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
x	8.0000	0.1000	0.1000	8.0000	8.5000	7.5000	0.0000	0.0000
y	3.0000	0.1000	0.1000	3.0000	3.5000	2.5000	0.0000	0.0000
z	0.4947	0.1000	0.1000	0.4947	0.1428	0.8466	0.0000	0.0000
d	1.0000	0.1000	0.1000	1.0000	1.0000	1.0000	0.0000	0.0000
v	0.0000	0.5938	0.8046	-	-	-	-	-

```
END OF DIMENSION dimension_name (for LOCATION only)
```

See "Conventions" for field rules.

TYPE OF DIMENSION: denotes the dimension type. This field cannot be changed except by changing the name of the feature.

feat_1: name of feature to be dimensioned. This is a user editable field.

All nominal and tolerance values in a dimension are editable. To edit the nominal and tolerance values:

1. Move to the desired cell by pressing the TAB key, or double-click on the field using the left mouse button.
2. Type in the desired value.
3. Press the TAB key again or click outside of the dimension.

Press the ENTER key to add a line. (See "Dimensioning Location".)

Adding Lines

To create an additional line within the location dimension, do the following:

1. Place the cursor in the desired location.
2. Press the Enter key.
3. Type in the desired axis (X, Y, Z, D, R, A, V, T, L, S, H, M, PR, PA, PD).
4. Press the Tab or Enter key.

PC-DMIS then displays the values for the added axis. PC-DMIS adds the new line depending on where the cursor is located. If the cursor is in the middle of a command, a new line is created below the current line. If the cursor is placed at the beginning of a command line**,** PC-DMIS creates the new line above the current cursor position.

Deleting Lines

To delete a single axis within a location dimension, highlight the desired axis and then press the Backspace or Delete key. (See "Command Mode Keyboard Functions" in the "Using the Edit Window" chapter.)

Editing Lines

When the *tolerances* of a dimension are edited in the Edit window, a dialog box appears asking if you want to carry the changed tolerances forward to any of the same dimension types. If the response is Yes, PC-DMIS will search from that point forward in the measurement routine to copy the tolerances into any dimensions that were of the same type and had the same original tolerances.

When the *nominals* of a dimension are edited in the Edit window, a dialog box appears asking if you also want to carry the changed nominals back to the dimension’s feature. If the response is **Yes**, PC-DMIS will change the feature nominals to the new nominal values.

Conventions

The **Dimension** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**) enables you to define parameters for the displayed dimensions.

- All angles can be represented 1-5 places to the right of the decimal. The actual number displayed is determined by the parameters set in the **Dimension** tab.
- All lengths, distances (x_cord, y_cord, z_cord, diam, height, length, etc.), and vectors (i_vec, j_vec, k_vec) in the dimensions are represented 1-5 places to the right of the decimal. The actual nominal that will be displayed is determined by the parameters set in the **Dimension** tab.

For information on the **Dimension** tab, see the "Setup Options: Dimension tab" topic in the "Setting your Preferences" chapter.

Accessing the Dialog Box

To access the corresponding dimension dialog box from which an Edit window's dimension was created:

1. Make sure the Edit window is open (**View | Edit Window)**.

2. In the Edit window, click on the dimension.
3. Press F9 to show the dimension dialog box for that dimension.

With this dialog box, you can make any desired changes to the existing dimension. When you click the **Create** button, PC-DMIS then applies the changes to the command in the Edit window.

Printing Dimensions to the Inspection Report

You can easily generate dimensions inside the Edit window, which, in turn, will generate these dimensions inside your inspection report.

To Automatically Create Dimensions in the Edit Window

To have PC-DMIS automatically create dimensions in the Edit window:

1. Access the Edit window (**View | Edit Window**).
2. Place the cursor where you want the dimension information to appear.
3. Type the appropriate keyword.

   OR

1. Select **Edit | Preferences | Setup** to access the **Setup Options** dialog box.
2. Select the **Dimension** tab.
3. Select the **Auto Create Dimension** option.
4. Select any other options.
5. Click the **OK** button.

The next time you measure a feature an automatic dimension will be created and inserted into the Edit window.

To Modify the Dimension Prior to Printing

In many cases, you must modify the nominals, tolerances, or output format before you print the results.

To change the nominals or tolerances from the Edit window:

1. Access the Edit window and place it in Command Mode (**View | Edit Window**).

2. Click on the dimension in the Edit window.
3. Press the Tab key to move to the value you want to change.
4. Type a new value.

To edit the nominals or tolerances using the dimension's dialog box:

1. Click on the dimension in the Edit window.
2. Press F9 to access its dialog box.
3. Change the necessary values.
4. Click the **Create** button.

To change the contents of the dimension report, access the **Dimension** tab in the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**) (see "Parameter Settings: Dimension tab" in the "Setting your Preferences" chapter) and modify the dimension information that PC-DMIS should display.

The "Analysis Settings" section of each dimension's dialog box enables you to display the dimension printout in a format suitable for close examination.

Displaying Dimensions in RTF and PDF Files

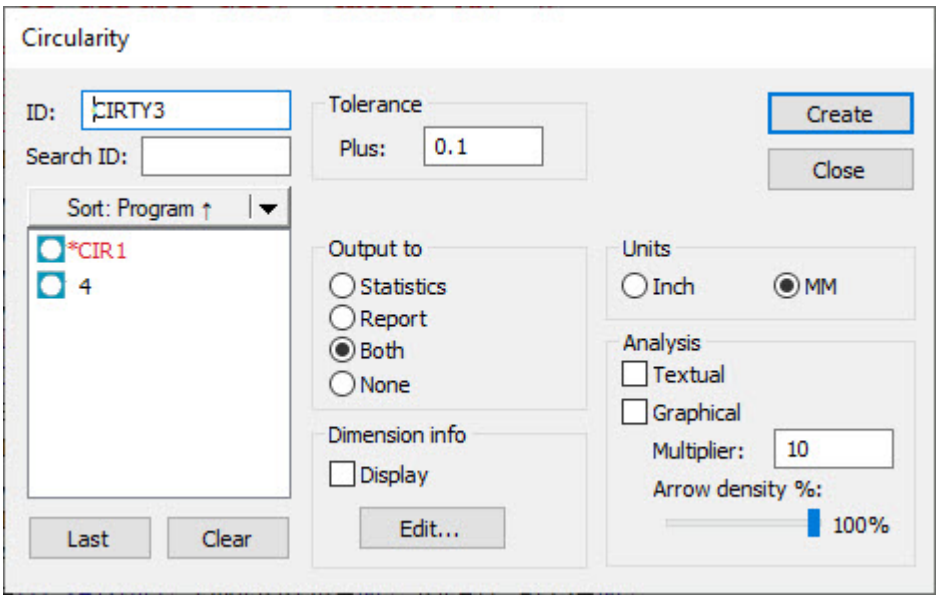
You can send the inspection report with its dimensions to an external RTF (Rich Text Format) or PDF (Portable Document Format) file by selecting the output options in the **Output Configuration** dialog box (**File | Printing | Report Window Print**) for the Edit window. See "Printing from the Inspection Report" in the "Using Basic File Options" chapter.

Be aware that because of formatting limitations between the RTF and PDF outputs, dimensions are displayed somewhat differently in the two files.

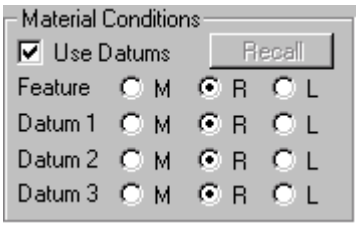
- *In RTF files*, dimensions are printed with a light border around the header, a blue background color, and the dimension symbol.
- *In PDF files*, dimensions are printed without a border, there is no background color, and no dimension symbol. Dimensions are printed with a left margin.

Common Dimension Dialog Box Options

Several dialog box options are common to many of the **Dimension** dialog boxes (**Insert | Dimension**).



Material Conditions



The **Material Conditions** area for a dimensioned feature (**Insert | Dimension**) contains the following tolerancing methods:

Option Button	Stands for	Description
M	MMC	Maximum material condition applied to either the feature or a datum.
R	RFS	Regardless of feature size applied to either the feature or a datum.
L	LMC	Least material condition applied to either the feature or a datum.

You can select material condition modifiers for Position, Parallelism, and Perpendicularity dimensions.

Be aware that for position dimensions, selecting the **Use Datums** check box causes the XYZ output values to use the alignment of the datums.

Recall Button

The **Recall** button in the **Material Conditions** area for a dimensioned feature (**Insert | Dimension**) enables you to select the same datums used in a previous similar dimension. This button remains unavailable for selection until you select a feature to dimension from the Feature list.

For example, suppose you used four circles, as your datums in a previous Position dimension. You can tell PC-DMIS to use these same circles as your features by selecting the feature to dimension from the **Feature** list, and clicking the **Recall** button. PC-DMIS will then select the datums from the **Feature** list.

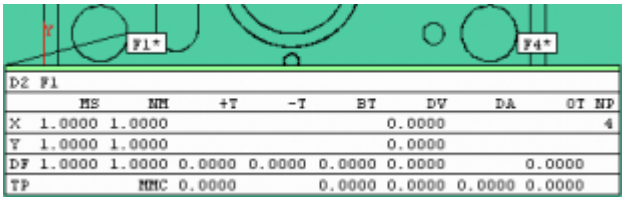
Display Dimension Information

The **Display** check box for a dimensioned feature (**Insert | Dimension**) creates a DIMINFO command in the Edit window after the dimension. This command will show all dimension information in the Graphic Display window next to the feature selected in the **Feature List** box. This DIMINFO command will also display the same dimension axes that are available in the Edit window for that particular dimension.

To use the **Display** option:

1. Select the **Display** check box.
2. Select the feature to dimension.
3. Click the **Create** button.

Dimension information then appears in the Graphic Display window.

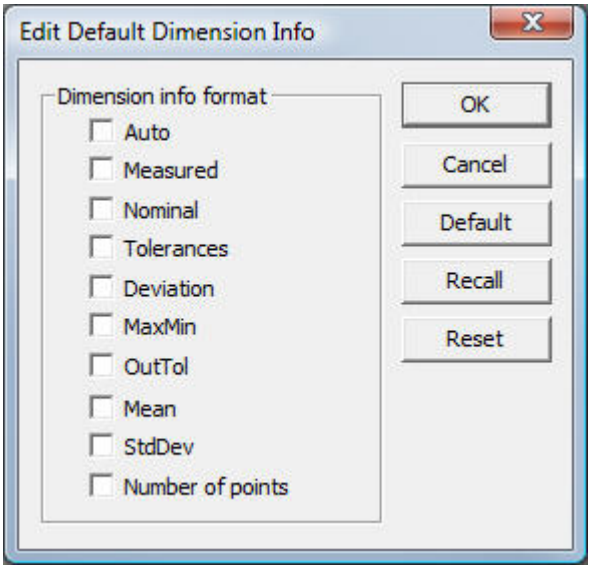


Example of the dimensional data in the Graphic Display window.

For more in-depth information on DIMINFO boxes and on rules that govern their creation, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

Edit Default Dimension Info

The **Edit** button for a dimensioned feature (**Insert | Dimension**) displays the **Edit Default Dimension Info** dialog box.



Edit Default Dimension Info Dialog Box

These check boxes determine the type of information to display in the Graphic Display window for each dimension. The available check boxes are:

Auto
This check box automatically displays the following information: Measured, Nominal, Tolerances, Deviation, MaxMin, OutTol.

Measured
This check box displays the actual measured dimensions.

Nominal
This check box displays the theoretical values for the dimension.

Tolerances
This check box displays the acceptable tolerance levels either greater than or less than the nominal.

Deviation
This check box displays the deviation of the measured value from the nominal.

MaxMin
This check box displays the maximum and minimum deviation values from the points that make up the dimension. While can mark this for any dimension, it only makes sense for Line Profile and Surface Profile dimensions. For an example of this, see

the "Parameter Settings: Dimension tab" topic of the "Setting Your Preferences" chapter.

OutTol
This check box displays how far out of tolerance the measured value is from the nominal and tolerance values.

Mean
This check box displays the average of all the deviations for the dimension.

StdDev
This check box displays the standard deviation of all the deviations for the dimension

Number of Points
This check box displays the number of points used to measure the feature of the dimension.

For more in depth information on editing dimensions and on rules governing their creation, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter. The following five buttons: **OK**, **Cancel**, **Default**, **Recall,** and **Reset** all deal with the **Edit Default Dimension Info** dialog box.

Reset Button

The **Reset** button for a dimensioned feature clears any selected check boxes in the **Edit Default Dimension Info** dialog box (**Insert | Dimension | Edit** button) and selects the **Auto** check box.

The **Reset** button completes the descriptions of the buttons found in the **Edit Default Dimension Info** dialog box. The following buttons and sections finish describing other common functions found among the various dimension dialog boxes.

Units

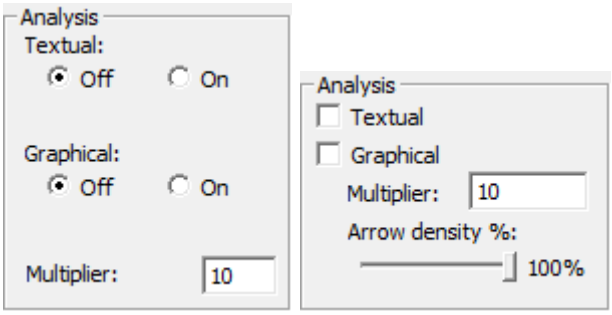
The **Units** area for a dimensioned feature (**Insert | Dimension**) enables you to choose between the following two options:

- Inch = Inches
- MM = Millimeters

When you create a dimension (either legacy or Geometric Tolerance) for the first time, PC-DMIS uses the measurement unit of the current measurement routine. Thereafter, PC-

DMIS recalls what you selected on your last created dimension when you next open a dimension or the **Geometric Tolerance** dialog box.

Analysis Settings



Analysis area for Location and Position dimensions (left) and all other dimensions (right).

The **Analysis** area for a dimensioned feature (**Insert | Dimension**) enables you to set the dimensional output analysis format to textual, graphical, or both when viewing a dimension's deviations.

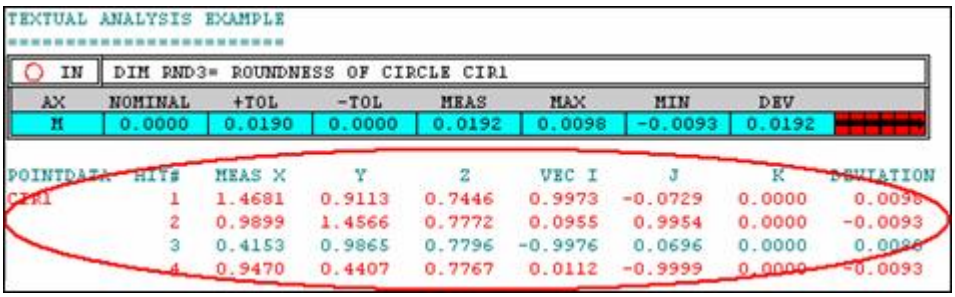
Unique to Location and Position dimensions, the **Analysis** area allows you to analyze not only a feature's location, but a feature's *form* as well.

> Use caution when interpreting a Location or Position dimension's graphical data along with its integrated form dimension. Analysis tolerance lines sometimes overlap, which makes it difficult to distinguish them.

Textual

When you select the **Textual** check box (or the **On** option for Location and Position dimensions) for a dimensioned feature (**Insert | Dimension**), PC-DMIS will print out the following in the inspection report for each individual hit used in the dimension:

- Measured X, Y, and Z values
- Measured I, J, and K values
- Deviation of each individual hit
- "MAX" or "MIN" marker at the end of the line whenever the hit produces either a maximum or minimum deviation.



TEXTUAL ANALYSIS EXAMPLE

IN | DIM RND3= ROUNDNESS OF CIRCLE CIR1

AX	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	DEV
M	0.0000	0.0190	0.0000	0.0192	0.0098	-0.0093	0.0192

POINTDATA	HIT#	MEAS X	Y	Z	VEC I	J	K	DEVIATION
CIR1	1	1.4681	0.9113	0.7446	0.9973	-0.0729	0.0000	0.0098
	2	0.9899	1.4566	0.7772	0.0955	0.9954	0.0000	-0.0093
	3	0.4153	0.9865	0.7796	-0.9976	0.0696	0.0000	0.0086
	4	0.9470	0.4407	0.7767	0.0112	-0.9999	0.0000	-0.0093

Example of a textual analysis report.

Graphical

When you select the **Graphical** check box (or the **On** option for Location and Position dimensions) for a dimensioned feature (**Insert | Dimension**), the software provides a graphical view of the dimension in the Graphic Display window.

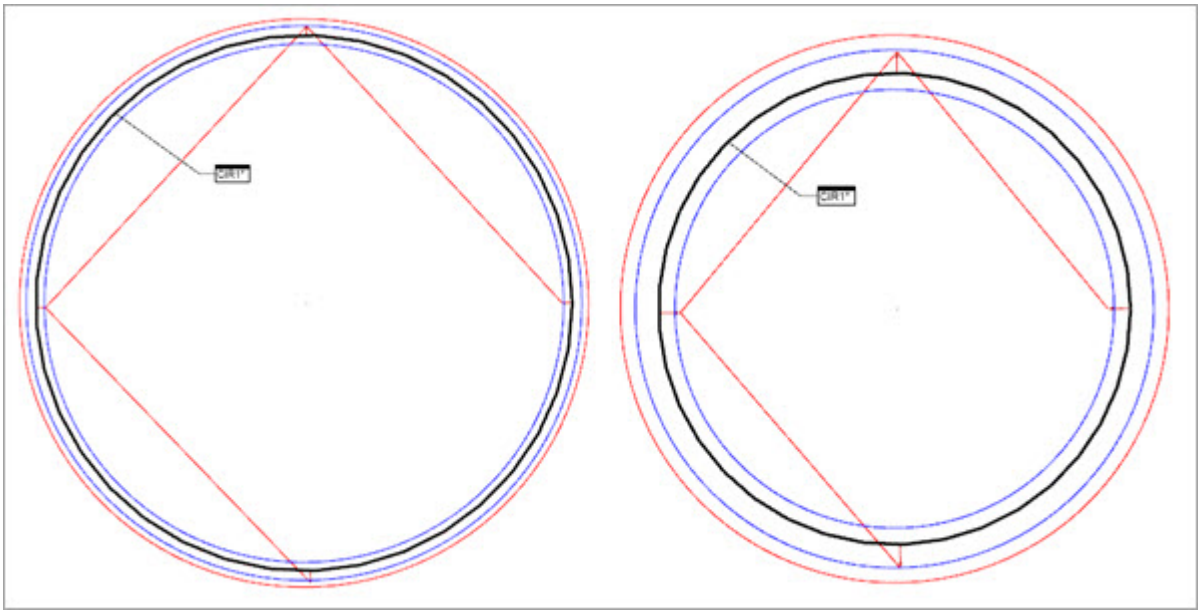
In that window, PC-DMIS displays the deviation of each dimension's individual probe hit as individual-colored arrows. These arrows, with their colors and directions, indicate the relative size of the deviation, as well as its direction.

For the Position, Concentricity, and Coaxiality dimensions, PC-DMIS shows the position of a feature with one or more arrows. For Symmetry dimensions of width features, PC-DMIS shows the points that make up the center plane.

Command line in the Edit window:

```
...GRAPH=BOTH TEXT=OFF MULT=10.00
```

Consider this example:



Example of a Location Dimension using Graphical Analysis with a multiplier of 10 (on the left) and 25 (on the right)

The Location example above contains various colored lines, arrows, and circles. The following items describe the colored elements in the example and what they mean:

Black Circle = Shows the nominal circle.

Red Circle = Shows the measured, out-of-tolerance circle.

Blue Circles = Show the tolerance bands.

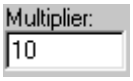
Colored Lines = Show the lines between hits.

Colored Arrows = Show the hits themselves (the tip of the arrow), the relative size of the deviation for each hit (by the colored arrow), and in the deviation direction (the direction the arrow points).

The colors from the example above are the default dimension colors. They correspond to the colors on the Dimension Colors window. For information, see "Editing Dimension Colors" in the "Editing the CAD Display" chapter.

For information on using the **Textual** check box to display dimension information for textual analysis, see the example in the "Textual" topic.

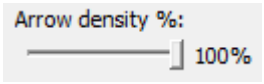
Multiplier



The **Multiplier** box for a dimensioned feature (**Insert | Dimension**) is a scaling factor that magnifies the deviation arrows and tolerance zone by the value typed for the graphical analysis mode. If you type a 2.0 value, PC-DMIS will scale the arrows two times the calculated deviation for each feature hit.

This changes the arrow's size for viewing purposes only. It does not affect the size of the deviation in the text printout in any way.

Arrow Density



This option for a dimensioned feature (**Insert | Dimension**) lets you set the percentage of dimension arrows, representing individual points, that are displayed in the Graphic Display window when graphical analysis is used on a dimension. This limits the amount of arrows drawn for these supported legacy dimensions: circularity, cylindricity, flatness, straightness, profile, and runout.

Both Option (for Position and Location)

- **Textual**: Selecting **Both** for a dimensioned feature (**Insert | Dimension**) displays deviations for both the location *and the form* axes in the inspection report.
- **Graphical:** Selecting **Both** displays tolerancing lines for both a feature's location dimension *and the integrated form dimension*, possibly overlapping one another.

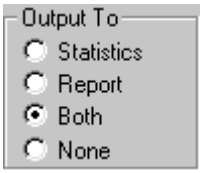
Form Option (for Position and Location)

With the **Form** option for a dimensioned feature (**Insert | Dimension**), you can analyze dimensional information for a feature's location *and* a feature's form at the same time.

You must select the **Form** check box in the **Axes** area of the dialog box to use the **Analysis** area to report on a feature's form. See "Default Axes" for additional information.

Selecting **Form** acts as if you automatically selected the **Both** option as well, since *both* a graphical and a textual analysis are generated.

Output To



Dimension output for a dimensioned feature (**Insert | Dimension**) can be printed to either the inspection report or to statistical files used by statistics software, to both of these, or to none at all. This is controlled through the **Output To** area of the dialog box, which contains these options:

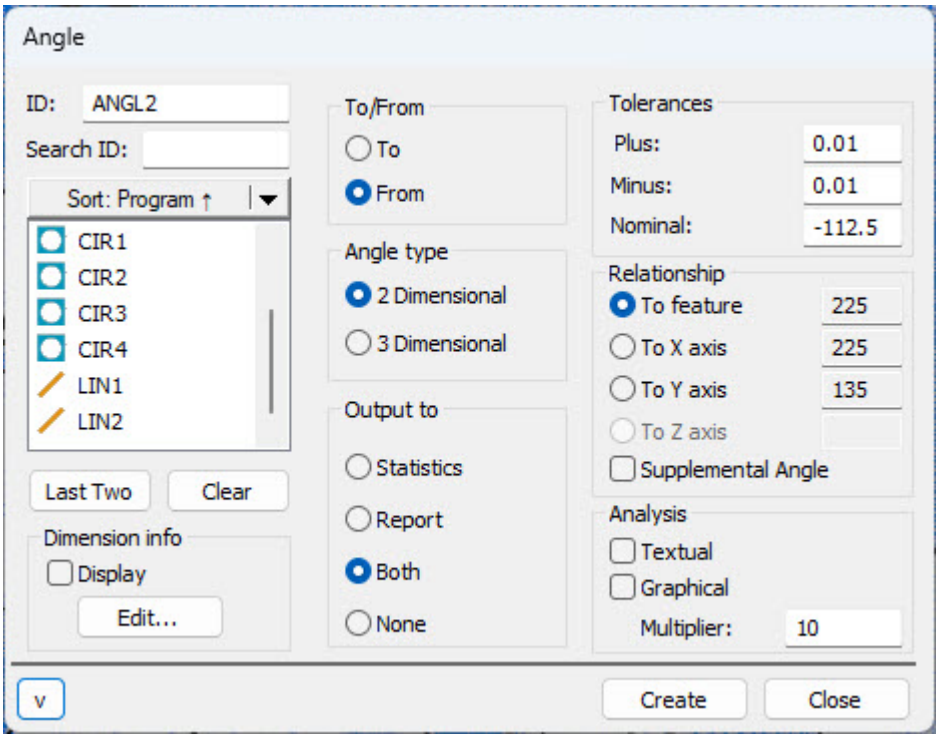
- **Statistics** - sends output to statistical files
- **Report** - sends output to inspection report
- **Both** - sends output to both inspection report and statistical files
- **None** - doesn't send dimension output anywhere

Upon the dimension's execution, the output will go to the inspection report, the stats file, both of these, or none of them (depending on the selection).

Note that if either the **Stats** option or the **Both** option is selected, a preceding `STATS/ON` command must exist inside the Edit window for this dimension to be sent to the stats file.

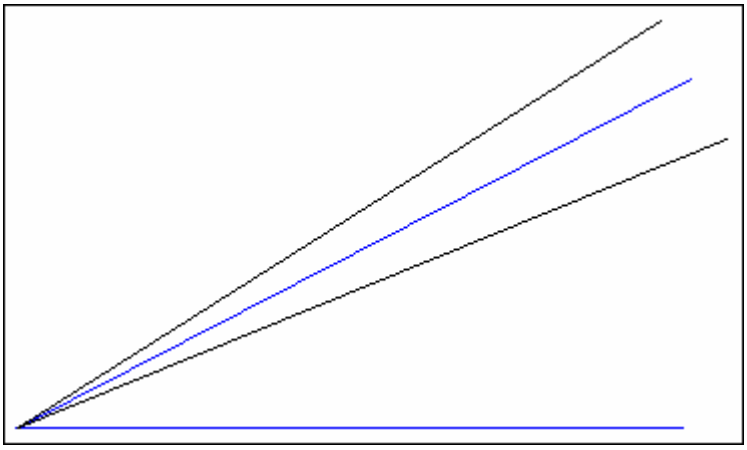
Dimensioning Angle

If you load a measurement routine from PC-DMIS version 3.7 or earlier, PC-DMIS displays a message to inform you that the way Angle dimensions were calculated has changed and that all Angle dimensions in the measurement routine were updated.



Dimension Angle Dialog Box

The **Insert | Dimension | Angle** menu option calculates the angle between a selected feature and either a second selected feature or a coordinate axis.



Example showing the measured angle (blue) and the angularity tolerance band (black).

For a 2D angle type, PC-DMIS projects the vectors onto the current workplane. It calculates the angle from the first feature to the second feature or axis.

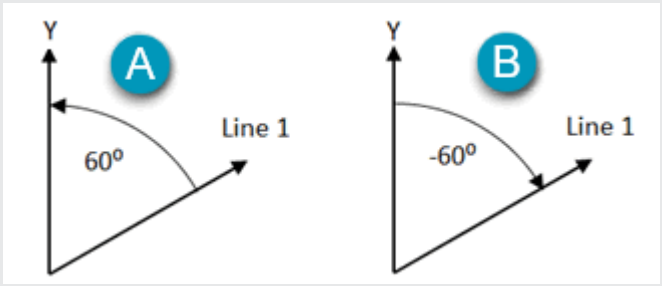
PC-DMIS gives a positive angle if it calculates the angle in a counterclockwise direction. It gives a negative angle if it calculates the angle in a clockwise direction.

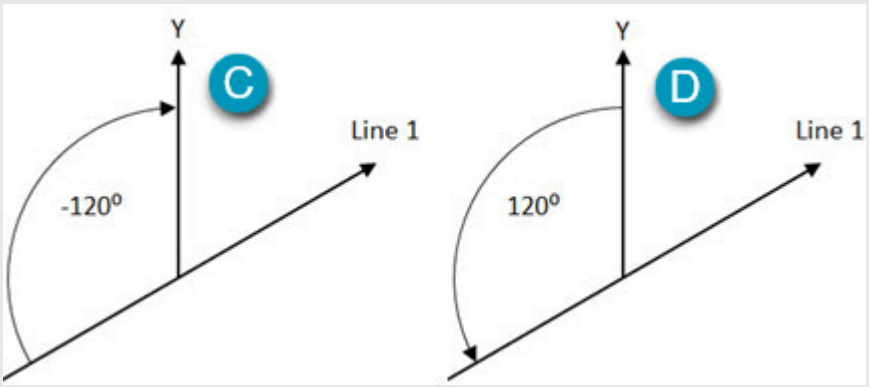
The **To** and **From** options determine the direction of the calculation and the resulting sign (positive or negative):

- If you select **To**, PC-DMIS calculates the angle from feature 1 to feature 2 (or the selected axis).
- If you select **From**, PC-DMIS calculates the angle to feature 1 from feature 2 (or the selected axis).

By default, PC-DMIS uses the angle from feature 1 to feature 2 (or the selected axis). If you want the supplemental angle (180 degrees - angle), mark the **Supplemental Angle** box.

Consider the following examples:





Example A - From line feature (Line 1) to the Y axis (**To** option)

Example B - To line feature (Line 1) from the Y axis (**From** option)

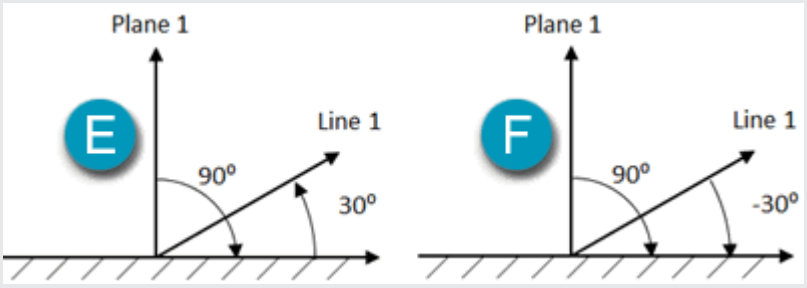
Example C - From line feature (Line 1) to the Y axis (**To** option with **Supplemental Angle**)

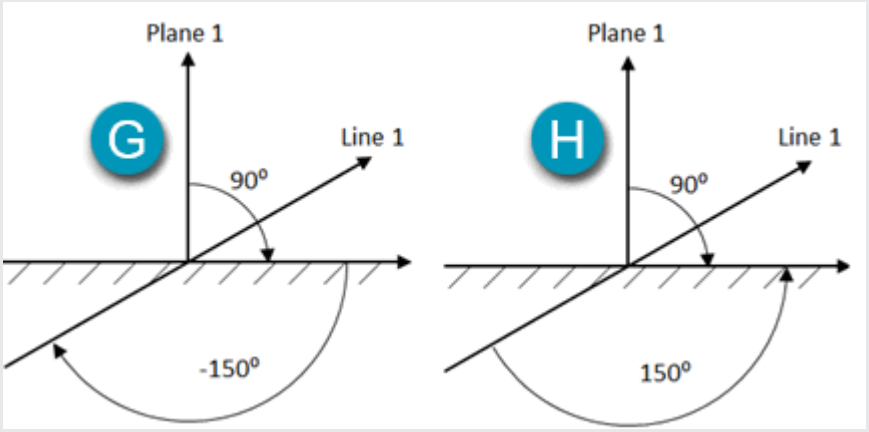
Example D - To line feature (Line 1) from the Y axis (**From** option with **Supplemental Angle**)

If you select one of these features (or feature types), PC-DMIS rotates the surface normal vector 90 degrees clockwise to represent a vector along the surface:

- Plane
- 2D feature with width
- 3D feature with width

Consider the following examples:





Example E - From plane feature with normal vector (Plane 1) to line feature (Line 1) (**To** option)

Example F - To plane feature with normal vector (Plane 1) from line feature (Line 1) (**From** option)

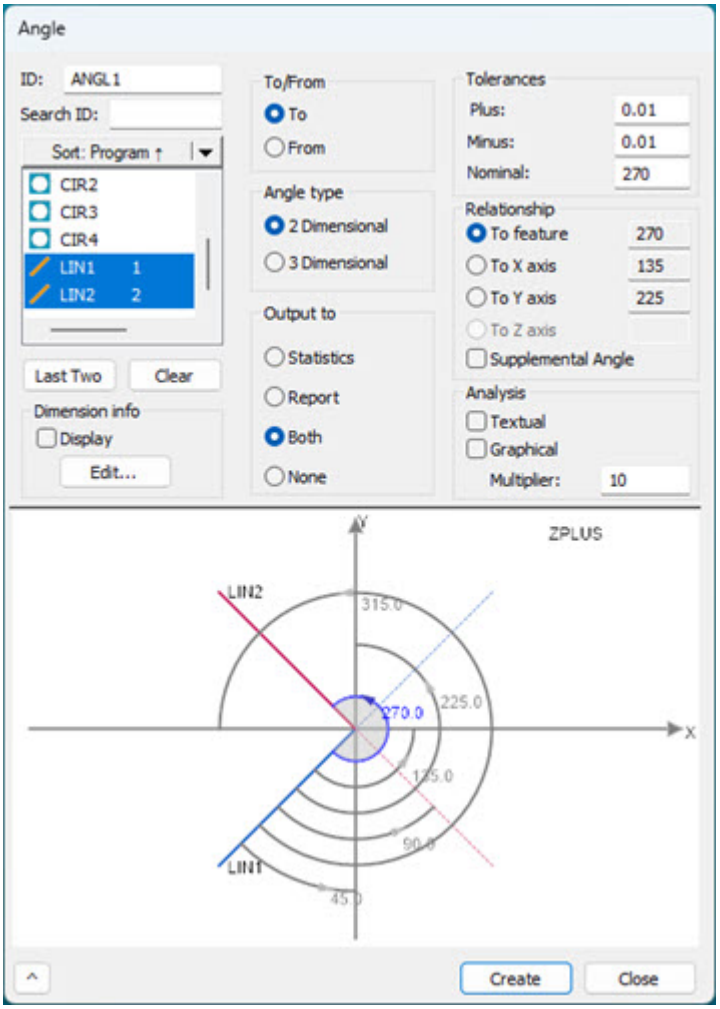
Example G - From plane feature with normal vector (Plane 1) to line feature (Line 1) (**To** option with **Supplemental Angle**)

Example H - To plane feature with normal vector (Plane 1) from line feature (Line 1) (**From** option with **Supplemental Angle**)

Angle Visualization

You can use the Angle Visualization function to view the possible angles associated with the **To/From** and **Angle type** (2D or 3D) relationship.

To do this, click the **Expand** button in the lower-left corner of the **Angle** dialog box to show the **Angle Visualization** area.



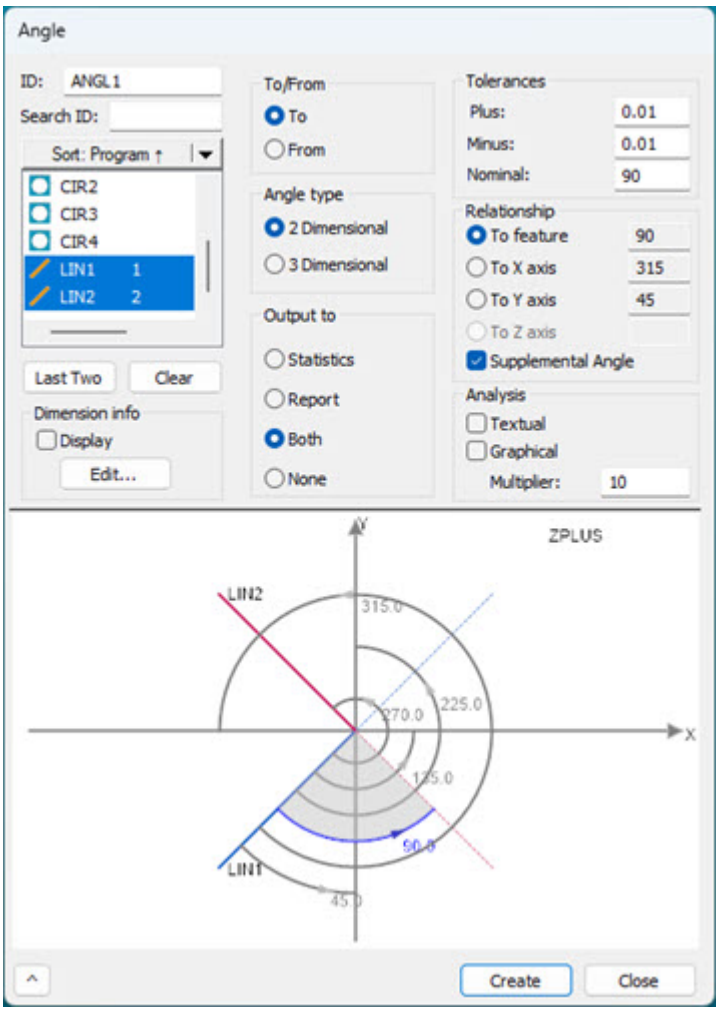
Example of the Angle dialog box for a 2D angle showing the Angle Visualization area.

For 2D angles, PC-DMIS draws the coordinate axes in the active workplane. In the image above, this is the ZPLUS workplane as shown in the upper-right of the Angle Visualization area.

PC-DMIS highlights the current selection in blue with a shadow to show the sweep angle.

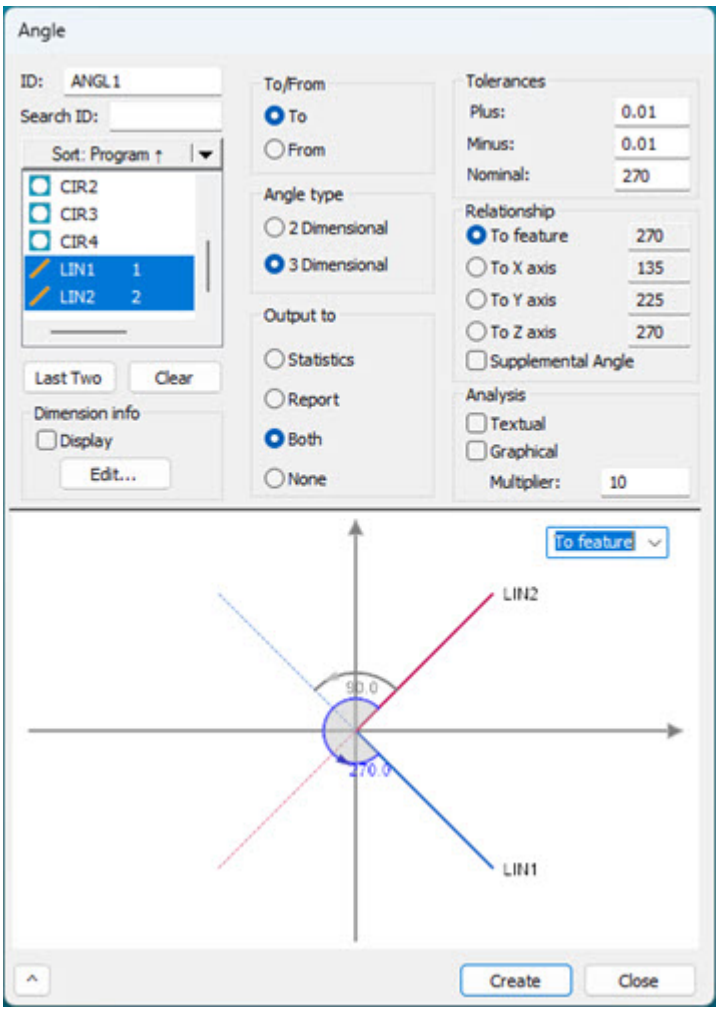
You can click any of the angles to view their properties. When you do, PC-DMIS dynamically:

- Updates the values for the selected angle in the **Tolerances** area
- Selects the appropriate **Relationship** option related to the angle and updates the values for that angle in the **Relationship** boxes
- Auto-selects the **Supplemental Angle** check box if applicable



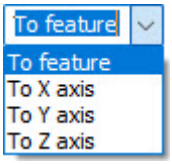
Example showing the updated properties when you select a different angle.

For 3D angles, PC-DMIS draws the references in the projection plane and shows the angles of the selected relationship.



Example of the Angle dialog box for a 3D angle showing the Angle Visualization area.

The projection plane varies with different input features. Select an option from the **Relationship** area or from the **Relationship** list to change the angle's view and properties.



Relationship List

You can also click a different angle sweep in the **Angle Visualization** area to view the properties for that angle.

Click the **Collapse** button to hide the **Angle Visualization** area.

To Dimension the Angle Between Two Features:

1. Select **Insert | Dimension | Angle** from the submenu. The **Angle** dialog box appears.
2. Select the feature or features to dimension from the **Feature List** box.

3. Type the plus tolerance value in the **Plus** box.
4. Type the minus tolerance value in the **Minus** box.
5. Type the nominal angle in the **Nominal** box.
6. Select either the **2 Dimensional** or **3 Dimensional** option in order to specify the angle type.
7. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
8. Select the **To Feature**, **To X Axis** , **To Y Axis**, or **To Z Axis** option to determine the relationship that defines the angle.
9. To change the sign of the angle and direction of the calculation, select the **From** option.
10. To switch to the supplemental angle (180 degrees - angle), mark the **Supplemental Angle** check box.
11. If you don't use a CAD model, and you have not set the theoretical vectors of the feature, you may need to edit the nominal angle.
12. Select the desired analysis options by selecting the **Textual** check box or the **Graphical** check box. If the Graphical check box was selected, type the multiplier value in the Multiplier box.
13. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
14. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = 2D_ANGLE,FROM feat_1,TO feat_2,
```

or

```
dimension_name = 3D_ANGLE,FROM feat_1,TO feat_2
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
A	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Editing Degrees, Minutes, and Seconds Values

When manually editing the Angle's nominal or tolerance value in the Edit window, and the angle value is displayed in Degrees, Minutes, and Seconds, you need to type a comma between each value. For example, if you wanted 100°33'51" as your Nominal angle value,

you would need to type 100,33,51 in the Nominal field and then press the Tab key for PC-DMIS to accept that value.

To determine whether or not PC-DMIS displays Angle in Degrees, Minutes, and Seconds, use the `AngleDegMinSec` entry in the **Option** section of the PC-DMIS Settings Editor. A value of 1 uses this format, while 0 uses the standard decimal value format.

Tolerances for Angle Dimensions

The **Tolerances** area in the **Angle** dialog box (**Insert | Dimension | Angle**) enables you to type plus and minus tolerances along the plus and minus directions.

Plus Tolerance

The **Plus** box in the **Angle** dialog box (**Insert | Dimension | Angle**) enables you to enter a tolerance value along the plus direction. This means that any profile that measures more than your nominal or theoretical profile can still be a valid measurement as long as it falls within the tolerance range specified.

PC-DMIS also allows a negative-upper tolerance (or a minus tolerance in the positive range). This is done by entering in a minus sign before the value in the **Plus** box.

Minus Tolerance

The **Minus** box in the **Angle** dialog box (**Insert | Dimension | Angle**) enables you to enter a tolerance value along the minus direction. This means that any profile that measures less than your nominal or theoretical profile can still be a valid measurement as long as it falls within the tolerance range specified.

PC-DMIS allows positive-lower tolerances (or a positive tolerance in the negative range). This is done by typing a minus sign before the value in the **Minus** box.

Nominal for Angle Between

Nominal values for Angle dimensions aren't always based on CAD data or measured data. Often you will obtain them from a hard copy printout, you can use the **Nominal** box in the **Angle** dialog box (**Insert | Dimension | Angle**) to enter the nominal angle value from the printout.

Angle Type

The **Angle type** area in the **Angle** dialog box (**Insert | Dimension | Angle**) determines whether or not the angle is 2 or 3 dimensional.

2 Dimensional Angle Type

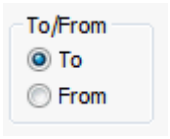
The **2 Dimensional** option in the **Angle** dialog box (**Insert | Dimension | Angle**) calculates the two-dimensional angle between features.

3 Dimensional Angle Type

The **3 Dimensional** option in the **Angle** dialog box (**Insert | Dimension | Angle**) calculates the three-dimensional angle between features. If only one feature is selected, the angle is calculated between the current workplane and the selected feature.

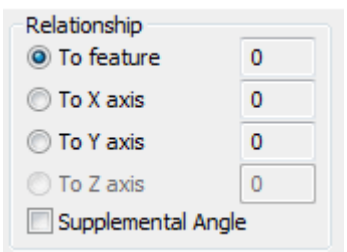
See "Dimensioning Angle" above for instructions on converting the quadrant of the reported angle.

Relationship for Angle Dimensions



The **To/From** area in the **Angle** dialog box (**Insert | Dimension | Angle**) determines the direction of the angle calculation and the resulting sign (positive or negative).

- If you select **To**, PC-DMIS calculates the angle from feature 1 to feature 2 (or selected axis). This also changes the options in the **Relationship** area begin with "To".
- If you select **From**, PC-DMIS calculates the angle to feature 1 from feature 2 (or selected axis). This also changes the options in the **Relationship** area begin with "From".



The **Relationship** area determines whether the Angle dimension can calculate the angle between two features, or between one feature and a particular axis: X axis, Y axis, and Z axis. As soon as you select at least one feature for the Angle dimension, PC-DMIS fills in the boxes next to the option buttons with the computed nominal angle values.

To feature / From feature
Select the **To feature** option if you are measuring the angle between two features. The second feature will be the datum feature.

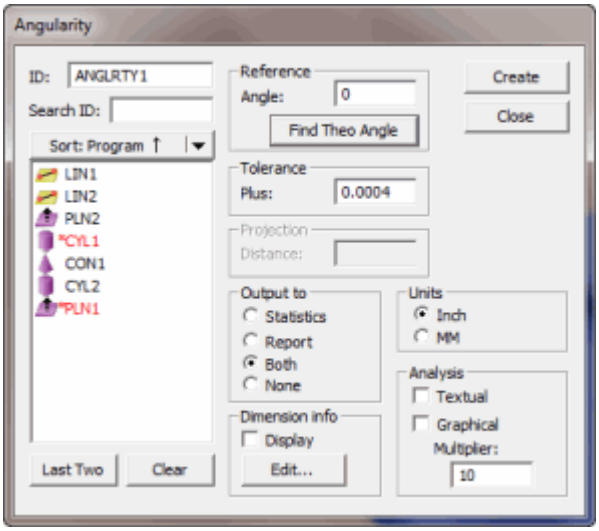
To X axis / From X axis
Select the **To X axis** check box if you are measuring the angle between a feature and the X axis.

To Y axis / From Y axis
Select the **To Y axis** check box if you are measuring the angle between a feature and the Y axis.

To Z axis / From Z axis
Select the **To Z axis** check box if you are measuring the angle between a feature and the Z axis.

Supplemental Angle
By default, PC-DMIS uses the angle from feature 1 to feature 2 (or the selected axis). If you want to use the supplemental angle (180 degrees - angle), mark the **Supplemental Angle** check box.

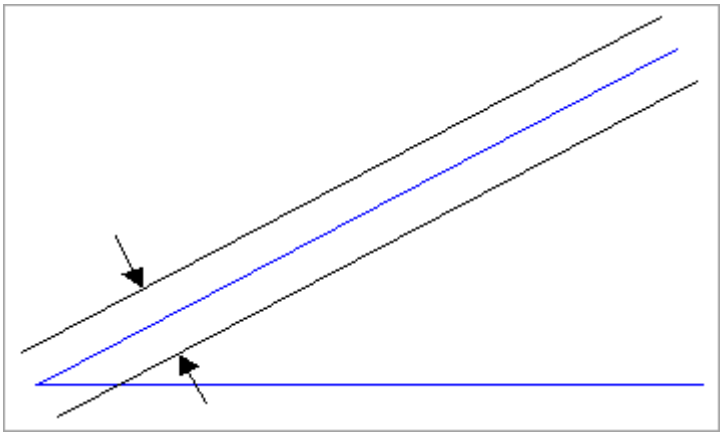
Dimensioning Angularity



Dimension Angularity Dialog Box

The **Insert | Dimension | Angularity** menu option computes the angularity error of a plane or line to a datum plane or line. (If only one feature is selected or entered, PC-DMIS uses the current workplane as the datum.)

Similar to perpendicularity or parallelism, the Angularity dimension allows you to specify an angle that isn't 90 degrees (perpendicularity) or 0 degrees (parallelism). For example, if you specify a 45-degree angle, PC-DMIS creates a tolerance band at 45 degrees and checks that the hits are within that tolerance band.



Example showing the measured angle (blue) and the angularity tolerance band (black) with two parallel lines or planes.

To Compute the Angularity Error by Using the Angularity Option

1. Select **Insert | Dimension | Angularity** from the submenu. The **Angularity** dialog box appears.
2. Type the nominal angle from the datum in the **Angle** box.
3. Select or type the plane or line, feature, and datum.
4. Type the plus tolerance value in the **Plus** box.
5. Type the projection distance in the **Distance** box.
6. Select either **Inch** or **MM** in the **Units** area.
7. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
8. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
9. Select either **Form Only** or **Form and Location** in the **Control Options** area of the dialog box.
10. Select the desired analysis options by selecting the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was selected, type the multiplier value in the **Multiplier** box.

11. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.

12. Click the **Create** button.

The dimension appears in the Edit window with this information:

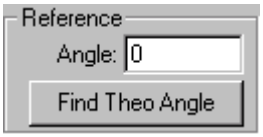
```
dimension_name = ANGULARITY FROM feat_1 TO feat_2
EXTENDLENGTH=n ANG=n UNITS=MM/IN

GRAPH=ON/OFF TEXT=ON/OFF MULT=n OUTPUT=NONE/BOTH/STATS/
REPORT
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

feat_2: LINE or PLANE.

PC-DMIS then computes the angularity error and displays it in the current dimension window.

Reference Angle



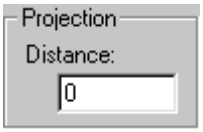
The **Angle** box in the **Angularity** dialog box (**Insert | Dimension | Angularity**) enables you to type the nominal angle from the datum feature. This is the angle between the two features. PC-DMIS then calculates the deviation of one feature from the angle.

Clicking the **Find Theo Angle** button calculates the nominal reference angle you should use from the theoretical values of the selected features. This makes it so you don't have to go and look up the angle in the Edit window.

Plus Tolerance for Angularity Dimensions

The **Plus** box in the **Angularity** dialog box (**Insert | Dimension | Angularity**) enables you to type a tolerance value along the plus direction. This means that any angularity error that differs from your nominal or theoretical angularity error can still be a valid measurement as long as it falls within the tolerance range specified.

Projected Distance for Angularity Dimensions

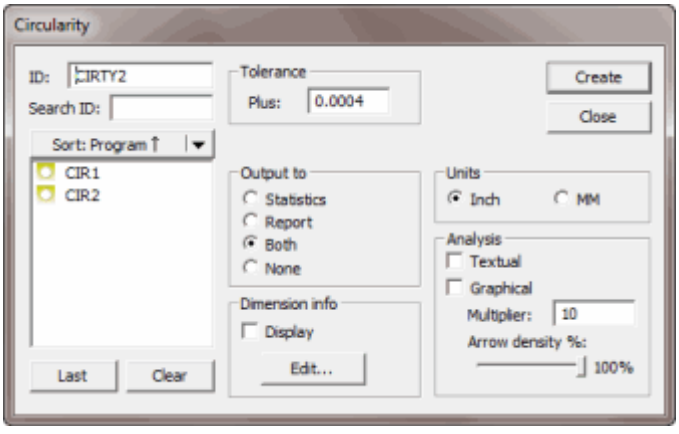


PC-DMIS allows you to project a reference distance. This option in the **Angularity** dialog box (**Insert | Dimension | Angularity**) is used with line features to calculate the angularity dimension.

For these features, the points used for the angularity calculation are the end point of the axis and a point projected from this end point along the feature axis.

The distance between these two points is the referenced distance. For other features, this distance has no effect on the dimension. This option is useful when the dimension needs to be calculated a certain distance along the feature.

Dimensioning Circularity



Dimension Circularity Dialog Box

One hit, plus the feature's minimum hit requirement is necessary to determine the roundness of that feature. Additional hits provide a better representation of the roundness of the entire feature.

The **Insert | Dimension | Circularity** menu option determines the roundness of a circle, sphericity of a sphere, or the conicity of a cone. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

To Dimension a Feature Using the CIRCULARITY Option

1. Select **Insert | Dimension | Circularity** from the submenu. PC-DMIS opens the **Circularity** dialog box.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** section.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired Analysis options by marking the **Textual** check box or the **Graphical** check box. If you selected the Graphical check box, type the Multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

PC-DMIS shows the dimension in the Edit window with this information:

```
dimension_name = CIRCULARITY,OF feat_1
```

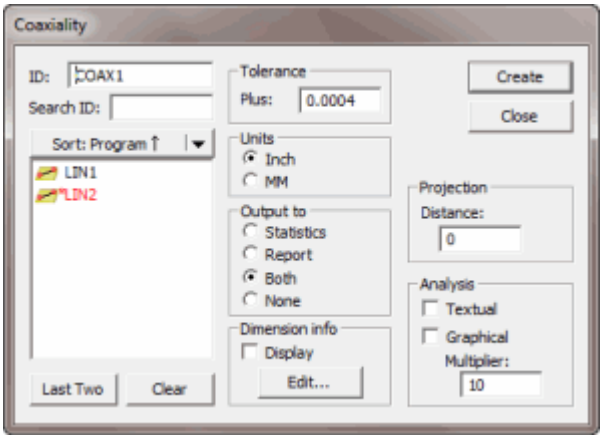
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Circularity Dimensions

The **Plus** box in the **Circularity** dialog box (**Insert | Dimension | Circularity**) enables you to enter a tolerance value along the plus direction. This means that any roundness that differs from your nominal or theoretical roundness can still be a valid measurement as long as it falls within the tolerance range specified.

For information on how PC-DMIS reports tolerance zones for form dimensions, see "Reported Tolerance Zones for Form Dimensions" in the "Reporting Measurement Results" chapter.

Dimensioning Coaxiality



Dimension Coaxiality Dialog Box

The **Insert | Dimension | Coaxiality** menu option calculates the coaxiality of a cylinder, cone, or line with a datum feature. The second feature that is entered is always the datum feature, and may be a cylinder, cone, line, or circle. If only one feature is selected, and you click **Create**, PC-DMIS will clear the list box and display a message informing you that it needs a second feature.

This dimension type is considered one sided, meaning a single positive value tolerance is applied.

To Dimension a Feature Using the COAXIALITY Option:

1. Select **Insert | Dimension | Coaxiality** from the submenu. The **Coaxiality** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** area.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** options.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired Analysis options by selecting one or both of the check boxes. If the **Graphical** check box was selected, enter the Multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.

9. Click the **Create** button.

The dimension appears in the Edit window with this information:

```
dimension_name = COAXIALITY,FROM feat_1,TO feat_2
```

or

```
dimension_name = COAXIALITY,FROM feat_1,TO THE ORIGIN
```

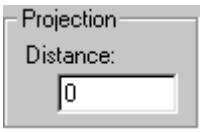
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	2.0000	0.0000	0.0000	0.0000	0.0000

After the dimension is created, the dialog box remains open. If desired, use your **Edit** button at this time to edit aspects of your dimension's information. See "Edit Default Dimension Info".

Plus Tolerance for Coaxiality Dimensions

This **Plus** box in the **Coaxiality** dialog box (**Insert | Dimension | Coaxiality**) enables you to enter a tolerance value along the plus direction. This means that any coaxiality that differs from your nominal or theoretical coaxiality can still be a valid measurement as long as it falls within the tolerance range specified.

Projected Distance for Coaxiality



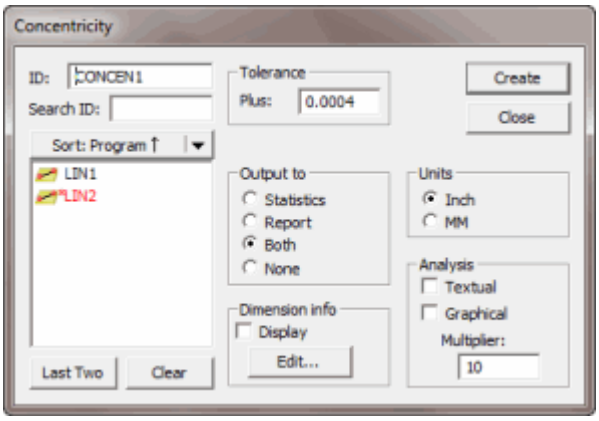
PC-DMIS allows you to type a reference distance. This option in the **Coaxiality** dialog box (**Insert | Dimension | Coaxiality**) is useful when you want to specify the points along the feature axis to be used in the coaxiality calculations. When this distance is zero, the two points used will be the endpoints of the feature's axis. When this distance is nonzero, the two points used will be the starting point of the axis, and along the axis vector the distance from the starting point.

Increasing the projected distance will magnify the error. Decreasing the distance lessens the error. For example, suppose two toothpicks lay on a table, sharing the same line and touching each other at one end. At the other end of the toothpicks, about two inches apart, the error between their centerlines might be 0.005 inches. Now imagine those toothpicks are 20 inches long, they would still touch at one end and the error would be 0.005 inches

at 2 inches away, but at 20 inches away from the first end, the toothpicks might have a 0.05 inch error.

The value you type in the **Distance** box reflects the total length of the feature.

Dimensioning Concentricity



Dimension Concentricity Dialog Box

If a sphere is selected as the first feature, the second feature must also be a sphere. PC-DMIS will then three-dimensionally calculate the concentricity between the two features.

The **Insert | Dimension | Concentricity** menu option calculates the concentricity of two circles or cylinders, cones, and spheres. The second feature that is entered is always the datum feature and may be a line feature that represents an axis. If only one feature is selected, the current workplane becomes the datum feature. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

To Dimension a Feature Using the CONCENTRICITY Option

1. Select **Insert | Dimension | Concentricity** from the submenu. The **Concentricity** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** section.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.

7. Select the desired analysis options by selecting the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, type the multiplier value in the **Multiplier** box.

8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.

9. Click the **Create** button.

The dimension appears in the Edit window with this information:

```
dimension_name = CONCENTRICITY,FROM feat_1,TO feat_2
```

or

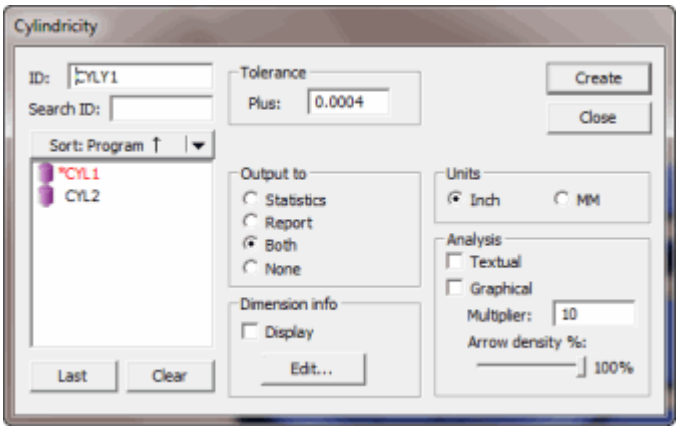
```
dimension_name = CONCENTRICITY,FROM feat_1,TO THE ORIGIN
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	2.0000	0.0000	0.0000	0.0000	0.0000

Tolerance for Concentricity Dimensions

The **Plus** box in the **Concentricity** dialog box (**Insert | Dimension | Concentricity**) enables you to enter a tolerance value along the plus direction. This means that any concentricity that differs from your nominal or theoretical concentricity can still be a valid measurement as long as it falls within the tolerance range specified.

Dimensioning Cylindricity



Dimension Cylindricity Dialog Box

The **Insert | Dimension | Cylindricity** menu option determines the cylindricity of a cylinder. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

To Dimension a Feature Using the CYLINDRICITY Option

1. Select **Insert | Dimension | Cylindricity** from the submenu. The **Cylindricity** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box. You can only select cylinder features.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** section.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired Analysis options by marking the **Textual** check box or the **Graphical** check box. If the Graphical check box was marked, type the Multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

The dimension appears in the Edit window with this information:

```
dimension_name = CYLINDRICITY,OF feat_1
```

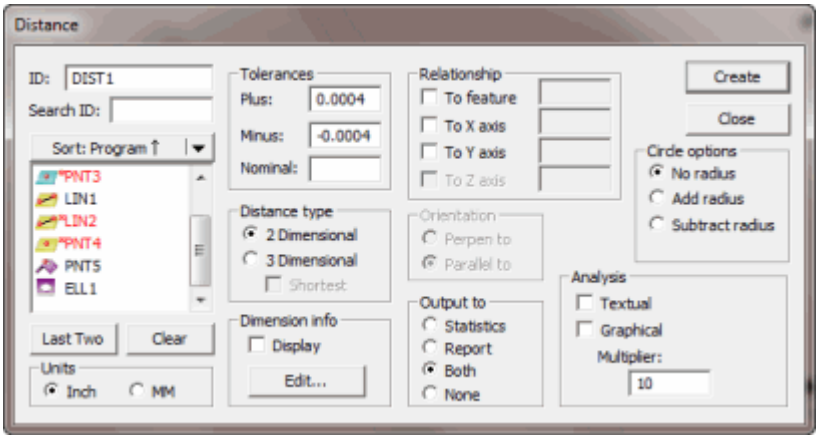
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Cylindricity Dimensions

The **Plus** box in the **Cylindricity** dialog box (**Insert | Dimension | Cylindricity**) enables you to enter a tolerance value along the plus direction. This means that any cylindricity that differs from your nominal or theoretical cylindricity can still be a valid measurement as long as it falls within the tolerance range specified.

For information on how PC-DMIS reports tolerance zones for form dimensions, see "Reported Tolerance Zones for Form Dimensions" in the "Reporting Measurement Results" chapter.

Dimensioning Distance



Distance Dialog Box

The **Insert | Dimension | Distance** menu option calculates the distance between two features. You can select a third feature or an axis for the direction to use in the calculation.

Calculating distance is a bit more difficult to visualize than most of the other dimension calculations. The distance that is calculated between two features uses either the default or straight line method. Two-dimensional distances using a straight line will be parallel to the workplane. Three-dimensional distances using a straight line will be from centroid to centroid.

To Dimension a Distance Using the DISTANCE option:

1. Select **Insert | Dimension | Distance** from the submenu. The **Distance** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Type the minus tolerance value in the **Minus** box.
5. Select either the **2 Dimensional** or **3 Dimensional** option in order to specify the distance type.
6. Select either **Inch** or **MM** in the **Units** area.
7. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both,** or **None** option.
8. Select either the **To Feature**, **To X Axis** , **To Y Axis**, or **To Z Axis** option to determine the relationship that defines the distance.
9. Select either the **Perpen To** or the **Parallel To** radio button.
10. Select the desired analysis options by selecting the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was selected, type the multiplier value in the **Multiplier** box.
11. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
12. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
 DIM dimension_name = 2D_DISTANCE FROM feat_1 TO feat_2 TOG1 TO
TOG2, TOG3, UNITS=MM/IN,
GRAPH=ON/OFF TEXT=ON/OFF MULT=n OUTPUT=NONE/REPORT/STATS
```

or

```
 DIM dimension_name = 3D_DISTANCE FROM feat_1 TO feat_2, TOG3,
UNITS=MM/IN,
GRAPH=ON/OFF TEXT=ON/OFF MULT=n OUTPUT=NONE/REPORT/STATS
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

TOG1 = PAR / PERP (Parallel or Perpendicular)

TOG2 = XAXIS / YAXIS / ZAXIS / feature

work plane = this field can be changed to any feature. The default is the current workplane.

TOG3= NO_RADIUS / ADD_RADIUS / SUB_RADIUS

Tolerances for Distance Dimensions

The **Tolerances** area allows you to type plus and minus tolerances for distances along the plus and minus directions.

Plus Tolerance

The **Plus** box in the **Distance** dialog box (**Insert | Dimension | Distance**) enables you to enter a tolerance value along the plus direction. This means that any angle that measures more than your nominal or theoretical angle can still be a valid angle as long as it falls within the tolerance range specified.

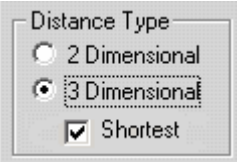
Minus Tolerance

This **Minus** box in the **Distance** dialog box (**Insert | Dimension | Distance**) enables you to enter a tolerance value along the minus direction. This means that any angle that measures less than your nominal or theoretical angle can still be a valid angle as long as it falls within the tolerance range specified.

Nominal for Distance

Nominal values for the distance between features aren't always based on CAD data or measured data. Often you will obtain them from a hard copy printout, you can use the **Nominal** box in the **Distance** dialog box (**Insert | Dimension | Distance**) to enter the nominal distance value from the printout.

General Rules for 2D and 3D Distance Dimensions



2D and 3D distance dimensions in the **Distance** dialog box (**Insert | Dimension | Distance**) apply the following rules according to the related features:

Treatment of Features

- Spheres, Points, Circles, and Sets are treated as *Points*.
- Slots, Cylinders, Cones, Lines, and 2D Widths are treated as *Lines*.
- Planes and 3D Widths are treated as Planes.

Other Rules

- If both elements are points (as defined above), PC-DMIS gives the shortest distance from point to point.
- If one element is a line (as defined above) and the other is a point, PC-DMIS gives the shortest distance between the line (or centerline) and the point.
- If both elements are lines, and the **Shortest** check box is *not selected*, PC-DMIS gives the shortest distance between the centroid of the first line to the second line. See the "Shortest check box" topic for what happens when it is selected.
- If one element is a plane and the other is a line, PC-DMIS gives the shortest distance between the centroid of the line and the plane
- If one element is a plane and the other is a point, PC-DMIS gives the shortest distance between the point and the plane.
- If both elements are planes, PC-DMIS gives the shortest distance between the centroid of the first plane to the second plane.

2D Distance

The maximum and minimum distance is the measured distance between two lines over the length of the lines.

The **2 Dimensional** option in the **Distance** dialog box (**Insert | Dimension | Distance**) calculates the two-dimensional distance between features. All features used to calculate the 2D distance are first projected into the current workplane before the distance is computed, except as noted below when you attempt to calculate the distance perpendicular to a third feature defined as a plane.

PC-DMIS computes the maximum, minimum, and average distances between the two features. If the distance is between two lines or planes, the maximum, minimum and average are computed using the measurement point data. (Verify that the distance type is set to 2 Dimensional.)

2D distances that are calculated using three features are either parallel or perpendicular to a datum feature. The datum feature can be any previously measured or constructed feature.

If the third feature is a plane and the distance is being calculated perpendicular to the datum feature, PC-DMIS ignores the current workplane.

To calculate the distance using three features:

1. Select the two features to calculate the distance.
2. Select the third (datum) feature. (Using a line for the third feature provides the best results.)
3. Verify that the correct orientation is marked.
4. Select the **To Feature** check box.
5. Click the **Create** button.

PC-DMIS calculates the distance between the first two features parallel or perpendicular to the third (datum) feature or axis.

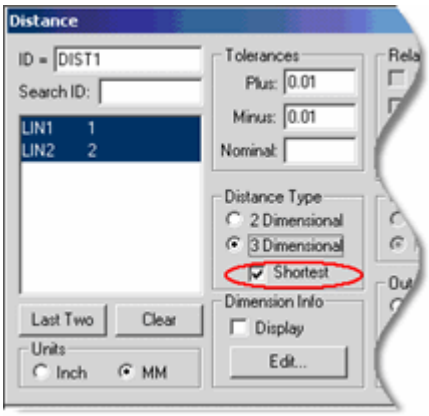
When you click **Close**, PC-DMIS closes the **Distance** dialog box without creating another dimension.

3D Distance

The **3 Dimensional** option in the **Distance** dialog box (**Insert | Dimension | Distance**) calculates the three-dimensional distance between two features.

- If one of the input features is a line, centerline, or plane, PC-DMIS computes the 3D distance normal to that feature.
- If both features are lines, centerlines, or planes the second feature is used for the datum. If both features are planes, the 3D distance is the perpendicular distance from the centroid of the first plane to the second plane; the second plane acts as the datum.
- If neither input feature is a line, centerline, or plane, PC-DMIS computes the shortest distance between the two features. (Verify that the mode is set to 3 Dimensional.)

Shortest Check Box



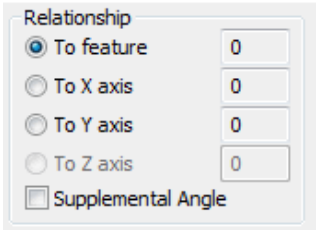
Distance Dialog Box - Shortest Check Box

The **Shortest** check box in the **Distance** dialog box (**Insert | Dimension | Distance**), when selected, calculates the true shortest 3D distance between two lines. This check box becomes enabled for selection when you choose **3 Dimensional** from the **Distance Type** area and the input features for the dimension are two lines.

By default, assuming you are working on a dimension created in version 4.3 and later, PC-DMIS automatically selects the check box when the above conditions are met. However, if you access the **Distance** dialog box for a distance dimension created prior to 4.3, the check box will not be selected by default.

- If you select the **Shortest** check box, PC-DMIS calculates the distance between two lines as the minimum distance between any point on the first line and any point on the second line. The two lines are considered to be unbounded (infinite length).
- If you do not select this check box, PC-DMIS behaves as it did prior to 4.3 and returns the shortest distance between the centroid of the first line to the second line.

Relationship for Distance Dimensions



The options in the **Relationship** area in the **Distance** dialog box (**Insert | Dimension | Distance**) enable you to specify whether the distance measured between two features will be perpendicular to or parallel to a specific axis, or perpendicular to or parallel to a selected feature:

To Feature check box

When you select the **To Feature** check box in the **Distance** dialog box (**Insert | Dimension | Distance**), the **Perpen To** and **Parallel To** options in the **Orientation** area become available for selection. These options tell PC-DMIS to compute the distance between the first selected feature and the second selected feature parallel to, or perpendicular to a certain feature.

- If only *two features* are selected from the list, PC-DMIS computes the distance between feature 1 and feature 2, parallel or perpendicular to feature 2.
- If *three features* are selected from the list, PC-DMIS computes the distance between feature 1 and feature 2, parallel or perpendicular to feature 3.

The feature used to establish the relationship should be a linear feature.

To X Axis check box

Select the **To X Axis** check box in the **Distance** dialog box (**Insert | Dimension | Distance**) if you are measuring the distance from the first selected feature to the second selected feature, perpendicular to or parallel to the X axis.

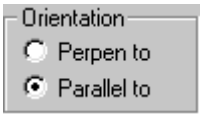
To Y Axis check box

Select the **To Y Axis** check box in the **Distance** dialog box (**Insert | Dimension | Distance**) if you are measuring the distance from the first selected feature to the second selected feature, perpendicular to or parallel to the Y axis.

To Z Axis check box

Select the **To Z Axis** check box in the **Distance** dialog box (**Insert | Dimension | Distance**) if you are measuring the distance from the first selected feature to the second selected feature, perpendicular to or parallel to the Z axis.

Orientation for Distance Dimensions



When measuring the distance between two features, you have the option of determining how the distance is to be measured by using these orientation options in the **Distance** dialog box (**Insert | Dimension | Distance**).

- Measure the distance between the first selected feature parallel or perpendicular to either the second or another selected feature.

- Measure the distance between the first selected feature and the second selected feature parallel to or perpendicular to a specific axis.

The **Perpendicular To** and the **Parallel To** options enable you to determine the orientation of the features.

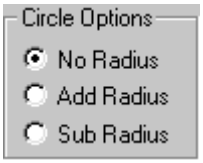
Perpendicular To

Select the **Perpendicular To** option in the **Distance** dialog box (**Insert | Dimension | Distance**) if the distance between the two features is perpendicular to either the second feature, or perpendicular to a specified axis.

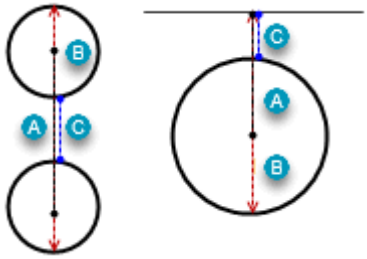
Parallel To

Select the **Parallel To** option in the **Distance** dialog box (**Insert | Dimension | Distance**) if the distance between the two features is parallel to either the second feature, or parallel to a specified axis.

Circle Options



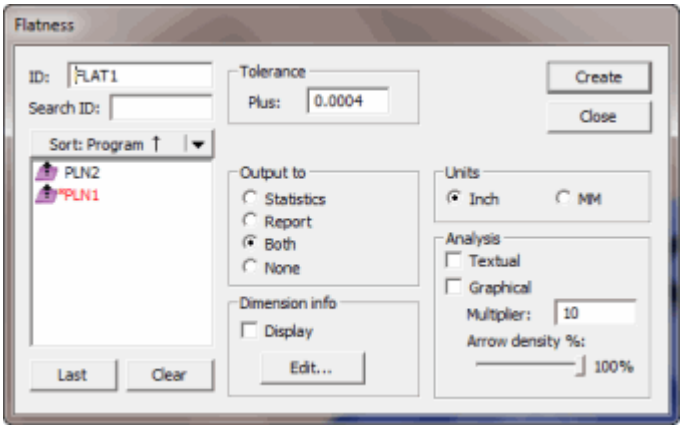
With the **Circle Options** area in the **Distance** dialog box (**Insert | Dimension | Distance**), you can use the **Add Radius** and **Subtract Radius** options to tell PC-DMIS to either add or subtract the radius of the measured feature to the total distance that is measured. The amount that is added or subtracted will always be along the same vector that calculates the distance. Only one option is available at a time.



A - Normal calculated distance between features
B - Add Radius
C - Subtract Radius

Using the **No Radius** option does not apply the radius of the feature to the distance measured.

Dimensioning Flatness



Dimension Flatness Dialog Box

A minimum of four hits are necessary to determine the flatness of a plane.

The **Insert | Dimension | Flatness** menu option determines the flatness of a plane. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

To Dimension a Feature Using the FLATNESS Option

1. Select **Insert | Dimension | Flatness** from the submenu. The **Flatness** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** section.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.

6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired Analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, type the multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

The dimension appears in the Edit window with this information:

```
dimension_name = FLATNESS,OF feat_1
```

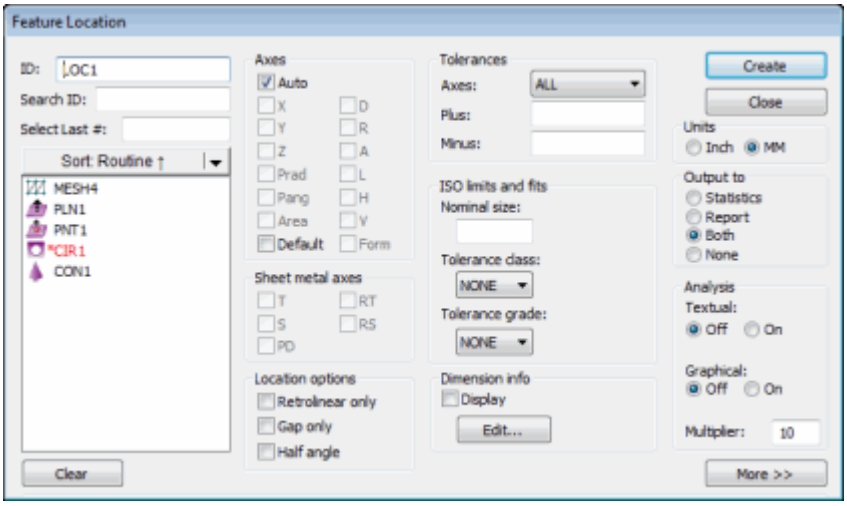
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Flatness

The **Plus** box in the **Flatness** dialog box (**Insert | Dimension | Flatness**) enables you to enter a tolerance value along the plus direction. This means that any flatness that differs from your nominal or theoretical flatness can still be a valid measurement as long as it falls within the tolerance range specified.

For information on how PC-DMIS reports tolerance zones for form dimensions, see "Reported Tolerance Zones for Form Dimensions" in the "Reporting Measurement Results" chapter.

Dimensioning Location



Dimension Feature Location Dialog Box

The **Insert | Dimension | Location** menu option calculates the distance from the feature to the X, Y, Z origin, parallel to its respective axis. The feature's diameter, angle, and vector are also part of the calculation. This section relates only to location or coordinate dimensioning. For position dimensions, see "Dimensioning Position".

Location can be calculated using Cartesian or Polar coordinates, Position or box tolerancing.

- To switch between Cartesian and Polar coordinates, select **Pang** or **Prad** in the **Feature Location** dialog box.
- To switch between `POSITION` and `RECT` tolerancing methods, see the "Dimensioning Position" dimension option.

By default, the dialog box marks the **Auto** check box. To change the default behavior for your measurement routine, see the "Default Axes for Location Dimensions" topic.

For legacy Form dimensions (Circularity, Cylindricity, Flatness, and Straightness), as well as a Location dimension's RN Line, PC-DMIS uses the feature solution to compute the dimension. By default, this is Least Squares. However, you can choose to solve the feature using Minimum Separation, Maximum Inscribed, Minimum Circumscribed, or Fixed Radius regression algorithms.

PC-DMIS computes Geometric Tolerance Form commands, on the other hand, using the Chebyshev algorithm (Min/Max) as required by the Y14.5 standard. Because of the change in calculation, PC-DMIS generally computes Geometric Tolerance Form dimension commands to a slightly smaller value than their legacy counterparts.

To Dimension a Feature Using the LOCATION Option

1. Select **Insert | Dimension | Location** from the submenu. The **Feature Location** dialog box appears.
2. Select the desired axes from the **Axes** area. The **Auto** check box is marked as the default.

   If you mark the **Auto** check box, PC-DMIS automatically determines the default axes to display in the dimension. The default axes are based on the feature type, as shown in the table below.

   The default output format for the following features is:

   CIRCLE = X, Y, D (based on the workplane).

   CONE = X, Y, Z, A

   CYLINDER = X, Y, Z, D, L (based on the workplane).

   ELLIPSE = X, Y, Z, D, A, L

   LINE = Based on the axis that is perpendicular to the axis closest to (and the axis related to) the workplane that the line is measured in.

   PLANE = Based on the axis most closely related to the plane.

   POINT = X, Y, Z, T

   SLOT = X, Y, D, R, L (based on the workplane).

   SPHERE = X, Y, Z, D
3. Select the feature(s) to dimension from the **Feature List** box.
4. Select the axes from the **Axes** drop-down list to which you want to apply Plus and Minus tolerances.
5. Type the Plus tolerance value in the **Plus** box.
6. Type the Minus tolerance value in the **Minus** box.
7. Type the Nominal size value in the **Nominal Size** box.
8. Select a Tolerance Class from the **Tolerance Class** drop-down list.
9. Select a Tolerance Grade from the **Tolerance Grade** drop-down list.
10. Select either the **Inch** or **MM** option from the **Units** area of the dialog box.

11. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both,** or **None** option.
12. Select the desired Analysis options by selecting the **Textual** check box or the **Graphical** check box. If you select the **Graphical** check box, type the Multiplier value in the **Multiplier** box.
13. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format that you would like to display in the Graphic Display window.
14. Click the **Create** button. If you did not select any check boxes from the **Axes** area, the **Create** button is unavailable for selection.

The dimension appears in the Edit window with this information:

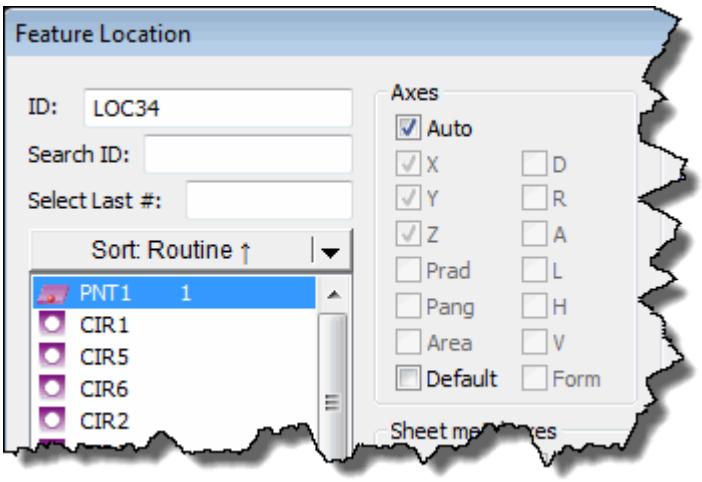
```
dimension_name = TYPE OF DIMENSION,feat_1 UNITS=IN ,$
GRAPH=OFF TEXT=OFF MULT=1.00 OUTPUT=BOTH
```

	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
x	8.0000	0.1000	0.1000	8.0000	8.5000	7.5000	0.0000	0.0000
y	3.0000	0.1000	0.1000	3.0000	3.5000	2.5000	0.0000	0.0000
z	0.4947	0.1000	0.1000	0.4947	0.1428	0.8466	0.0000	0.0000
d	1.0000	0.1000	0.1000	1.0000	1.0000	1.0000	0.0000	0.0000
v	0.0000	0.5938	0.8046	-	-	-	-	-

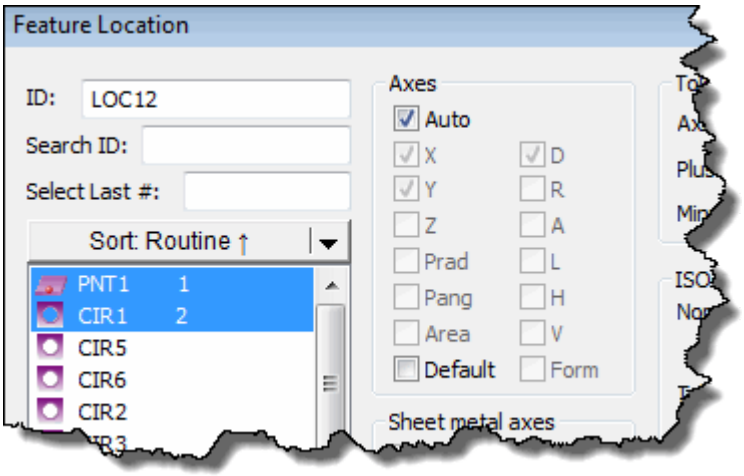
`END OF DIMENSION dimension_name` (for LOCATION only)

Default Axes for Location Dimensions

When the **Auto** check box in the **Axes** area in the **Feature Location** dialog box (**Insert | Dimension | Location**) is checked, the check boxes for the axes that appear in the dimension are selected according to the default axes of the feature type. In this case, the check boxes for those axes are not available to edit:



If you select multiple features of different types (for example a circle feature and a point feature), the **Axes** area selects the axes to use based on the last-selected feature:



By default, PC-DMIS marks the **Auto** check box. But if you want to choose your own axes, you can clear **Auto** and choose them manually. If you want to then save your selection as the new default for this routine, you can mark **Default** and create your dimension. The next time you open the **Feature Location** dialog box in that routine, PC-DMIS uses your defaults.

Changing the Default Axes

In some circumstances, it may be necessary to override the default setting. The **Default** check box remains available so that you can alter the format of the default output.

To alter the default output:

1. Select each feature to dimension.
2. Select the **Default** check box. The **Auto** check box clears, and all axes become available for selection.
3. Select the appropriate axes as needed.

X = Prints the X axis value.

Y = Prints the Y axis value.

Z = Prints the Z axis value.

Prad = Prints the Polar Radius.

Pang = Prints the Polar Angle.

Area = Prints the area of a selected Blob feature. It appears as AR in the report and in the Edit window in Command Mode.

D = Prints the Diameter value. For an ellipse, this is the minor diameter value (same as H).

R = Prints the Radius (half of the Diameter) value.

A = Prints the Angle value.

L = Prints the Length (used for cylinder, cones, slots, and ellipses). For an ellipse, L gives you the major diameter value.

H = Prints the Height (used on cones, cylinders, and ellipses). For an ellipse, H gives you the minor diameter value.

V = Prints the Vector location.

Form = Prints the feature's integrated form dimension with the location dimension.

- For a circle, cylinder, or cone feature, this is the Roundness (RN) dimension.
- For a plane feature, this is the Flatness (FL) dimension.
- For a line feature, this is the Straightness (ST) dimension.

4. Click the **Create** button.

Once the output values have been changed to something other than the default, PC-DMIS uses the new settings for all subsequent dimensions.

Resetting to the Original Axes

If you modified the default axes, but you want PC-DMIS to use the original default axes, you must reset the format to the default settings.

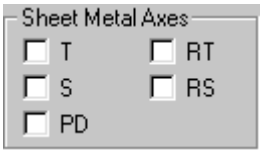
To reset the format to the default settings:

1. Select the **Default** check box.
2. Select the **Auto** check box. All of the axes check boxes become unavailable.
3. Select each feature to dimension. The original default axes are selected.

4. Click the **Create** button.

PC-DMIS automatically resets the dimension to print the default axes according to the feature type.

Sheet Metal Axes



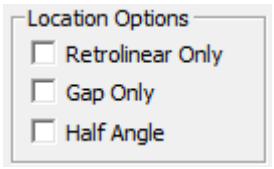
The **Sheet Metal Axes** area in the **Feature Location** dialog box (**Insert | Dimension | Location**) contains check boxes only available when sheet metal features (Auto Features) are being dimensioned.

T = prints the deviation along the approach vector (for points on curved surfaces).
S = prints the deviation along the surface vector.
RT = prints the deviation along the report vector.
RS = prints the deviation along the surface report.
PD = prints the diameter of a circle (perpendicular to the pin vector).

If a **T**, **RT**, **S**, or **RS** axis is used, none of the other location dimensions is used for the analysis (graphical or textual).

For Location dimensions of Vector Point, Surface Point, Edge Point, and Angle Point features, the default **Sheet Metal Axes** output format is **T**. Location dimensions of all the other Auto feature types do not initially have any of these axes selected.

Location Options



PC-DMIS also allows you to show these specialty axes:

- Retrolinear Only
- Gap Only
- Half Angle

These options can be selected in the **Locations Options** area in the **Feature Location** dialog box (**Insert | Dimension | Location**). All future locations of vector, surface, and

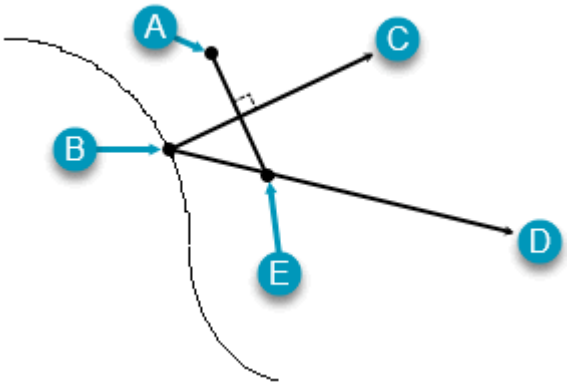
edge points within the measurement routine is printed using the appropriate specialty axes until the option is turned OFF.

Retrolinear Only for Location Dimensions

This check box is available when dimensioning vector and surface points. When the retrolinear only option has been selected for valid points, the location axes are calculated by:

- Finding the largest component of the theoretical surface normal vector (largest in x, y, or z direction).
- Projecting the measured point to this largest component vector in a manner that the projection is perpendicular to the original theoretical surface normal vector.

The location axes are then calculated from this new projected point.



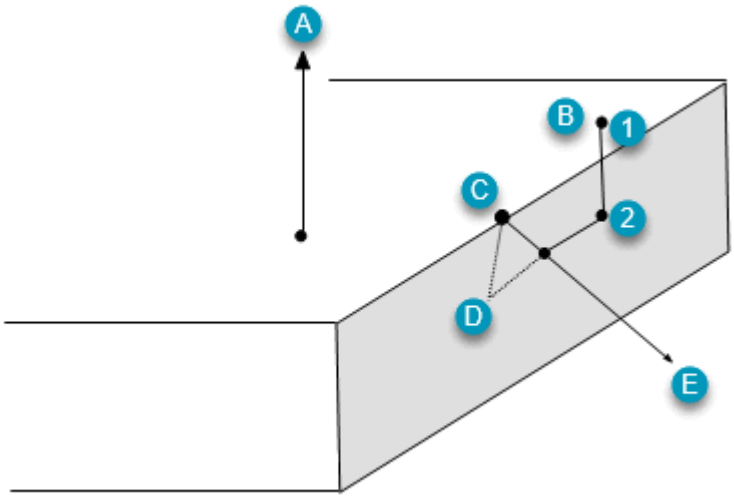
A - Actual Measured Point
B - Theoretical Point
C - Theo Surface Normal Vector
D - Largest component of the Theo Surface Normal Vector
E - Projected Point (reported point)

Gap Only for Location Dimensions

The **Gap Only** check box is available when dimensioning edge points. When you select the **Gap Only** check box, and an edge point is measured, the location axes are calculated by:

- Projecting the measured point (1) to the theoretical surface.
- Projecting this new point on to the theoretical approach vector.

Any location axes are then calculated from this new point.



A. *Theoretical Surface Normal*

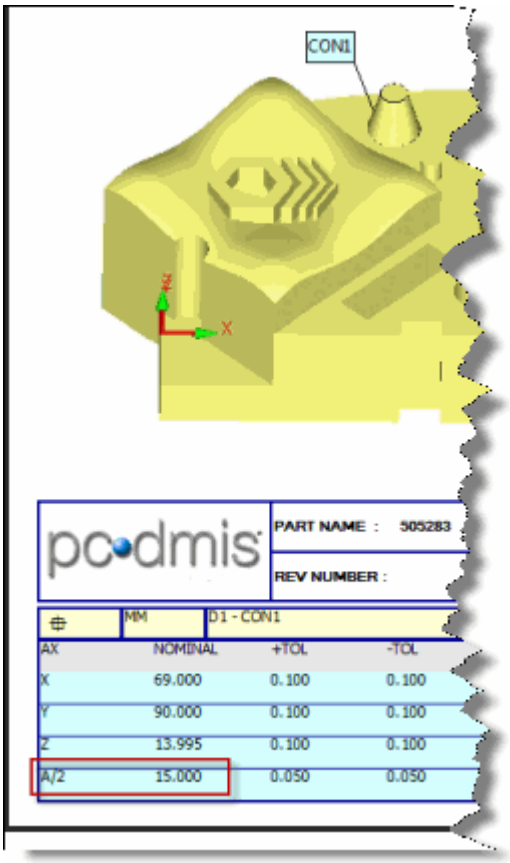
B. *Measured Point*

C. *Theo Point*

D. *Gap*

E. *Theoretical Approach Vector*

Half Angle for Location Dimensions



Example of a cone dimension with half-angle.

The **Half Angle** check box lets you display the angle axis (the **A** axis in the **Axes** area) at half its usual value. When you select this check box, the **A** axis in the **Axes** area changes to display **A/2**. In addition, the expanding grid area in the **Feature Location** dialog box and the dimension in the Edit window, Report window, and Diminfo boxes will also change to display the angle at half its value with label of A/2.

Tolerances for Location Dimensions

The **Feature Location** dialog box (**Insert | Dimension | Location**) provides you with two methods of entering Plus and / or Minus tolerances for these available axes:

ALL = All the axes and options displayed in the **Axes** drop down list in the **Tolerances** area.

X = X axis value

Y = Y axis value

Z = Z axis value

D = Diameter

R = Radius (half the Diameter)

A = Angle (for Cones)

L = Length (used for cylinders, slots, cones and ellipses)

H = Height

PR = Polar Radius

PA = Polar Angle

AR = Area (for Blob features)

T = Error along the approach vector (for points on curved surfaces)

RT = Deviation along the Report vector

S = Deviation along the Surface vector

RS = Deviation along the Surface report

PD = Diameter of a circle (perpendicular to the pin vector)

FORM = Feature's integrated form dimension.

- For a circle or cylinder feature, this is the Roundness (RN) dimension.

- For a plane feature, this is the Flatness (FL) dimension.
- For a line feature, this is the Straightness (ST) dimension.

1) Old Method - Use the Tolerances Area

The **Tolerances** area allows you to type plus and minus tolerances for each of the following axes found in the **Axes** drop-down list.

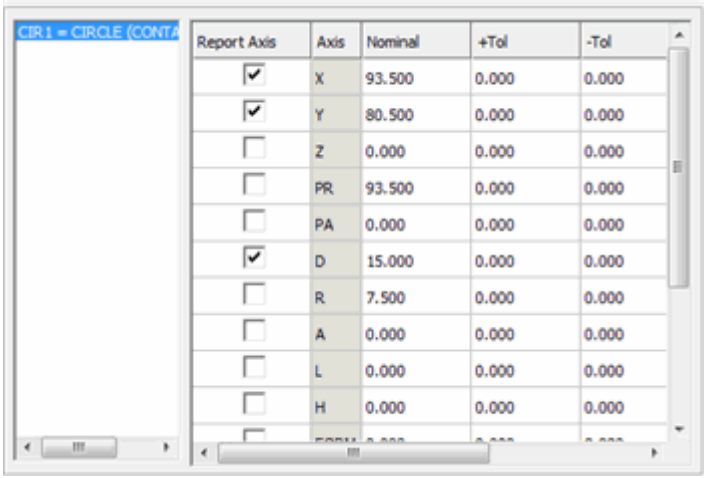
To enter in plus and minus tolerances:

1. From the **Axes** drop-down list, select the axis that you want to edit.
2. In the **Plus** box, type the plus tolerance value for the selected axis.
3. In the **Minus** box, type the Minus tolerance value for the selected axis.
4. Repeat the above steps for any axes with tolerance values.

2) New Method - Use the Expanding Grid

When you select a feature from the main feature list and click the **More>>** button from the **Feature Location** dialog box, the dialog box expands vertically to provide you with a grid. You can use this grid to type in the values to specify the tolerance and nominal values for the desired axes. In addition, an **Update Feature Nominal** column lets you update the associated feature's nominal values as desired.

PC-DMIS shows a blank grid unless you select one or more features from the dialog's main **Feature List** box.



List of Features - The list to the left of the grid's columns contains the feature or features you selected from the dialog's main **Feature List** box. When you select a feature from the

list, PC-DMIS displays the possible axes for that feature which you can then apply tolerance values.

Report Axis - This column contains check boxes for each axis. A selected check box sends that axis to the report.

Axis - This column lists available axes for the selected feature.

Nominal - This column contains nominal values. You can click each one and type a different value as needed.

+Tol / -Tol - The plus and minus tolerance columns provide fields into which you can type tolerances for the various axes used in your dimension.

Update Feature Nominal - This column lets you specify whether or not any changes you made to the nominal values should get applied not only to the dimension but to the feature as well.

Entering Positive-lower and Negative-upper Tolerances

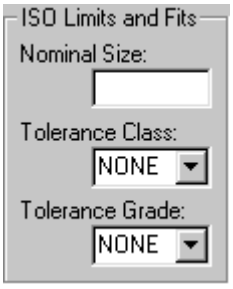
PC-DMIS allows positive-lower tolerances (or a positive tolerance in the negative range). To do this, add a minus sign before the value in the **Minus** box.

PC-DMIS also allows a negative-upper tolerance (or a minus tolerance in the positive range). This is done by entering in a minus sign before the value in the **Plus** box.

If 1.000 is entered for the nominal, .003 is entered for the plus tolerance and .001 is entered for the minus tolerance it would be interpreted as 1.000 + .003 / –.001. To get the lower tolerance to be +.001 (sign change) simply type in 1.000 (nominal), .003 (plus tol) and –.001 (minus tol) thereby achieving a tolerance range of 1.000 + .003 / + .001.

Negative lower tolerances may be displayed with a minus sign if the **Minus Tols Show Negative** option has been selected. See "Minus Tols Show Negative" in the "Setting Your Preferences" chapter.

ISO Limits and Fits



The **ISO Limits and Fits** area in the **Feature Location** dialog box (**Insert | Dimension | Location**) allows you to apply ISO standardized tolerances to the diameter of circular features. When a tolerance class and grade have been selected, PC-DMIS will look up the appropriate tolerances for the diameter of a cylinder or circle from the International Organization for Standardization's tables of Limits and Fits. These tables define the tolerances for various design classes and grades of diameters.

Although PC-DMIS allows the calculation of ISO tolerances in English (Inch) and metric (mm) units, they are only defined by ISO to be applied with metric (mm) units. Also, because these tables use various classes and grades of diameters, cylinders and circles are the only appropriate features for this option. If no nominal diameter is entered, PC-DMIS uses the feature's nominal diameter for the tolerance calculations.

Nominal Size

The **Nominal Size** box in the **Feature Location** dialog box (**Insert | Dimension | Location**) enables you to enter the nominal diameter of the feature selected.

Tolerance Class

The **Tolerance Class** list in the **Feature Location** dialog box (**Insert | Dimension | Location**) enables you to select from the following ISO Tolerance classes:

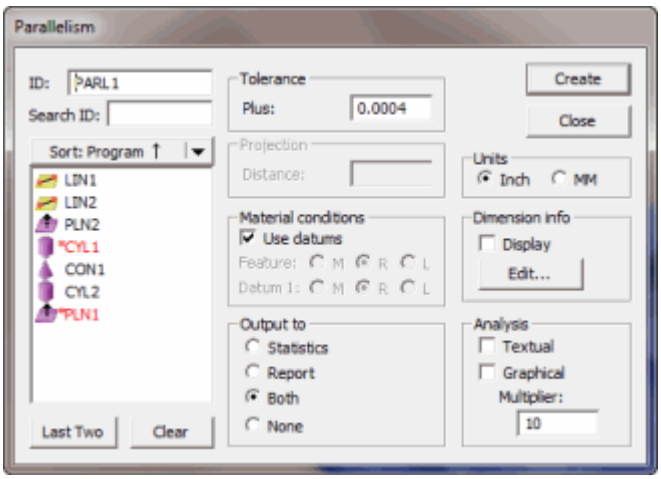
A B C CD D E EF F FG G	H J JS K M N P R S	T U V X Y Z ZA ZB ZC

Tolerance Grade

The **Tolerance Grade** list in the **Feature Location** dialog box (**Insert | Dimension | Location**) enables you to select from the following ISO Tolerance grades:

IT1	IT7	IT13
IT2	IT8	IT14
IT3	IT9	IT15
IT4	IT10	IT16
IT5	IT11	IT17
IT6	IT12	IT18

Dimensioning Parallelism



Dimension Parallelism Dialog Box

The **Insert | Dimension | Parallelism** menu option measures the parallelism between two features. The second feature is always the datum feature. When only one feature is selected, the current workplane becomes the datum feature. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

The evaluation of parallelism is 3-dimensional regardless of the workplane or the feature being dimensioned.

To perform a 2-dimensional evaluation, or when you require more advanced support, (for example, when you need to reference multiple datums or you want to add the tangential plane modifier), you must use the newer Geometric Tolerance command. For details, see the "Using Geometric Tolerances" chapter of the PC-DMIS Core documentation.

To Perform a Dimension Using PARALLELISM:

1. Select **Insert | Dimension | Parallelism** from the submenu. The **Parallelism** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. If a datum feature is desired, select the **Use Datums** check box.
4. If a datum feature is desired, select another feature in the **Feature List** box.
5. Select the appropriate **Material Condition** options for the feature(s) and datum.
6. Type the plus tolerance value in the **Plus** box.
7. Type the projection distance in the **Distance** box.
8. Select either the **Inch** or **MM** option in the **Units** area.
9. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
10. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
11. Select the desired analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, enter the Multiplier value in the **Multiplier** box.
12. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
13. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = PARALLELISM,OF feat_1,TOG1,TO feat_2,TOG1
```

or

```
dimension_name = PARALLELISM,OF feat_1,TOG1,
```
TO workplane

AX	NOM	+TOL	-TOL	BON	MEAS	MAX	DEV	OUT-TOL
DF	1.0000	0.0100	0.0100	0.0000	1.0000		0.0000	0.0000
D1	0.0000	0.0100	0.0100	0.0000	0.0000	0.0000	0.0000	0.0000
M	0.0000	0.0100	0.0100	0.0000	0.0000		0.0000	0.0000

TOG1= MMC / RFS / LMC

Explanation of Axes

DF is the Diameter or Width of the feature. This appears when you set LMC or MMC conditions.

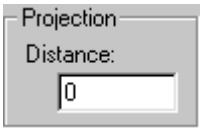
D1 is the Diameter or Width of the first datum. This appears when you set a datum feature.

M is the measured value. This always appears.

Plus Tolerance for Parallelism Dimensions

The **Plus** box in the **Parallelism** dialog box (**Insert | Dimension | Parallelism**) enables you to enter a tolerance value along the plus direction. This means that any parallelism that differs from your nominal or theoretical parallelism can still be a valid measurement as long as it falls within the tolerance range specified.

Projected Distance for Parallelism Dimensions



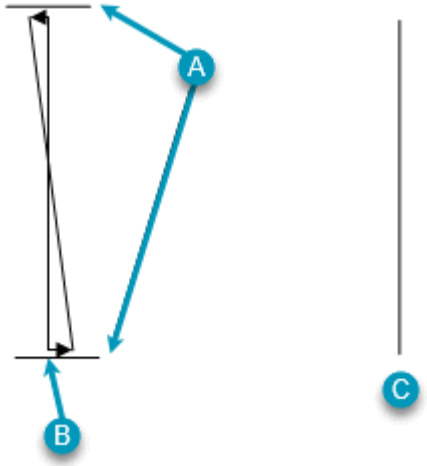
PC-DMIS allows you to type a reference distance in the **Distance** box in the **Parallelism** dialog box (**Insert | Dimension | Parallelism**). PC-DMIS uses this value with cylinder, cone, and line features to calculate the parallelism dimension.

For these features, the points used for the parallelism calculation are the end point of the axis and a point projected from this end point along the feature axis.

The distance between these two points is the referenced distance. For other features, this distance has no effect on the dimension. This option is useful when the dimension needs to be calculated a certain distance along the feature.

Understanding Parallelism's Distance Projection

Usually a parallelism dimension takes the datum line's vector, and checks the endpoints of the dimensioned feature to see if they are within tolerance as shown here:

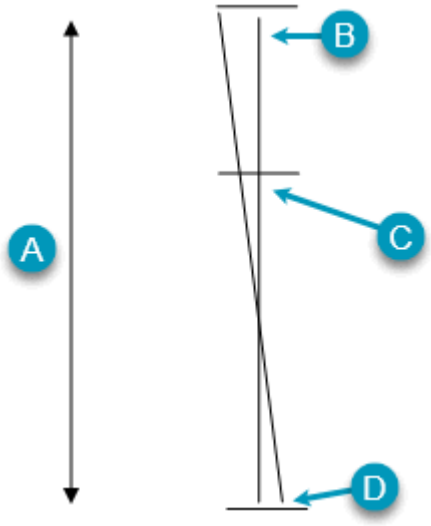


A - Tolerance width (or cylinder).
B - Datum vector positioned at measured feature.
C - Datum line.

Sometimes, however, you can't always measure right to the edge of the line, so you will need to extend the line out by a certain distance.

Since the extreme deviations are usually at the ends of the line, if you extend out the line to a distance longer than the actual length of the line you will usually get larger deviations. You can also extend the line's opposite direction by using a negative reference length.

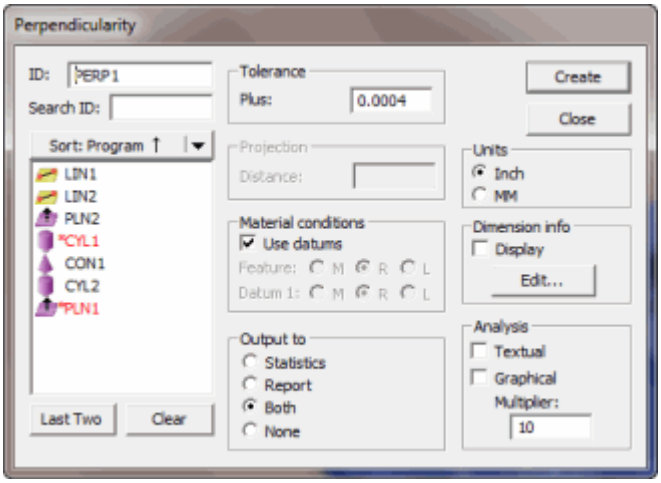
An extended line may look like this:



A - Reference Length
B - Extended point, from reference length greater than line length
C - End point of line, or reference length same as line length.
D - Start point of line, or 0.0 reference length.

You can also use the **Distance** box to specify a reference length to check the axis of a cylinder.

Dimensioning Perpendicularity



Dimension Perpendicularity Dialog Box

The **Insert | Dimension | Perpendicularity** menu option measures the perpendicularity between two features. The second feature is always the datum feature. When you select only one feature, the current workplane becomes the datum feature. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

When you require more advanced support, (for example, when you need to reference multiple datums or you want to add the tangential plane modifier), you must use the newer Geometric Tolerance command. For details, see the "Using Geometric Tolerances" chapter of the PC-DMIS Core documentation.

To Dimension a Feature Using the PERPENDICULARITY Option:

1. Select **Insert | Dimension | Perpendicularity** from the submenu. The **Perpendicularity** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Select the appropriate **Material Condition** options for the feature(s) and datum.
4. If a datum feature is desired, select the **Use Datums** check box and select another feature in the **Feature List** box.
5. Select the appropriate **Material Condition** options for the feature(s) and datum.
6. Type the plus tolerance value in the **Plus** box.
7. Enter the projected distance in the **Distance** box.
8. Select either **Inch** or **MM** in the **Units** section.

9. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
10. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
11. Select the desired Analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, type the Multiplier value in the **Multiplier** box.
12. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
13. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = PERPENDICULARITY,OF feat_1,TO feat_2
```

or

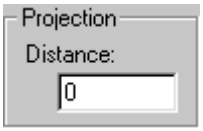
```
dimension_name = PERPENDICULARITY,OF feat_1,
```
TO workplane

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Perpendicularity Dimensions

The **Plus** box in the **Perpendicularity** dialog box (**Insert | Dimension | Perpendicularity**) enables you to enter a tolerance value along the plus direction. This means that any perpendicularity that differs from your nominal or theoretical perpendicularity can still be a valid measurement as long as it falls within the tolerance range specified.

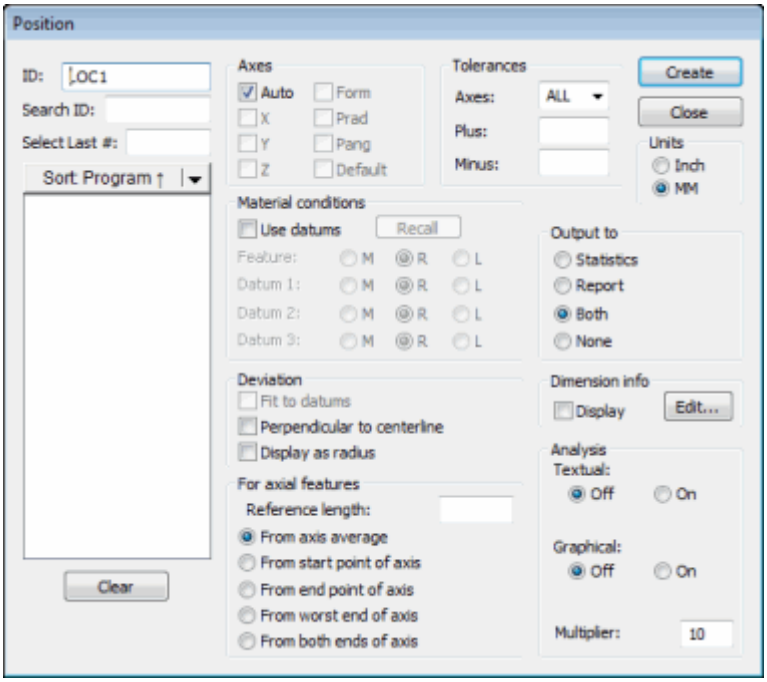
Projected Distance for Perpendicularity Dimensions



PC-DMIS allows you to type a reference distance in the **Distance** box in the **Perpendicularity** dialog box (**Insert | Dimension | Perpendicularity**). This option is

useful when the feature not only needs to be perpendicular to the datum feature, but needs to be calculated a certain distance away from the datum as well.

Dimensioning Position



Dimension Position Dialog Box

The **Insert | Dimension | Position** menu option calculates the position from the feature to the X, Y, or Z origin parallel to its respective axis. The feature's diameter, angle, and vector are also part of the calculation.

The **Position** dialog shown above is available only when the **Use Legacy Dimension** option is selected from the **Insert | Dimension** menu.

This section relates only to *Position* dimensions. Location or coordinate dimensioning is discussed in the "Dimensioning Location" section.

If datums are used, the measured and deviation XYZ values are calculated with respect to the alignment of the datums, but are displayed in the current alignment in order to interpret the values. This means that a feature reported with one position dimension can have different measured and deviation values than another position dimension if the dimensions have different or no datums defined, even if they have the same nominal values.

To Dimension a Feature Using the POSITION Option

1. Select **Insert | Dimension | Position** from the submenu. The **Position** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. If a datum is desired, select the **Use Datums** check box. Be aware that selecting this will cause the dimension to be calculated with respect to the datums. However, the XYZ output values will be displayed with respect to the current alignment in the measurement routine.
4. Select any desired datum features from the **Feature List** box.
5. Select the appropriate options in the **Material Conditions** area to set the Material Conditions for the feature(s) and datum(s).
6. Select the appropriate check boxes from the **Deviation** area.
7. If you are dimensioning an axial feature (such as a cylinder), type a value in the **Reference Length** box and select the appropriate option from the **For Axial Features** area.
8. Select the desired axes from the **Axes** area. The software selects the **Auto** check box by default.
9. Select the axes to which you want to apply plus and minus tolerances.
10. Type the plus tolerance value in the **Plus** box.
11. Type the minus tolerance value in the **Minus** box.
12. Select either the **Inch** or **MM** option in the **Units** area.
13. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both,** or **None** option.
14. Select the desired analysis options by marking the **Textual** check box or the **Graphical** check box. If you selected the **Graphical** check box, type the Multiplier value in the **Multiplier** box.
15. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
16. Click the **Create** button. If you did not select any check boxes from the **Axes** area, the **Create** button is not available for selection.

PC-DMIS shows the dimension in the Edit window with this information:

```
DIM dimension_name = POSITION OF FEAT_ID UNITS = IN/MM,
 GRAPH=ON/OFF TEXT=ON/OFF MULT=n OUTPUT=REPORT/STATS/BOTH/
NONE FIT TO DATUMS=ON/OFF
 DEV PERPEN CENTERLINE=ON/OFF DISPLAY=DIAMETER/RADIUS
```

AX	NOM	MEAS	+TOL	-TOL	BON	DEV	DEVANG	OUTTOL
X	0.7500	.07500	0.0000	-	-	0.0000	-	-
Y	3.0000	3.0000	0.0000	-	-	0.0000	-	-
DF	1.0000	1.0000	0.0000	0.0000	0.0000	0.0000	-	0.0000
D1 D2 D3 TP	1.0000 1.0000 MMC	1.0000 1.0000 1.0000	0.0000 0.0000 0.0000	0.0000 0.0000	0.0000 0.0000 0.0000	PLANE CIRCLE CIRCLE 0.0000	P1 AT RFS C1 AT MMC C2 AT MMC 0.000	0.0000

```
END OF dimension_name
```

Using Datums

Although there are many different combinations of features that may be used as datums for Position dimensions, there are some general principles to follow when selecting datum features. One common scenario for designs that utilize Position dimensions is to use a circle or cylinder as the sole datum feature.

Another accepted practice is to select a set of datum features that follow 3-2-1 alignment principles. (Remember, the minimum definition for a datum is 3 datum points to describe the first datum, 2 datum points to describe the second datum and 1 datum point to describe the third datum.) This means that the selected features would be a plane, a line, and then a single point.

However, 3-2-1 alignment principles can also be used with plane/line/line, plane/line/circle, plane/cylinder/cylinder, and many other combinations.

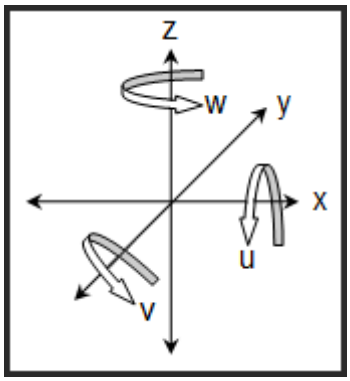
When multiple circular features have been used with the Maximum Material Condition (MMC) or Least Material Condition (LMC),

- The bonus tolerance from the TP axis will not likely be a simple addition of other bonus tolerances. This is because a best-fit calculation determines the correct bonus tolerance.
- Any circular or slot features may be measured using MMC or LMC.
- All non-circular features are measured Regardless of Feature Size (RFS).

When datum features have been selected, the Position's X,Y,Z, PA, and PR axes are calculated with respect to the alignment of the datums, but are displayed in the current alignment in order to interpret the values. Also, the datum features must be selected using the general steps discussed in this topic or else the dimension is likely to give unexpected results.

General Rules for Position Dimensions when Use Datums is Selected:

- Select all the datums specified in the feature control frame so that proper fitting is performed. The features selected for datum 1, datum 2, and datum 3 represent the Primary, Secondary, and Tertiary Datums and are used to constrain up to *six degrees of freedom* (3 degrees of translation and 3 degrees of rotation).



Example of the six degrees of freedom in 3D space (X,Y,Z,U,V, and W).

- Ensure that all Measured Feature commands (both datums and features) contain the correct nominal values (X,Y,Z,I,J,K) in the THEO field. (The fitting process references the measured feature commands to calculate datum constraints and the dimensional results.)
- The Measured Feature command and the associated Position Dimension command must come from the same alignment to ensure that the nominal values are correct and the same as what is called out as basic dimensions on the drawing. This is critical when programming without CAD as it will require editing measured feature commands (Guess Mode) to provide the correct nominal values (used for fitting).
- The datums must be mutually perpendicular to each other.
- DO NOT use the **Ignore CAD to Part** option on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**).

For more complex datum reference frames built from combinations of non-orthogonal features or when there are modifier(s) (MMC or LMC) on the datums, we recommend that you use the Geometric Tolerance Position method of dimensioning Position. The legacy Position command with **Use Datums** selected is available for measurement routine migration.

Bonus Refinement

When calculating the bonus on a datum, if there is a previous Position dimension or orientation dimension on that datum feature, then the Position and/or orientation tolerance is added to the datum bonus to calculate the virtual size of the datum.

Using the Same Datums when Repeating Dimensions

To use the same datums from a previous Position dimension in your current Position dimension, select the Feature to dimension, and click the **Recall** button available in the **Material Conditions** area in the **Position** dialog box (**Insert | Dimension | Position**). See "Recall Button".

Examples of Use Datums with Legacy Position Dimension

The **Use Datums** option in the legacy Position command was created over 20 years ago and its behavior is based on the best interpretation of the ASME Y14.5 standard at that time. Standards continually evolve and, over the years, there have been many changes, amendments and improvements, particularly on the subject of datum fitting, bonus calculation and, more recently, full mathematical definitions for datum feature calculations.

The legacy position command does not take any of these changes into consideration and as such, does not fully comply with the requirements of any GD&T standard. It is maintained primarily to support legacy program migration and, although it can be useful for providing process information, we strongly advise using the Geometric Tolerance Position method of dimensioning instead – particularly in these cases:

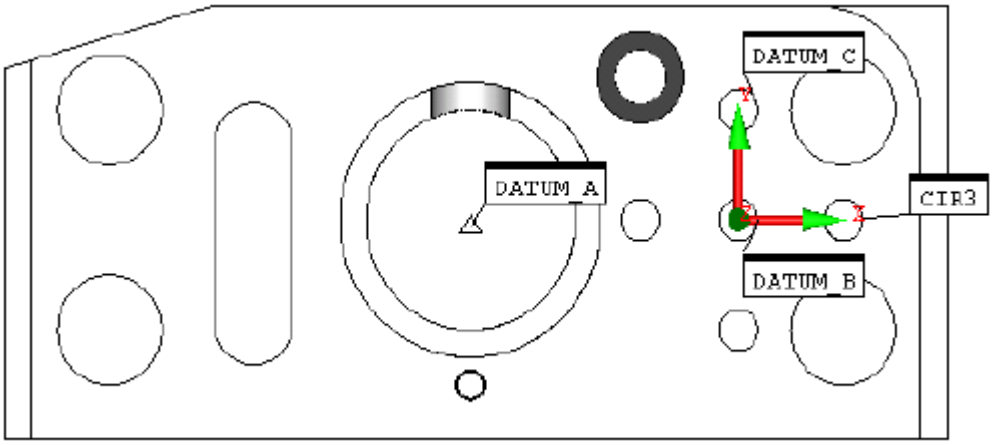
- Where there are material modifiers on datums that require PC-DMIS to perform datum shift calculations on.
- Where PC-DMIS needs to evaluate multiple position (and profile) dimensions simultaneously, since legacy dimensioning does not support simultaneous evaluations.

For details on the Geometric Tolerance Position method, see the "Position" topic in the "Using Geometric Tolerances" chapter of the PC-DMIS Core documentation.

The **Use Datums** option in the **Position** dialog box (**Insert | Dimension | Position**) enables you to perform the analysis in three ways.

- From the active alignment. See Option #1 below
- From a mathematical Virtual Hard Gage simulation (Fit to Datums: ON). See Option #2 below
- From a Datum Reference Frame simulation (Fit to Datums: OFF). See Option #3 below

These are discussed in turn. All examples reference the illustration below (Features, Datums, and Alignment):

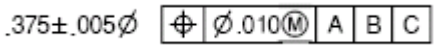


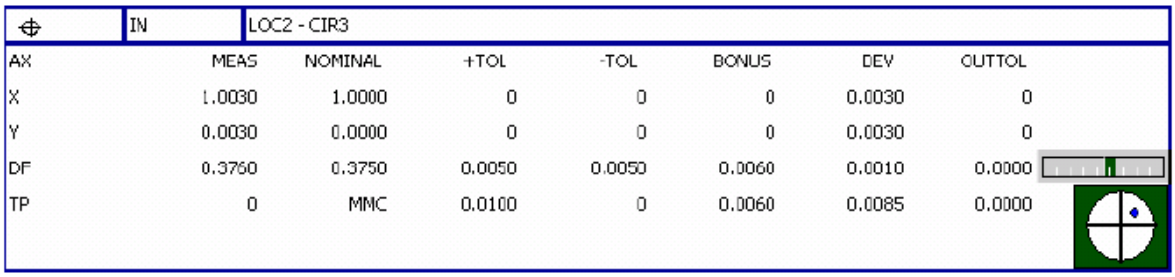
Option #1: Use Datums: OFF

Application: Use this method when there are no modifier(s) (MMC or LMC) on the datum(s) and when checking the Position of one or multiple features (creates a single position command for each selected feature) from the currently active alignment. Bonus tolerance is only available on the feature(s).

Results: The Position of the selected feature(s) is evaluated in the active alignment. Therefore, the active alignment must be set up to reflect the specified Datum Reference Frame (DRF) before creating the Position dimension(s).

You can see the affect of setting **Use Datums** to Off in the images below:





	IN	LOC2 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0030	1.0000	0	0	0	0.0030	0
Y	0.0030	0.0000	0	0	0	0.0030	0
DF	0.3760	0.3750	0.0050	0.0050	0.0060	0.0010	0.0000
TP	0	MMC	0.0100	0	0.0060	0.0085	0.0000

Option #2: Use Datums: ON and Fit to Datums: ON

Application: Use this method when:

- There are modifier(s) (MMC or LMC) on the datum(s)
- Those datums are mutually perpendicular to each other and...
- When checking the Position of a single feature (multiple features are not supported in legacy dimension mode).

Position of multiple features are supported when you use the newer Position Geometric Tolerance method of dimensioning if they meet these requirements:

- MMC/LMC is applied to the Datum feature(s)
- Simultaneous evaluations of multiple position tolerances
- The use of complex datum reference frames built from non-orthogonal features

For details on the latest Geometric Tolerance methods, see the "Using Geometric Tolerances" chapter in the PC-DMIS Core documentation.

Results: A "Virtual Hard Gage" simulation is performed mathematically by translating and rotating based on the amount of bonus tolerance from the datum(s). This results in the measured values for the feature changing by the amount of "datum shift" which simulates the jiggling of the gage. In many cases, the measured values may be the same as the nominal values when there is sufficient bonus from the datum features. When the feature is out of tolerance, no fitting occurs, and the measured values and deviations reflect the actual location of the feature such that process adjustments can be made or an engineering analysis of the nonconformance can be performed.

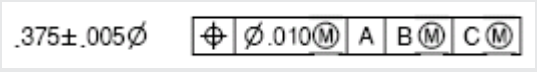
The results are representing "Pass/Fail" analysis just as a functional gage does; therefore, it is not possible to monitor for process variation or perform statistical studies.

Bonus Column: The bonus column in the report shows the calculated amount of bonus of the feature (DF) and the calculated amount of bonus of each datum feature of size (D1 primary, D2 secondary, D3 tertiary). The total bonus value is determined based on the following conditions:

Condition #1A: In-Tolerance

When sufficient bonus tolerance from the datum(s) allows datum shift such that there is no deviation from the nominal values without utilizing up to 100% of the allowable bonus from the datum(s), the total bonus amount is the sum of the bonus from the feature and the unused amount of bonus from the datum(s).

Example of Condition #1A In-Tolerance:



The measured values changed based on the allowable datum shift derived from the bonus of datums D2 and D3 (simulates a functional gage). In this case the measured values check nominal resulting in zero deviation, and the total bonus is the sum of the bonus from the feature (0.006) and the unused bonus from the datums (0.002) for a total of 0.008 bonus tolerance.

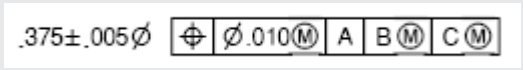
- Two degrees of freedom from bonus on D2 (0.008 bonus): translation in X and Y axis
- One degree of freedom from bonus on D3 (0.008 bonus): rotation about Z axis

⌖	IN	LOC1 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0000	1.0000	0	0	0	0.0000	0
Y	0.0000	0.0000	0	0	0	0.0000	0
DF	0.3760	0.3750	0.0050	0.0050	0.0060	0.0010	0.0000
D1	0	0	0	0	0	PLANE	
D2	0.3780	0.3750	0.0050	0.0050	0.0080	CIRCLE	
D3	0.3780	0.3750	0.0050	0.0050	0.0080	CIRCLE	
TP	0	MMC	0.0100	0	0.0080	0.0000	0.0000

Condition #1B: In-Tolerance

When 100% of the bonus from the datum(s) is used then the total bonus amount is only the bonus from the feature. Typically this case would show some amount of deviation from the nominal values.

Example of Condition #1B In-Tolerance:



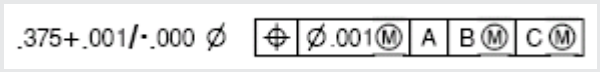
The tolerance for D2 and D3 were changed to illustrate using 100% of the bonus from the datums where there is deviation from nominal yet still an In-Tolerance condition.

	IN	LOC4 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0015	1.0000	0	0	0	0.0015	0
Y	0.0000	0.0000	0	0	0	0.0000	0
DF	0.3760	0.3750	0.0050	0.0050	0.0060	0.0010	0.0000
D1	0	0	0	0	0	PLANE	
D2	0.3780	0.3750	0.0030	0.0000	0.0030	CIRCLE	
D3	0.3780	0.3750	0.0030	0.0000	0.0030	CIRCLE	
TP	0	MMC	0.0100	0	0.0060	0.0030	0.0000

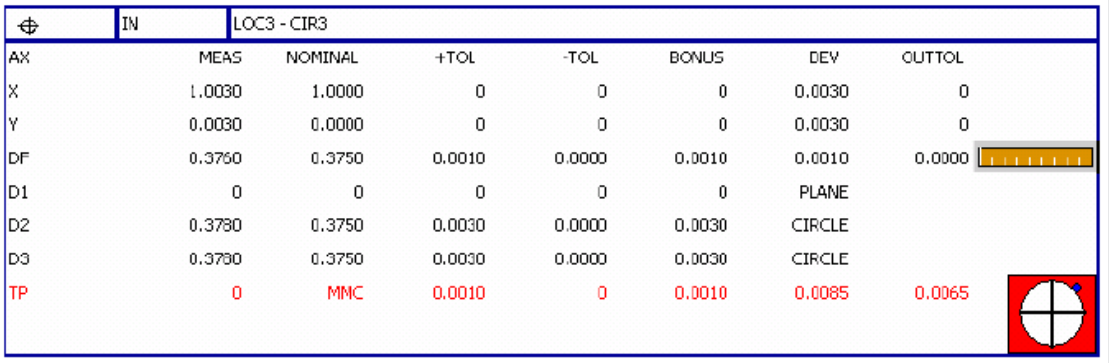
Condition #2: Out of Tolerance

When there is insufficient bonus from the datum(s) to allow datum shift such that the feature is In-Tolerance, no fitting is performed, the measured values are not altered and the total bonus is only the bonus from the feature.

Example of Condition #2 Out of Tolerance:



The tolerance for the DF, D2, D3 and the TP were changed to illustrate the out of tolerance condition.



	IN	LOC3 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0030	1.0000	0	0	0	0.0030	0
Y	0.0030	0.0000	0	0	0	0.0030	0
DF	0.3760	0.3750	0.0010	0.0000	0.0010	0.0010	0.0000
D1	0	0	0	0	0	PLANE	
D2	0.3780	0.3750	0.0030	0.0000	0.0030	CIRCLE	
D3	0.3780	0.3750	0.0030	0.0000	0.0030	CIRCLE	
TP	0	MMC	0.0010	0	0.0010	0.0085	0.0065

Option #3: Use Datums: ON and Fit to Datums: OFF

Application: Use this method when there are modifier(s) (MMC or LMC) on the Datum(s) and you do not want the measured data fitted (no datum shift). This option was added based on customer request to provide the ability to incorporate bonus from a Datum and still monitor for process variation (the measured values are not altered because no "datum shift" is applied).

This method is not compliant with ASME Y14.5 or ISO 1101.

Results: A "Datum Reference Frame" simulation is performed mathematically by Translating and Rotating based on the degrees of freedom constrained by the selected Datum(s). The X, Y, or Z measured values and deviations are from the active alignment and are not altered based on the bonus from the datum features (no datum shift).

Bonus Column: The bonus column in the report shows the calculated amount of bonus of the feature (DF) and the calculated amount of bonus of each datum feature of size (D1 primary, D2 secondary, D3 tertiary). The total bonus value is the sum of the bonus from the feature and the datum with the smallest amount of bonus amongst the selected datums.

Example of Condition #1 In-Tolerance:

.375±.005Ø | ⌖ | Ø.010Ⓜ | A | B Ⓜ | C Ⓜ

⌖	IN	LOC7 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0030	1.0000	0	0	0	0.0030	0
Y	0.0030	0.0000	0	0	0	0.0030	0
DF	0.3760	0.3750	0.0050	0.0050	0.0060	0.0010	0.0000
D1	0	0	0	0	0	PLANE	
D2	0.3780	0.3750	0.0030	0.0030	0.0060	CIRCLE	
D3	0.3780	0.3750	0.0050	0.0050	0.0080	CIRCLE	
TP	0	MMC	0.0100	0	0.0120	0.0085	0.0000

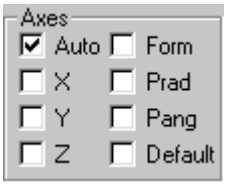
Example of Condition #2 Out of Tolerance:

.375+.001/-.000 Ø | ⌖ | Ø.001Ⓜ | A | B Ⓜ | C Ⓜ

The tolerance for the DF, D2, D3, and TP were changed to illustrate the out of tolerance condition.

⌖	IN	LOC5 - CIR3					
AX	MEAS	NOMINAL	+TOL	-TOL	BONUS	DEV	OUTTOL
X	1.0030	1.0000	0	0	0	0.0030	0
Y	0.0030	0.0000	0	0	0	0.0030	0
DF	0.3760	0.3750	0.0010	0.0000	0.0010	0.0010	0.0000
D1	0	0	0	0	0	PLANE	
D2	0.3780	0.3750	0.0030	0.0000	0.0030	CIRCLE	
D3	0.3780	0.3750	0.0050	0.0050	0.0080	CIRCLE	
TP	0	MMC	0.0010	0	0.0040	0.0085	0.0035

Default Axes for Position Dimensions



The **Default** check box in the **Position** dialog box (**Insert | Dimension | Position**) enables you to alter the format of the default output. When the **Auto** check box is checked, the axes that appear in the dimension are selected according to the default axes of the feature type. However, in some circumstances, it may be necessary to override the default setting. To do this, you would select the **Default** check box and any other desired axes.

To alter the output, select the desired check boxes:

Auto = Prints the default axes based on the feature type.

X = Prints the X axis value.

Y = Prints the Y axis value.

Z = Prints the Z axis value.

Form = Prints the integrated form dimension for the feature.

- For a circle or cylinder feature, this is the Roundness (RN) dimension.
- For a plane feature, this is the Flatness (FL) dimension.
- For a line feature, this is the Straightness (ST) dimension.

Pang = Prints the polar angle (PA) value.

Prad = Prints the polar radius (PR) value.

Default = Alters the format of the default output.

Once the output values have been changed to something other than the default, PC-DMIS uses the new settings for all subsequent dimensions. If you want PC-DMIS to then use the original default axes, you must reset the format to the default settings.

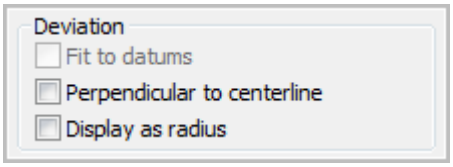
To reset the format to the default settings:

1. Select the **Auto** check box.
2. Select the **Default** check box.
3. Click the **Create** button.

PC-DMIS automatically resets the dimension to print the default axes according to the feature type.

If the **Auto** check box is marked, PC-DMIS automatically determines the default axes to print. The default output format is based on the feature type. PC-DMIS automatically determines the position of the selected feature using the documented (see box above) axis format.

Deviation



The **Deviation** area in the **Position** dialog box (**Insert | Dimension | Position**) enables you to select options that determine how you want deviations performed and how you want deviations displayed in the Edit window.

Fit to Datums

The **Fit To Datums** check box in the **Position** dialog box (**Insert | Dimension | Position**) determines whether PC-DMIS calls the fitting algorithm when datums are used.

- When selected, PC-DMIS calls the fitting algorithm when datums are specified. It reports the considered feature in the fitted position, with the total bonus being the remaining available bonus after fitting.
- When cleared, PC-DMIS analyses the Position dimension with the features in their measured position, with no fitting applied. The reported total bonus is the total available bonus from the considered feature and the datum based on the datum constraints.

For a detailed explanation of how the legacy PC-DMIS Position command behaves with different combination of Use Datums and Fit to Datums, see the "Examples of Use Datums with Legacy Position" topic in the PC-DMIS Core documentation.

Perpendicular to Centerline

The **Perpendicular to Centerline** check box in the **Position** dialog box (**Insert | Dimension | Position**) determines whether PC-DMIS calculates the deviation perpendicular to the feature's theoretical centerline or perpendicular to the direct X, Y, and Z values.

- If marked, PC-DMIS calculates the deviation perpendicular to the feature's theoretical centerline, ignoring any deviation in the direction of the centerline.

- If not marked, PC-DMIS calculates the deviation perpendicular to the X, Y, and Z values.

This check box is especially useful with sheetmetal points which may have approach vectors not oriented along the X, Y, or Z axis. This check box will allow the deviation to be found perpendicular to the approach vector.

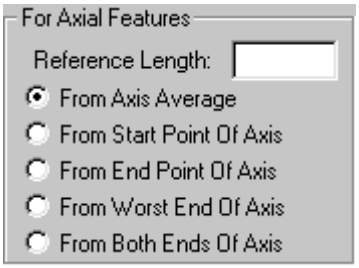
Display As Radius

The **Display As Radius** check box in the **Position** dialog box (**Insert | Dimension | Position**) determines whether or not PC-DMIS will display the radii of the features and datums instead of displaying their diameters.

- If selected, PC-DMIS displays the radii in the DF, D1, D2, D3, and TP lines and places text in the Edit window's dimension field showing whether the dimension is displaying the features' radius or diameter.
- If deselected, PC-DMIS displays the diameters as usual.

If you use this check box, be aware that it is *not* compliant with ISO standards.

Axial Features



For axial features, such as cylinders, you can specify where along the axis PC-DMIS should create the Position dimension. You can tell PC-DMIS to create the dimension at the axis' average (or centroid), at the axis' start point, at the axis' end point, at the axis' worst end (the end with the greatest deviation), or at both ends of the axis. You can also create the dimension at a specified reference length away from the selected point on the axis.

To use the reference length with a Position dimension:

1. Type the desired value in the **Reference Length** box in the **Position** dialog box (**Insert | Dimension | Position**).
2. Select one of the option buttons (**From Axis Average**, **From Start Point Of Axis**, **From End Point Of Axis**, **From Worst End Of Axis**, or **From Both Ends Of Axis**).

3. Click the **Create** button. PC-DMIS locates the selected point (or points if you've selected **From Both Ends Of Axis**) and then moves along the axis the distance of the specified reference length and creates the dimension(s) there.

   Reference Length
   The value in this box specifies the length from the selected point or axis end where PC-DMIS will create the Position dimension.

   From Axis Average
   This option tells PC-DMIS to dimension the axial feature, from the centroid (or average) of the axis.

   From Start Point of Axis
   This option tells PC-DMIS to dimension the axial feature from the Start Point of the axis.

   From End Point of Axis
   This option tells PC-DMIS to dimension the axial feature from the End Point of the axis.

   From Worst End of Axis
   This option tells PC-DMIS to dimension the axial feature from the start or end point that gives the worst-case scenario for the dimension. PC-DMIS creates the dimension at the axis end that deviates the farthest from the nominal. For example if your part contains a skewed cylinder, the start or end point along the cylinder's axis that deviates farthest from the nominal is the point that gives the worst-case scenario.

   From Both Ends of Axis
   This option tells PC-DMIS to dimension the axial feature, from both the start and end point of the axis selected. PC-DMIS creates a dimension at both ends of the axis.

Tolerances for Position Dimensions

The **Tolerances** area in the **Position** dialog box (**Insert | Dimension | Position**) enables you to type plus and minus tolerances for each axis found in the **Axes** drop-down list; it also lets you define certain material condition modifiers.

To enter in axes tolerances:

1. From the **Axes** list, select the axis that you want to apply the tolerance.

2. In the **Plus** box, enter the plus tolerance value for the selected axis.

3. In the **Minus** box, enter the minus tolerance value for the selected axis. Since the TP and FORM axes require a plus tolerance, PC-DMIS ignores the values entered for these if they are minus entries.

4. Select the materials condition modifiers from the **Materials Condition** area.

You can set tolerances for multiple axes from the list. PC-DMIS stores your defined tolerances with the selected list item. Even if you select a different item from the **Axes** list and give it different tolerances, you can switch between the available axes to view and set their individual stored tolerances.

Axes

The **Axes** list in the **Position** dialog box (**Insert | Dimension | Position**) provides you with a list of available axes to which you can apply plus and / or minus tolerances. The list contains the following:

ALL = All the axes and options displayed in the drop down list

D1 = Diameter /Width of the first datum

D2 = Diameter /Width of the second datum

D3 = Diameter /Width of the third datum

DF = Diameter /Width of the feature

LF = Length of the feature if the feature is a slot

WF = Width of the feature if the feature is a slot

LD = Length of the datum if the datum is a slot

WD = Width of the datum if the datum is a slot

TP = Position tolerance and its associated deviation

FORM = The integrated form dimension for the feature.

- For a circle or cylinder feature, this is the Roundness (RN) dimension.
- For a plane feature this is the Flatness (FL) dimension.
- For a line feature this is the Straightness (ST) dimension.

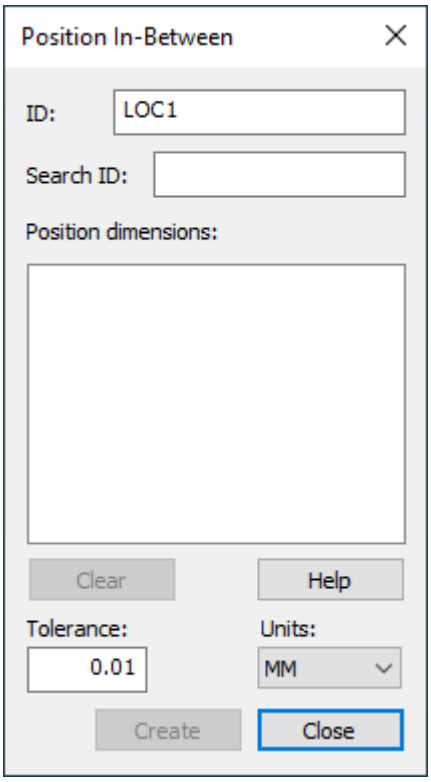
Plus

In the **Plus** box in the **Position** dialog box (**Insert | Dimension | Position**), you can enter the plus values for tolerances of the axis or axes selected from the **Axes** list.

Minus

In the **Minus** box in the **Position** dialog box (**Insert | Dimension | Position**), you can enter the minus values for tolerances of the axis or axes selected from the **Axes** list.

Dimensioning Position In-Between



Position In-Between Dialog Box

The **Insert | Dimension | Position In Between** menu option computes the in-between position deviation and deviation angle from two specified Position dimensions. This dimension is useful when you have a set of position dimensions on a pattern of holes and you want to report the position deviation and deviation angle of one hole relative to another where each individual hole of the pattern has been dimensioned with a separate position callout.

The ANSI or ISO standards do not include Position In-Between. This is a special implementation unique to PC-DMIS.

To create this dimension:

1. Access the **Position In Between** menu option to open the **Position In-Between** dialog box.

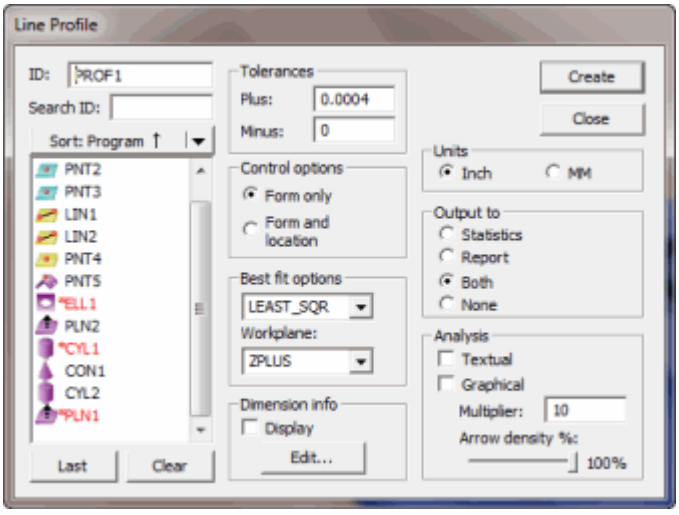
2. Select two dimensions from the **Position Dimensions** list. If you select a composite FCF Position or a multiple single-segment FCF Position (a Position with more than one segment), PC-DMIS only uses the Primary Dimension to compute the Position In-Between dimension.
3. Type a label ID for the dimension by typing in the **ID=** box.
4. Type a tolerance value for the dimension in the **Tolerance** box.
5. Select the measurement units to use from the **Units** list.
6. Click **Create**.

   The dimension appears in the Edit window with this information:

```
dimension_name = POSITION IN BETWEEN dim_1 AND dim_2,
UNITS=IN/MM
AX     NOMINAL        MEAS        +TOL        -TOL
     BONUS          DEV      DEVANG      OUTTOL
TP      0.0000      0.0000      0.0100
                            0.0000   -172.4842      0.0000
#-------
```

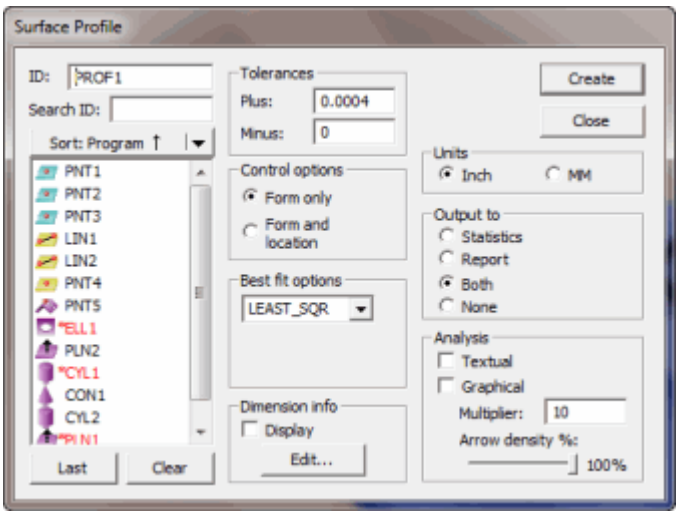
   DEV is the deviation of the In-Between dimension. DEVANG is the deviation angle of the In-Between dimension.

Dimensioning Profile - Line or Surface



Dimension Line Profile Dialog Box

The profile's **Line** menu option (**Insert | Dimension | Profile | Line**) computes the two-dimensional profile error of a curve.



Dimension Surface Profile Dialog Box

The profile's **Surface** menu option (**Insert | Dimension | Profile | Surface**) computes the three-dimensional profile error of a surface or curve feature.

You can use any type of feature for a profile scan.

For dimensioning of vision 2D Profile Auto-Features, the theoretical vectors are projected to the workplane. The deviations reported are 2D relative to this workplane rather than surface normal deviations.

By default, legacy profile form and location, reports the two-value method, as per ASME Y14.5.1 - 1994. PC-DMIS compares the MAX and MIN deviations to the +TOL and -TOL values to determine conformance.

An alternate single-value method is available from the PC-DMIS Settings Editor. From the **Option** section, you can change the `UseISOCalculations` setting to 1. This causes PC-DMIS to calculate a single MEAS value that is two times the maximum deviation. It then compares that single measured value to the +TOL. This method is in accordance with ASME Y14.5.1 - 2019.

To Dimension a Feature Using the Surface Profile Option:

1. Select **Insert | Dimension | Profile | Surface** from the submenu. The **Surface Profile** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box. If a datum feature is selected, it must be a plane.
3. Type the plus tolerance value in the **Plus** box.

4. Type the minus tolerance value in the **Minus** box.
5. Select either **Inch** or **MM** in the **Units** area.
6. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
7. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
8. Select either **Form Only** or **Form and Location** in the **Control Options** area of the dialog box.
9. If you selected **Form Only**, select the **Use Best Fit** check box. This causes the dimension to use the internal best fit alignment to allow the shape to rotate or translate until it finds the best fit for the feature.
10. Select the desired analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, type the multiplier value in the **Multiplier** box.
11. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
    - If you've selected the **Form and Location** option earlier, be sure to select **MaxMin** check box from the **Dimension Info Format** area of the **Edit Default Dimension Info** dialog box.
    - If you've selected the **Form Only** option earlier, be sure to select the **Measured** check box from the **Dimension Info Format** area of the **Edit Default Dimension Info** dialog box.
12. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = PROFILE,OF feat_1
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Displaying Surface Profile Contour Plots

PC-DMIS allows you to display a colored surface contour plot that rests on the surface of the CAD model in the Graphic Display window when you use Patch scans or UV scans and create a profile dimension from them.

> Contour Plots are only supported for Legacy Profile of a Surface. You cannot use them with the newer Geometric Tolerances.

To do this:

1. Import a solid CAD model.
2. Select the **Edit | Graphic Display Window | View Setup** menu option to open the **View Setup** dialog box.
3. From the **View Setup** dialog box, click the **Solid** check box for one of your views and then close the dialog box.
4. From the **Graphic Modes** toolbar (**View | Toolbars | Graphic Modes**), click the **Surface Mode** icon to place PC-DMIS in Surface Mode.
5. Create a Patch or UV scan. For details on these scan types, see "Performing a Patch Advanced Scan" and "Performing a UV Advanced Scan" topics in the "Scanning" chapter of the PC-DMIS CMM documentation.
6. Select the **Edit | Graphic Display Window | Analysis Options** menu option to open the **Analysis** dialog box.
7. From the **Analysis** dialog box, select the **Show Contour Plot** option and click **OK** until you return to the main PC-DMIS screen.
8. Select the **Insert | Dimension | Profile | Surface**) menu option to open the **Surface Profile** dialog box.
9. From the **Surface Profile** dialog box, select the Patch or UV scan from the **Feature List** box.
10. In the **Analysis** area of the **Surface Profile** dialog box, select the **Graphical** check box.
11. Make any other desired changes to the **Surface Profile** dialog box.
12. Click **Create** to generate the dimension.

You can see that PC-DMIS places a colored contour plot directly onto the model surface where the scan was.

To Dimension a 2D Feature Using the Line PROFILE Option:

1. Select **Insert | Dimension | Profile | Line** from the submenu. The **Line Profile** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box. You can select any 2D feature. If a datum feature is selected, it must be a plane.
3. Type the plus tolerance value in the **Plus** box.
4. Type the minus tolerance value in the **Minus** box.
5. Select either **Inch** or **MM** in the **Units** area.
6. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
7. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
8. Select either **Form Only** or **Form and Location** in the **Control Options** area of the dialog box.
9. If you selected the **Form Only** option, select the **Use Best Fit** check box. This causes the dimension to use the internal best fit alignment to allow the shape to rotate or translate until it finds the best fit for the feature.
10. If you selected the **Use Best Fit** check box, select the desired workplane to allow rotation and translation. Using the workplane restricts the alignment to be a two-dimensional alignment.
11. Select the desired analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, type the multiplier value in the **Multiplier** box.
12. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
    - If you've selected the **Form and Location** option earlier, be sure to select **MaxMin** check box from the **Dimension Info Format** area of the **Edit Dimension Info** dialog box.
    - If you've selected the **Form Only** option earlier, be sure to select the **Measured** check box from the **Dimension Info Format** area of the **Edit Dimension Info** dialog box.
13. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = PROFILE,OF feat_1
```

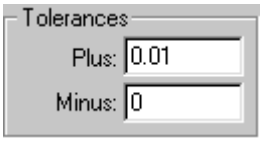
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Displaying Line Profile Contour Plots

PC-DMIS allows you to display a colored contour plot that rests on the surface of the CAD model in the Graphic Display window for linear or curve features.

For a Surface Profile, follow the instructions listed in "Displaying Surface Profile Contour Plots", and substitute the input feature with the appropriate line or curve feature (or scan).

Tolerances for Surface Profile Dimensions



This dimension type is considered either one sided, meaning a single positive value tolerance is applied, or two sided, meaning upper and lower tolerance values are applied.

- If the **Form Only** option in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) has been selected, only a plus tolerance can be applied.
- If the **Form and Location** has been selected, the profile dimension can be one sided or two sided.

Plus Tolerance

The **Plus** box in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) enables you to enter a tolerance value along the plus direction. This means that any profile that measures more than your nominal or theoretical profile can still be a valid measurement as long as it falls within the tolerance range specified.

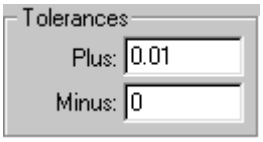
PC-DMIS also allows a negative-upper tolerance (or a minus tolerance in the positive range). This is done by entering in a minus sign before the value in the **Plus** box.

Minus Tolerance

The **Minus** box in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) enables you to enter a tolerance value along the minus direction. This means that any profile that measures less than your nominal or theoretical profile can still be a valid measurement as long as it falls within the tolerance range specified.

PC-DMIS allows positive-lower tolerances (or a positive tolerance in the negative range). This is done by typing a minus sign before the value in the **Minus** box.

Tolerances for Line Profile Dimensions



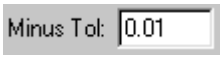
This dimension type is considered either one sided, meaning a single positive value tolerance is applied, or two sided, meaning upper and lower tolerance values are applied.

- If the **Form Only** option in the **Line Profile** dialog box (**Insert | Dimension | Profile | Line**) has been selected, only a plus tolerance can be applied.
- If the **Form and Location** has been selected, the profile dimension can be one sided or two sided.

Plus Tolerance

The **Plus Tol** box enables you to enter a tolerance value along the plus direction. This means that any measurement that measures more than your nominal or theoretical measurement can still be a valid measurement as long as it falls within the tolerance range specified.

Minus Tolerance



The **Minus Tol** box enables you to enter a tolerance value along the minus direction. This means that any measurement that measures less than your nominal or theoretical measurement can still be a valid measurement as long as it falls within the tolerance range specified.

Control Options for Surface Profile Dimensions

The options in this area in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) determine if a profile matches the same form, or form and location as its nominal.

Form Only

When dimensioning a profile, selecting the **Form Only** option in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**), you can tell PC-DMIS to only determine if a particular profile matches the same form as its nominal. This option ignores whether the location of the profile is outside of acceptable tolerances.

For a **Form Only** dimension, pay attention to the measured values in the generated dimension.

Form and Location

When dimensioning a profile, you can use the **Form and Location** option in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) to determine if a particular profile matches the same form and the same location as its nominal. This differs from the **Form Only** option because the location of the profile must also be within acceptable tolerances.

For a **Form and Location** dimension, pay attention to the Max/Min values in the generated dimension.

Control Options for Line Profile Dimensions

The options in this area in the **Line Profile** dialog box (**Insert | Dimension | Profile | Line**) determine if a profile matches the same form, or form and location as its nominal.

Form Only

When dimensioning a profile, you can use the **Form Only** option in the **Line Profile** dialog box (**Insert | Dimension | Profile | Line**) to tell PC-DMIS to only determine if a particular profile matches the same form as its nominal. This option ignores whether the location of the profile is outside of acceptable tolerances.

For a **Form Only** dimension, pay attention to the measured values in the generated dimension.

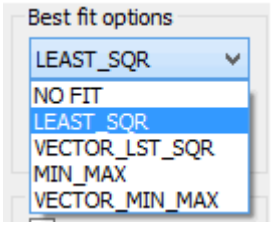
PC-DMIS 2009 and later support the Form Only Profile of a Line when the considered feature is a constructed curve based on a linear scan. The theoretical points are projected onto the theoretical curve to determine what theoretical points and vectors to use in the profile calculation.

Form and Location

When dimensioning a profile, using the **Form and Location** option in the **Line Profile** dialog box (**Insert | Dimension | Profile | Line**), you can determine if a particular profile matches the same form and the same location as its nominal. This differs from the **Form Only** option, because the location of the profile must also be within acceptable tolerances.

For a **Form and Location** dimension, pay attention to Max / Min values in the generated dimension.

Best Fit Options for Surface Profile Dimensions



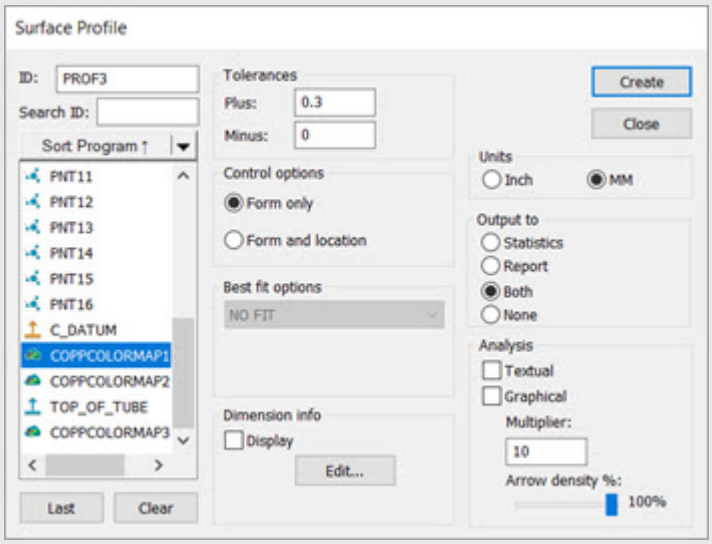
This area only appears in the dialog box if you select the **Form Only** option in the **Control options** area.

The **Best Fit options** area in the **Surface Profile** dialog box (**Insert | Dimension | Profile | Surface**) lets you create an internal Best Fit Alignment on the dimensioned feature.

NO FIT - This does not create an alignment.

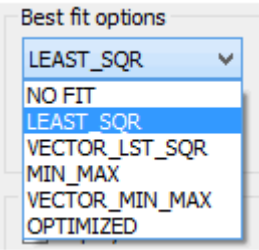
For descriptions of **LEAST_SQR**, **VECTOR_LST_SQR**, **MIN_MAX**, and **VECTOR_MIN_MAX**, see "The Best Fit Alignment Methods" in the "Creating and Using Alignments" chapter.

For legacy surface profiles, if you select a colormap from the Feature List Box, PC-DMIS automatically selects the **NO FIT** option for the surface profile and disables the **Best fit options** list.



Surface Profile dialog box with colormap feature selected

Best Fit Options for Line Profile Dimensions



This area only appears in the dialog box if you select the **Form Only** option in the **Control options** area.

The **Best Fit Options** area in the **Line Profile** dialog box (**Insert | Dimension | Profile | Line**) lets you create an internal Best Fit alignment on the dimensioned feature using one of these available fitting methods:

NO FIT - This does not create an alignment.

For descriptions of **LEAST_SQR**, **VECTOR_LST_SQR**, **MIN_MAX**, and **VECTOR_MIN_MAX**, see "The Best Fit Alignment Methods" in the "Creating and Using Alignments" chapter.

OPTIMIZED - This fit uses a genetic algorithm to determine if the data passes or fails the Line Profile dimension. You can also specify the lower tolerance value. (Typically, with the other fitting methods, you can't change the tolerance from 0.0.)

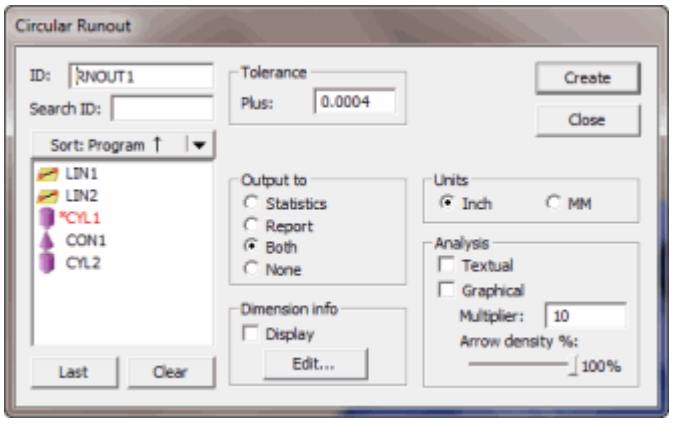
Workplane

When you use the line profile dimension, PC-DMIS creates an internal two-dimensional alignment that rotates and translates in all directions within the selected workplane to minimize the deviations of the individual hits. If you select a workplane that differs from the two-dimensional projection of the curve, an incorrect calculation results.

Dimensioning Runout - Circular or Total

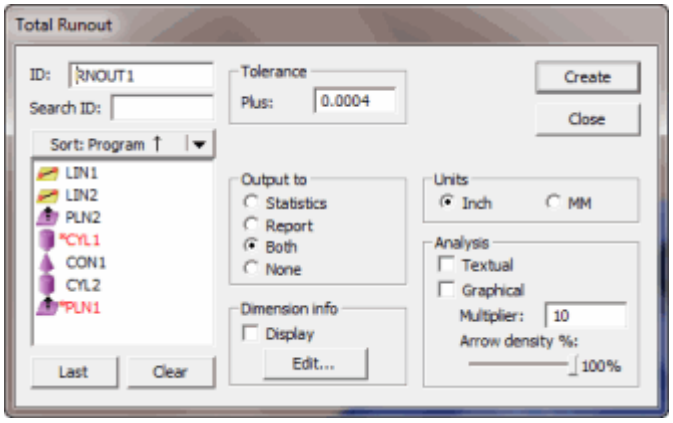
The **Insert | Dimension | Runout | Circular** and **Total** menu options determine the runout of the first feature with respect to the second feature (i.e., the second feature becomes the datum feature). If only one feature is selected, the origin and the workplane become the datum feature. In this case, PC-DMIS uses the position from the origin and the direction from the workplane for the datum. The text in the Edit window for the datum feature will read "THE ORIGIN".

- Circular Runout works for circles, cones, cylinders, and spheres.



Circular Runout Dialog Box

- Total Runout works for cylinders, cones, and planes.

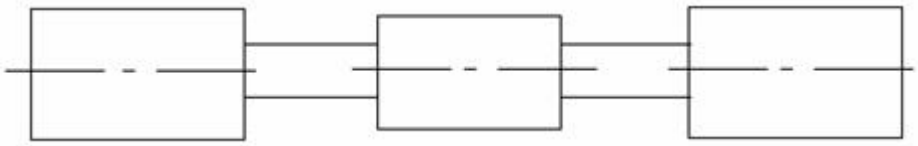


Total Runout Dialog Box

This dimension type is considered one sided, meaning a single positive value tolerance is applied.

Understanding Runout

Runout dimensioning is generally used in camshaft operations. A camshaft is designed to rotate around a center line. Since the separate cylinders comprising the camshaft are all on the same center line, you need to ensure that the centerline is not only parallel, but that the cylinders are also coaxial (or concentric).



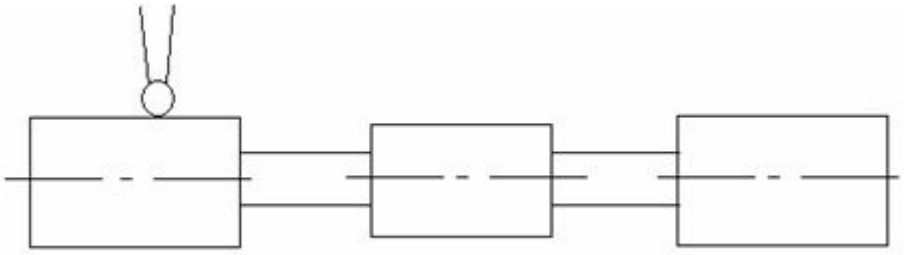
A sample Camshaft.

Additionally, you need to make sure the surface of the cylinders is round and straight (straight when compared to a separate datum axis, not just its own axis).

The main difference between runout and parallelism is that runout is checking points on a *surface* of a cylinder, and not just the *axis* of the cylinder. Runout also differs from a cylinder's straightness because it compares the surface of the cylinder to a datum axis, not just to itself. A runout dimension checks for these conditions.

How Runout is Measured

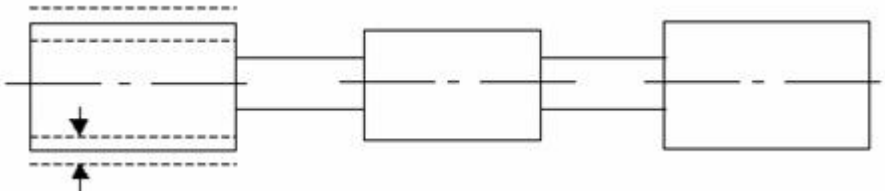
The typical, non-CMM way to measure runout is to put a small dial indicator on the surface of the cylinder, zero-out the indicator, and then spin the cylinder. This indicator measures any difference along that circle as the cylinder rotates.



Example of a runout being measured with a dial indicator.

A CMM essentially does the same thing. Instead of spinning the cylinder, however, the CMM's probe rotates around the cylinder and takes a finite number of hits.

The tolerance band for runout would look something like this:



Example showing a runout tolerance band (dotted lines).

The tolerance band has these properties:

- It has a width the size of the plus tolerance.
- It is parallel to the datum vector (the axis of some other line or cylinder).
- It is fixed at a certain radius from the cylinder’s axis.

Two Categories of Runout: Circular or Total

There are two categories for runout:

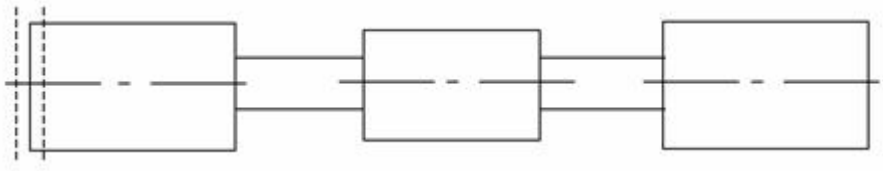
- Circular Runout measures just one circle—one row of hits—around the cylinder.
- Total Runout measures multiple circles up and down the entire surface of the cylinder with multiple rows of hits.

Obviously, you are somewhat limited using a CMM for total runout because you probably will not take 100 rows of hits around your cylinder, and something as little as 3 rows of hits clearly isn’t enough for determining total runout.

Nevertheless, PC-DMIS provides you with the ability to dimension both circular and total runout and lets you determine whether or not you've measured enough rows of the cylinder.

Using Runout to Check a Cylinder End

PC-DMIS also allows you to test for another case of runout specified in the standard: Checking the plane of one of the cylinder's ends. The tolerance for this type of runout is at a fixed orientation perpendicular to the datum axis.



Example showing runout on a cylinder end.

To Dimension a Feature Using the RUNOUT Option:

1. Select **Insert | Dimension | Runout | Circular** or **Insert | Dimension | Runout | Total** from the submenu. The **Total Runout** or **Circular Runout** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** area.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired analysis options by marking the **Textual** check box or the **Graphical** check box. If the **Graphical** check box was marked, enter the Multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

The dimension will appear in the Edit window with this information:

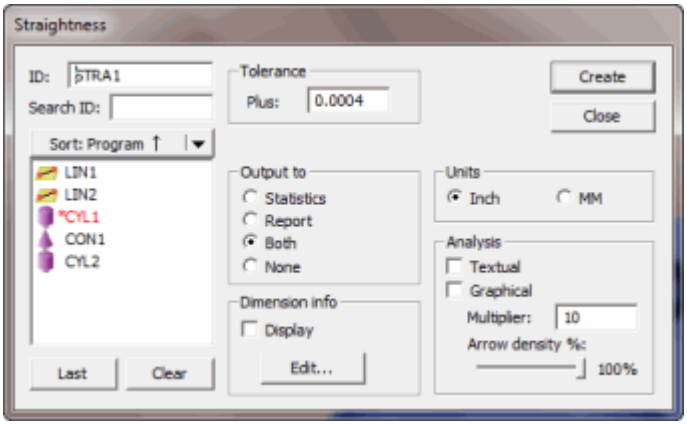
```
dimension_name = RUNOUT,OF feat_1,TO feat_2
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Runout Dimensions

The **Plus** box in the **Total Runout** or **Circular Runout** dialog box (**Insert | Dimension | Runout | Total** or **Insert | Dimension | Runout | Circular**) enables you to enter a tolerance value along the plus direction. This means that any runout that differs from your nominal or theoretical runout can still be a valid measurement as long as it falls within the tolerance range specified.

Dimensioning Straightness



Dimension Straightness Dialog Box

A minimum of three hits are necessary to determine the straightness of a line.

The **Insert | Dimension | Straightness** option calculates the straightness of a line. This dimension type is considered one sided, meaning a single positive value tolerance is applied.

To Dimension a Feature Using the STRAIGHTNESS Option:

1. Select **Insert | Dimension | Straightness** from the submenu. The **Straightness** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** section.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired Analysis options by marking the **Textual** check box or the **Graphical** check box. If the Graphical check box was marked, enter the Multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = STRAIGHTNESS,OF feat_1
```

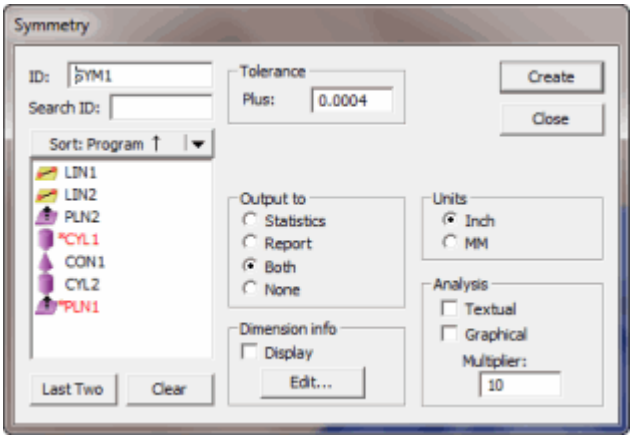
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Plus Tolerance for Straightness Dimensions

This **Plus** box in the **Straightness** dialog box (**Insert | Dimension | Straightness**) enables you to enter a tolerance value along the plus direction. This means that any straightness that differs from your nominal or theoretical straightness can still be a valid measurement as long as it falls within the tolerance range specified.

For information on how PC-DMIS reports tolerance zones for form dimensions, see "Reported Tolerance Zones for Form Dimensions" in the "Reporting Measurement Results" chapter.

Dimensioning Symmetry



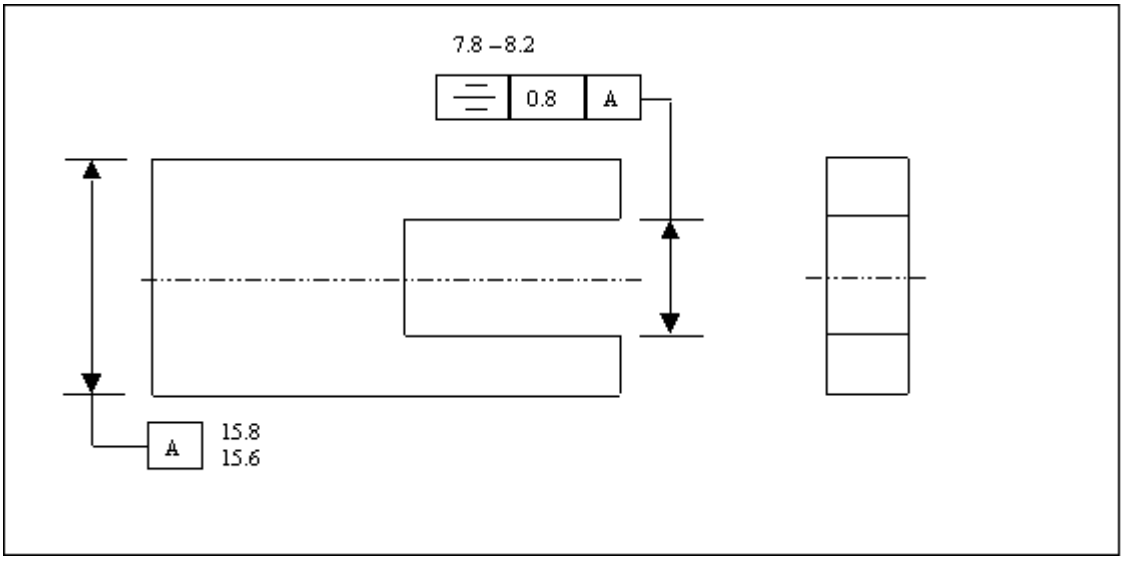
Symmetry Dialog Box

The **Insert | Dimension | Symmetry** menu option calculates the symmetry of a set of points with a datum feature, or two opposing lines with a datum feature.

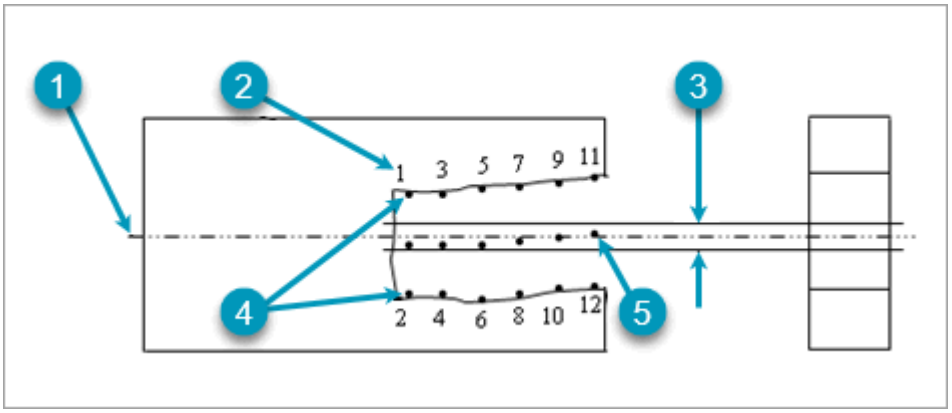
- If the first feature is a *set*, the second feature that is entered is the datum feature and must be either a plane or a line.

- If the first feature is a *line*, the second feature must also be a line and the third feature that is entered is the datum feature. In this case the third feature must be a plane or a line. This dimension type is considered one sided, meaning a single positive value tolerance is applied.
- If the third feature is a *plane*, the symmetry algorithm attempts to find a line on that plane that represents the datum. To find the correct line, it intersects (or crosses) the chosen plane with the workplane. For this reason, you should ensure that you are using the correct workplane.

According to the geometric dimensioning and tolerancing standard, the following graphic indicates how a symmetry dimension is displayed and interpreted.



Example of a symmetry dimension.



Example showing the interpretation of the symmetry dimension.

1 - The center plane of datum feature A.

2 - Order of point measurements.

3 - 0.8 wide tolerance zone.

4 - Opposed elements, with alternating points.

5 - Derived median points.

Explanation of Previous Graphic

Within the limits of size and RFS, all median points of opposed elements of the slot must lie between two parallel planes 0.8 apart, the two planes being equally disposed about datum plane A. The specified tolerance and the datum reference can only apply on an RFS basis.

Because of the way the symmetry is defined, PC-DMIS limits the features that can be used for the symmetry dimension. Since the points must be equally arranged about the datum in order to find midpoints between them, you must either select a feature set with alternating points or two opposing lines with the same number of points.

To Dimension a Feature Using the SYMMETRY Option:

1. Select **Insert | Dimension | Symmetry** from the submenu. The **Symmetry** dialog box appears.
2. Select the feature(s) to dimension from the **Feature List** box.
3. Type the plus tolerance value in the **Plus** box.
4. Select either **Inch** or **MM** in the **Units** area.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** options.
6. Select the optional **Display** check box if you want to view dimension information in the Graphic Display window.
7. Select the desired **Analysis** options by selecting one or both of the check boxes. If the **Graphical** check box was selected, enter the multiplier value in the **Multiplier** box.
8. If desired, select the **Display** check box in the **Dimension Info** area and click **Edit** to select the Dimension Info Format you would like displayed in the Graphic Display window.
9. Click the **Create** button.

The dimension will appear in the Edit window with this information:

```
dimension_name = SYMMETRY,FROM feat_1 TO feat_2
```

or

```
dimension_name = SYMMETRY,FROM feat_1 AND feat 2 TO feat 3
```

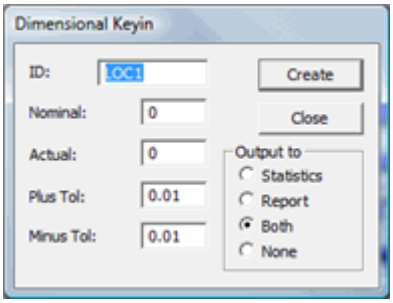
AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	0.0000	0.0100	0.0100	2.0000	0.0000	0.0000	0.0000	0.0000

After the dimension is created, the dialog box remains open. If desired, use your **Edit** button at this time to edit aspects of your dimension's information. See "Edit Default Dimension Info".

Plus Tolerance for Symmetry

This **Plus** box in the **Symmetry** dialog box (**Insert | Dimension | Symmetry**) enables you to enter a tolerance value along the plus direction. This means that any symmetry that differs from your nominal or theoretical symmetry can still be a valid measurement as long as it falls within the tolerance range specified.

Dimensioning through Keyboard Input



Dimensional Keyin Dialog Box

The **Insert | Dimension | Keyin** menu option allows you to capture and "key in" through the keyboard data not measured with the CMM (for instance, adding a dimension that is measured with calipers). This option allows you to print out all inspection results on the inspection report (not just features measured with the CMM). It is also useful when collecting data for statistical analysis.

To Add Dimensions Using the KEY IN Option:

1. Select **Insert | Dimension | Keyin** from the submenu. The **Dimensional Keyin** dialog box appears.
2. Type the ID number for the dimension in the **ID =** box.
3. Type the nominal value in the **Nominal** box.

4. Type the actual value in the **Actual** box.
5. Select where to output the dimension information. Select the **Statistics**, **Report**, **Both**, or **None** option.
6. Click the **Create** button.

The dimension then appears in the Edit window with this information:

```
dimension_name = KEYIN,feat_1
```

AX	NOM	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUT-TOL
M	5.0000	0.0100	0.0100	5.0000	0.0000	0.0000	0.0000	0.0000

Nominal

The **Nominal** box in the **Dimensional Keyin** dialog box (**Insert | Dimension | Keyin**) enables you to enter in the nominal (or theoretically perfect) measurement for a feature.

Actual

The **Actual** box in the **Dimensional Keyin** dialog box (**Insert | Dimension | Keyin**) enables you to enter in the actual measurement for a feature.

Tolerances for Keyboard Created Dimensions

You can type tolerances along the plus and minus directions of the newly generated dimension.

Plus Tolerance

The **Plus Tol** box enables you to enter a tolerance value along the plus direction. This means that any measurement that measures more than your nominal or theoretical

measurement can still be a valid measurement as long as it falls within the tolerance range specified.

Minus Tolerance

The **Minus Tol** box enables you to enter a tolerance value along the minus direction. This means that any measurement that measures less than your nominal or theoretical measurement can still be a valid measurement as long as it falls within the tolerance range specified.

Dimensioning Variables

There may be instances when working with expressions and variables that you want to create dimensions from variables that contain stored values. One popular way to do this is to first create a generic feature, populate the fields of the Generic Feature with the variable expressions you want to use, and then create a dimension of the Generic Feature.

For information on Generic Features, see the "Creating Generic Features" chapter. For information on variables, see the "Using Expressions and Variables" chapter.

Dimensioning a Variable Example

Suppose that you have these variables and you want to create a Location dimension using them:

```
ASSIGN/V_THEOX=10
ASSIGN/V_THEOY=5
ASSIGN/V_THEOZ=1
ASSIGN/V_MEASX=10.008
ASSIGN/V_MEASY=5.035
ASSIGN/V_MEASZ=0.997
```

For ease of use, this example gives the variables constant values. In real life application, your variables will most likely have dynamic values that change based on user input or from other external sources.

To dimension these variables:

1. Select the **Insert | Feature | Generic** menu option to open **Construct Generic Feature** dialog box.

2. Select the **Point** option.
3. Select the **Measured Values** option and type zero values into the X, Y, and Z boxes, then select **Nominal Values** and do the same thing.
4. Click **OK**. PC-DMIS inserts an empty Generic Feature (usually labeled F1 if it's your first Generic Feature) into the Edit window.
5. Place the Edit window in Command Mode to see the generic feature:

```
F1   =GENERIC/POINT,DEPENDENT,RECT,$
      NOM/XYZ,0,0,0,$
      MEAS/XYZ,0,0,0,$
      NOM/IJK,0,0,1,$
      MEAS/IJK,0,0,1
```

6. Now, navigate to the line `NOM/XYZ` , and in the first three zero fields, type **V_THEOX**, **V_THEOY**, and **V_THEOZ** respectively. These are your nominal variables.
7. Do the same thing on the `MEAS/XYZ` line, except type **V_MEASX**, **V_MEASY**, and **V_MEASZ** into the three zero fields of that line. Your command should now look like this:

```
F1    =GENERIC/POINT,DEPENDENT,RECT,$
       NOM/XYZ,V_THEOX,V_THEOY,V_THEOZ,$
       MEAS/XYZ,V_MEASX,V_MEASY,V_MEASZ,$
       NOM/IJK,0,0,1,$
       MEAS/IJK,0,0,1
```

8. Press F3 to mark the Generic Feature.
9. Create a Location dimension using this feature. Give it a plus and minus tolerance of .02.
10. Execute the measurement routine.

You should get a dimension that looks something like this in your report:

MM | DIM LOC1= LOCATION OF POINT F1

AX	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	DEV	OUTTOL
X	10.000	0.020	0.010	10.008	10.008	10.008	0.008	0.000
Y	5.000	0.020	0.020	5.035	5.035	5.035	0.035	0.015
Z	1.000	0.020	0.010	0.997	0.997	0.997	-0.003	0.000

Using Geometric Tolerances

Introduction to Geometric Tolerances and Feature Control Frames

In this Topic

Introduction to Geometric Tolerances and Feature Control Frames
- Introduction
- Conceptual Process for Evaluating Geometric Tolerances
- Specification Versus Verification
  - Recommended Steps for a Careful Study
- Considered and Toleranced Features
- Phases of Evaluation
- Comparisons to Past Practice

Introduction

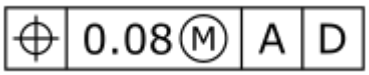
A manufactured part has functional requirements. These requirements often involve the shapes, sizes, orientations, and locations of the features—they are geometric requirements. The features must meet the geometric requirements either alone or with other features.

Geometric tolerances communicate the geometric requirements precisely. A drawing or CAD model specifies geometric tolerances in these ways:

- Feature Control Frames (ASME Y14.5)
- Tolerance Indicators (ISO 1101)

Terminology Note: ASME feature control frames and ISO tolerance indicators appear and function similarly to each other. Because of this, we use the term "Feature Control Frame" to refer to either one. We also use the term "Geometric Dimensioning and Tolerancing Standard" (or "GD&T Standard" for short). This term refers to both the standards involved (ASME Y14.5 or ISO 1101) even though the ISO term is “geometrical product specifications.”

A Feature Control Frame (FCF) uses numbers and symbols in rectangular boxes, like this:



While PC-DMIS can verify geometric tolerances according to any standard, it can most easily do this when tolerances are written according to these specific standards:

- ASME Y14.5 1994 / 2009 / 2018
- ASME Y14.5.1 1994 / 2019
- ISO 1101 : 2012/2017
- ISO 5459 : 2011
- ISO 5458 : 1998
- ISO 14405-1 : 2010
- ISO 17450-3 : 2016
- ISO 2692 : 2014
- ISO 1660 : 2017

Terminology Note: For brevity, we refer to the ASME group of standards as "ASME Y14.5" and the ISO group of standards as "ISO 1101".

Conceptual Process for Evaluating Geometric Tolerances

A geometric tolerance always includes these items:

- One or more toleranced features
- A tolerance zone for each toleranced feature
- Zero or more datum features. These constrain how toleranced features may be optimized into the tolerance zone or zones.

When you evaluate a geometric tolerance, you need to know all the basic dimensions (ASME Y14.5) or the theoretically exact dimensions (ISO 1101). These are the nominal relationships between all the features in question. This means all the features must have correct nominals (THEOs). If these are incorrect, PC-DMIS can evaluate the geometric tolerances incorrectly.

The easiest way to ensure the nominals are correct is to construct the measurement routine from the CAD model.

Specification Versus Verification

The ASME and ISO standards families for GD&T are *specification* standards. Geometric tolerances are one type of specification. The standards define the specifications—what the

tolerances mean—but they do not tell you how to verify that your part meets its specification.

The specification world deals with *perfect* information. The specification is defined in terms of the entire actual surface. There are infinite points with zero measurement uncertainty.

The verification world deals with *imperfect* information. The verification is defined in terms of the measured points. These are finite points with measurement uncertainty. When you choose math options, your goal is to make the verification math yield a result as close as possible to the specification. In other words, the specification is what you're trying to measure (the "measurand"), while the measurement is your best approximation to the specification. Sometimes the best verification math is very different than the specification math.

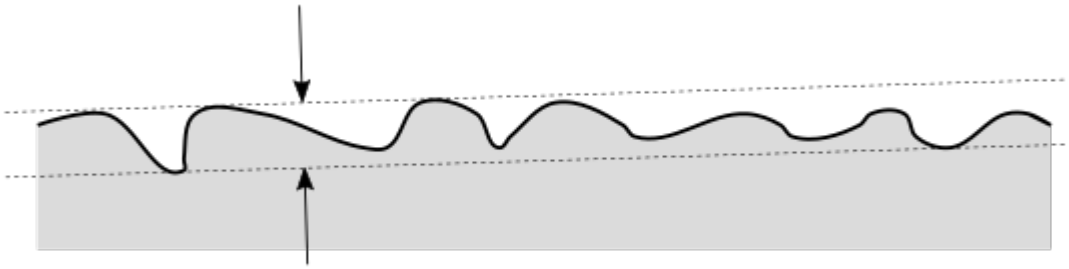
Because of this difference between specification and verification, datums and tolerance values usually come in pairs. For example, there are both actual datums and measured datums, as well as actual tolerances and measured tolerances:

- Actual datums and actual tolerance values are defined by the specification using perfect information about the actual surface.
- Measured datums and measured tolerance values are approximations to actual datums and tolerance values based on measured data.

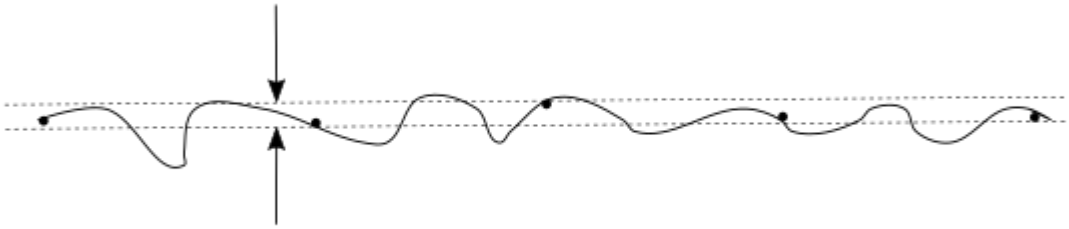
Unfortunately, there are no verification standards for GD&T, so it is very difficult to meaningfully compare different software packages that are intended to evaluate the same specification. Different software packages use different algorithms to approximate the actual datums and tolerance values, so they get different measured datums and different measured values.

For example, the actual straightness of a line is based on perfect information about the surface. The measured straightness is based on the measured points. The measured straightness can be smaller than the actual straightness if the actual high and low points are not measured. Alternatively, the measured straightness can be larger than the actual straightness if the measured points' uncertainty is large.

Below is an illustration of a surface's actual straightness. The entire actual surface cross section must lie between two parallel lines, where the distance between the lines is minimal. The distance between the lines is the actual value.



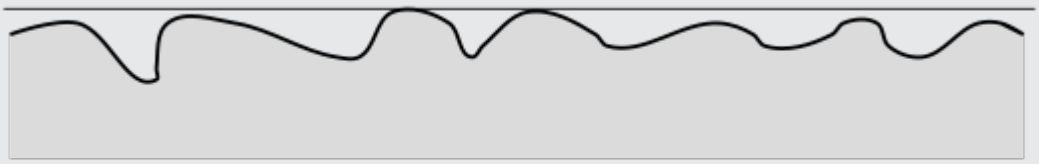
Below is an illustration of a surface's measured straightness. The measured points on the surface cross section must lie between two parallel lines. The distance between the lines is the measured value. The thin solid line represents the actual surface (perfect information), while the small dots represent the measured surface points (imperfect information). In this case, too few points were measured, so the measured value is smaller than the actual value.



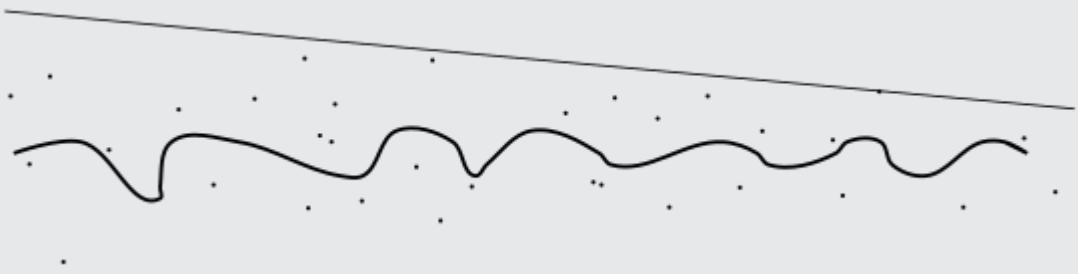
In the specification world, ISO 5459 : 2011 tells us that a primary datum plane is defined as a constrained min-max plane. This plane is external to the material. It contacts at least one high point, and it minimizes the deviations to the low points (after filtering the surface).

In the verification world, if we've measured our points densely (lots of points) and we have much lower measurement uncertainty than the form error, then the best algorithm in this verification situation is a constrained min-max algorithm. That algorithm ensures our measured datum plane matches the specified datum plane as closely as possible. On the other hand, in the verification world, if our measured points have larger measurement uncertainty than the form error (this is common), then we should probably use a plain (unconstrained) least-squares algorithm. This is because almost all of the measured form error is really measurement error, and so contacting the high points is going to put the datum plane way off the actual surface. In this case, plain least squares is the better choice.

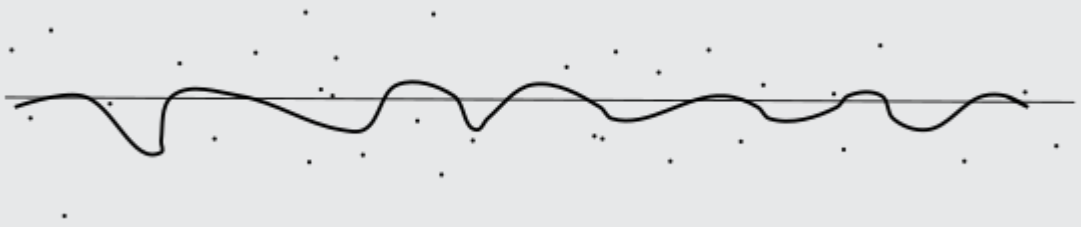
Below is an example of a primary datum plane, containing form error. The actual datum specified by ISO 5459 : 2011 is shown by the thin straight line.



Below is an example of that primary datum plane, which has been measured by a sensor where each measured point has large measurement uncertainty. The solid wavy line represents the actual surface. If you use the specification math (constrained min-max after void filtration), the measured datum is very far from the actual datum, as shown by the thin straight line.



Below is an example of the same primary datum, with the same measured points, but using plain (unconstrained) least-squares math. This measured datum is a much better approximation to the actual datum.



Frequently, you may need to use different algorithms during verification than the specification uses. Because of this, geometric tolerance commands give you *math options* that allow you to control what algorithms are used for the verification. You may find it difficult to choose the best math options. The only way to really be certain you've chosen the best math options is to do a careful study.

Recommended Steps for a Careful Study

1. Take several actual parts that represent the variety of errors the manufacturing process can produce.
2. Densely measure all the parts with lots of cross sections and use equipment that offers much less measurement uncertainty than the form error.
3. Choose math types that closely approximate the specification.
4. Measure the same parts in the way you expect to actually measure your parts in production. Use the same sensors and measurement strategies you plan to use.
5. Choose a wide variety of math types and compare how well those math types approximate your dense-and-high-accuracy measurements. That lets you pick the math type combination that most closely approximates the specification.

Typically, the best math option depends on the ratio between your measurement uncertainty and your form error. If the measurement uncertainty is much larger than the form error, then you can't measure the real form error with your sensor anyway, and it's best to choose something simple like plain least squares for the datum and feature math types. On the other hand, if the measurement uncertainty is much less than the form error, then it's best to choose math types that closely approximate the specification.

The discussion of how to pick math types is completely separate from the question of whether your sensor is able to verify your specifications. That complex topic will not be discussed in this documentation. However, it's critical that verification engineers choose sensors and measurement strategies that are accurate enough, with the chosen math types, to verify the specification.

Considered and Toleranced Features

There is a difference between considered features and toleranced features.

A considered feature is a measured surface that represents the controlled surface. You measure the considered features in your measurement routine. A considered feature is, in the language of ISO 1101, a measurement of a real integral feature. When you use the geometric tolerance command, you choose the considered features for each tolerance.

The toleranced feature is what goes inside the tolerance zone. The tolerance feature is sometimes the considered feature surface. Sometimes it is something derived from the considered feature surface. For example, it might be the axis of the unrelated actual mating envelope (ASME Y14.5) or the extracted median line (ISO 1101). For more information, see "Deriving the Toleranced Feature".

Phases of Evaluation

The actual evaluation of a geometric tolerance then consists of several phases:

1. Measurement of the considered features' surfaces and datum features' surfaces
2. Computation of the datums in their hierarchical order of precedence
3. Production of the toleranced feature from the considered feature as necessary
4. Evaluation of each toleranced feature within its tolerance zone. This is subject to the datums' constraints.
5. Reporting the evaluated result

In most cases, you are responsible for phase 1 of this evaluation process. The PC-DMIS geometric tolerance command handles the other phases in a way that complies with ASME Y14.5 or ISO 1101.

Because you are responsible for phase 1 of this evaluation process, you must measure the surfaces densely enough, and with enough cross sections, so that the measured datums and measured values can closely approximate the actual datums and actual values. This means you must thoroughly understand the specifications, the strengths and weaknesses of your measurement equipment, and the types of errors your manufacturing process can produce.

Comparisons to Past Practice

PC-DMIS 2020 R2 introduced the geometric tolerance command. Before this, PC-DMIS had some FCF capabilities that supported older standards and that were more limited.

Terminology Note: This documentation refers to the old capabilities as "XactMeasure". This is because in prior versions of PC-DMIS, the feature control frame dialog box had the text “XactMeasure” in its title bar. The current geometric tolerance command has the text “Geometric Tolerance”.

Several comparisons to past practice are in this documentation. They compare the XactMeasure capabilities and behavior to the geometric tolerance command’s capabilities and behavior.

Structuring your Measurement Routine for Geometric Tolerances

In this Topic

Structuring your Measurement Routine for Geometric Tolerances
- Introduction
  - Comparison to Past Practice - Referencing the GD&T Standard
  - Comparison to Past Practice - Constructed Features
  - Comparison to Past Practice - Measured Data in Cross Sections
  - Comparison to Past Practice - Considered Features and Measured Data
  - Comparison to Past Practice - ISO Math Types

Introduction

Starting with PC-DMIS 2023.2, when you create a new measurement routine, the **New Measurement Routine** dialog box requires that you select the appropriate GD&T standard (see "Creating New Measurement Routines" in the PC-DMIS Core documentation). PC-DMIS applies the GD&T standard that you select (ASME Y14.5 1994, ASME Y14.5 2009, ASME Y14.5 2018 or ISO 1101 2012/2017) for every Geometric Tolerance and Size command that you create in your new measurement routine. For more information, see the "Comparison to Past Practices - Referencing the GD&T Standard" section below.

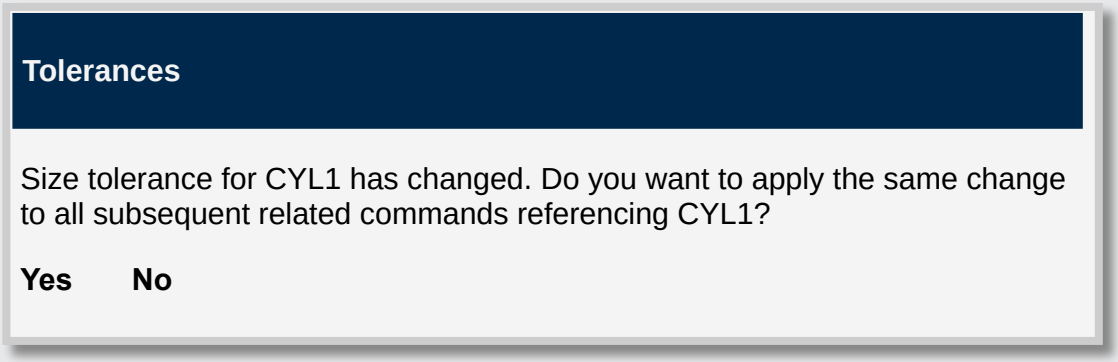
In most cases, for your measurement routine, we recommend a simple structure like this:

1. Construct an initial alignment to find the part in 3D space. For information, see the "Creating and Using Alignments" chapter in the PC-DMIS Core documentation.
2. Measure all the considered feature surfaces and datum feature surfaces.
3. Define the datums using datum definition commands.
4. Define the specified size tolerances and geometric tolerances using geometric tolerance commands.
5. Define any simultaneous tolerances using simultaneous tolerance commands.

You need to create the size tolerances and geometric tolerances on datums before any geometric tolerances that reference those datums. This is because geometric tolerances that reference a datum frequently need to know about any size tolerances and geometric tolerances on that datum. If you later edit any size tolerance on a datum, please ensure that all subsequent geometric tolerances that reference that datum have the correct size tolerance information for the datum. For more information, see "How PC-DMIS Solves and Uses Datums" in the PC-DMIS Core documentation.

If you edit a feature's upper or lower tolerances from the Edit window or from the **Geometric Tolerance** dialog box (**Feature Control Frame** or **Nominals** tab), and the same feature is used as a datum or as a considered feature, PC-DMIS displays a message asking if you want to apply the same changes to all subsequent commands referencing that feature.

For example:



If you click **Yes**, PC-DMIS updates the size tolerances for any Geometric Tolerance commands below the cursor position that references the same feature, either as a considered feature or datum.

If you click **No**, PC-DMIS updates the edited size tolerance only. PC-DMIS does not update any of the respective size tolerances for any related Geometric Tolerance commands below the cursor position that uses the same edited feature either as a considered feature or datum.

We do not recommend using copy/paste or paste-with-pattern to replicate the geometric tolerance command. There are some cases where it works, and there are other cases where paste-with-pattern is fundamentally incapable of working correctly. We also do not recommend putting the geometric tolerance command inside a loop, for similar reasons: some cases work, and some cases cannot ever work. It is okay to put the entire measurement routine inside of one loop.

Comparison to Past Practice - Referencing the GD&T Standard

In the past, XactMeasure allowed you to create feature control frame commands that reference different GD&T standards within the same measurement routine. You could also toggle the referenced GD&T standard from within the command. Earlier versions of the

Geometric Tolerance command also supported this behavior for migration purposes. However, it does not make sense to mix GD&T standards within the same measurement routine. This is because parts are designed according to only one standard.

For this reason, starting with PC-DMIS 2023.2, you can no longer reference both ASME and ISO standards within the same measurement routine.

Comparison to Past Practice - Constructed Features

In the past, because of XactMeasure limitations, you often had to use constructed feature commands. These were mid-planes, intersection lines, and so on. You used them as either considered features or datum features.

With Geometric Tolerances, however, most constructed feature commands get in the way. Constructed features can impede a Geometric Tolerance from understanding the measured surface. However, in these few cases it makes sense to use a constructed feature command:

- Constructed width commands are necessary to represent a width (ASME Y14.5) or opposed parallel plane (ISO 1101) features. This is because there is not yet an auto-width command in PC-DMIS. Constructed width commands keep all the surface data, and so they do not impede the geometric tolerance command.
- Constructed set commands are sometimes necessary. When all the inputs represent measured surfaces, constructed sets keep all the surface data, and so they do not impede the geometric tolerance command.
- In rare cases, it makes sense to measure a feature with separate point commands like vector points. You can then construct a Best Fit Recompensated (BFRE) feature from the vector points. The constructed feature retains the surface data, and so this does not impede the geometric tolerance command.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

- In rare cases, a considered feature or datum feature might need to be a derived geometry. This means it has no surface. An example of this is the minimum circumscribed circle that contains three pins in ASME Y14.5 2018 Figure 7-42 (b). In such cases, the only way to convey the specified intent to the geometric tolerance command is to construct a feature that lacks surface data. If you do this, you are responsible to comply with applicable standards.

The geometric tolerance command uses constructed features without surface information in these ways:

- As the toleranced feature
- As the pre-resolved datum feature (situation feature in ISO 5459 language)

In such cases, you are taking over phases 2 and 3 of the conceptual evaluation process. It is your responsibility to construct the feature according to appropriate standards. For information about the conceptual evaluation process, see "Introduction to Geometric Tolerances and Feature Control Frames" in the PC-DMIS Core documentation. For information about which feature types have and do not have surface data, see "Feature Types With and Without Surface Data" in the PC-DMIS Core documentation.

Comparison to Past Practice - Measured Data in Cross Sections

Certain types of geometric tolerances require evaluation of measured data in cross sections. For example, consider the specification of axis straightness on a cylinder. You need to measure the cylinder in several cross sections for these reasons:

- To compute the center of each cross section
- To evaluate the axis straightness of the circles' centers

The XactMeasure command required you to measure several circle commands. These may also have been child features of a scanning strategy. You then had to construct a 3D Best Fit (BF) line through the circle centers. Finally, you created an XactMeasure axis straightness tolerance on the BF line.

With the geometric tolerance command, you no longer need to go through so many steps. Now, you measure the cylinder (using a measurement strategy or not). You then evaluate axis straightness on the cylinder. The geometric tolerance command automatically divides the data into cross-sections (based on the distribution of the points). It then computes each circle's center and evaluates straightness on the centers.

Starting with PC-DMIS 2025.1, the geometric tolerance command vastly improved its automatic sectioning to support a wider range of different distribution of points within a feature and still calculate cross sections. This enhancement was mainly intended to support features measured with a laser sensor but is applied to all sensor types to allow more flexibility. Laser data is rarely organized into clean cross sections, and our previous cross section capabilities limited the types of tolerances that could be

evaluated. For example, you were previously unable to evaluate circularity, circular runout, or straightness (surface or axis) of a laser cylinder because there were typically no clean cross sections. In most cases, the enhanced cross sectioning functionality is now able to automatically extract linear or circular cross sections from any point pattern when features are measured with adequate point density. However, this may result in unwanted behavior in some cases. For example, a minimum of three hits spanning a minimum 90 degrees of arc are required for each circular cross section. This means that these extreme cases may fail:

- Features with very sparse hit data.
- Features with spiral hit patterns consisting of a single or very few revolutions and a steep pitch (Adaptive Spiral scan cylinders, auto cylinders, or auto cones with a pitch applied, constructed BF/BFRE cylinders or cones, manually measured cones or cylinders).
- Features with partial coverage.

In general, if you want to ensure that the geometric tolerance command sections the data exactly as you desire, you should measure the data in cross sections (as was required for previous PC-DMIS versions). For manual devices (such as portable arms), it is recommended to utilize Auto-Triggers to provide control to ensure the data is collected in a cross section. This also helps to maintain consistency when there are multiple users of the device.

Comparison to Past Practice - Considered Features and Measured Data

With the geometric tolerance command, every considered feature has a measured value. In some cases, this is different than XactMeasure's behavior.

For example, take a surface profile tolerance of three circles. In XactMeasure it yielded only one measured value. But with a geometric tolerance command, it now yields three measured values.

This means if you only want a single measured value, you need to construct a set of the input features and make the set the considered feature.

We chose this new behavior because it gives more flexibility in how to report measured values. Single measured values are still available with a constructed set. But separate measured values are also available where they weren’t before.

Comparison to Past Practice - ISO Math Types

In versions of PC-DMIS earlier than version 2025.1, when you selected ISO 1101 as the GDT standard, any of these three geometric tolerance command math types were available:

- **FEATURE_MATH** - This math type determined which math (**DEFAULT** or **LSQ**) that PC-DMIS would use for considered features. PC-DMIS would apply it to

both the size calculation and the way in which the considered feature was calculated for comparison to the associated geometric tolerance specification.

- **DATUM_MATH** - This math type determines which math (**DEFAULT**, **LSQ**, or **CL2**) that PC-DMIS uses for datum features. PC-DMIS applies it to both the datum size calculation and the datum fitting.

- **TOLERANCE_ZONE_MATH** - This math type determined which math (**DEFAULT** or **LSQ**) that PC-DMIS would use for form or profile calculations.

If you changed the feature math or, if your tolerance referenced the considered feature at MMC/LMC, it also affected the way size was reported (for details, see the "Evaluating Size with the Geometric Tolerance Command" topic in the PC-DMIS Core documentation). This made it easier to manually verify bonus calculations because all relevant size information was presented in the report.

Due to the rule of independency (see ISO 8015), ISO allows all characteristics of a geometric tolerance to be calculated separately. Starting with PC-DMIS 2025.1, the geometric tolerance command adopts these changes:

- **SIZE_MATH** - This is a new math type which PC-DMIS can use to determine how it calculates the feature's size. You can select **DEFAULT**, **LSQ**, or add an ISO modifier. For details on how to add an ISO modifier, see the "ISO Size Modifiers" section of the "Evaluating Size with the Geometric Tolerance Command" topic in the PC-DMIS Core documentation.

- **FEATURE_MATH** - This math type only determines the way in which the considered feature is calculated for comparison to the geometric tolerance specification. You can select **DEFAULT**, **LSQ**, or add an ISO Associated Toleranced Feature modifier to the tolerance section of the Feature Control Frame.

- **DATUM_MATH** - This math type determines which math (**DEFAULT**, **LSQ**, or **CL2**) that PC-DMIS uses for datum features. PC-DMIS applies it to both the datum size calculation and the datum fitting.

- **TOLERANCE_ZONE_MATH** - This math type separates form and profile tolerances as described below:

    - Form tolerances - You can select between **DEFAULT**, **LSQ**, or you can add an ISO Reference Feature Association modifier to the tolerance section of the Feature Control Frame.

    - Profile tolerances (without a datum reference) - You can select between **DEFAULT** and **LSQ**.

Depending on the combination of math types or modifiers that you select, be aware that it may no longer be possible in every situation to manually verify bonus calculations. PC-DMIS always uses the applicable mating envelope (E) to determine ISO bonus values, but that size may not be present in the report if you select an alternative size modifier.

For more information, please refer to ISO 1101:2017, section 8.2.2.2.2 (Associated Tolerance Feature specification element), and 8.2.3.1 (Reference Feature Associated specification element). Also see the "Deriving the Toleranced Feature" and "Evaluating Size with the Geometric Tolerance Command" sections of the PC-DMIS Core documentation.

Defining and Using Datums

Most geometric tolerances reference one or more datums. A datum reference consists of a datum identifier that refers to one or more datum features. A datum identifier is usually a single datum letter such as A or D, but it can be a sequence of up to five letters. In PC-DMIS, a datum identifier can refer to any of these:

- A single feature, such as a plane, line, point, cylinder, circle, cone, sphere, or width—this is often called a *single datum.*
- A pattern of like features of size that all have the same nominal size and size tolerance; this case is restricted to cylinders, circles, spheres, and widths—this is often called a *datum pattern.*

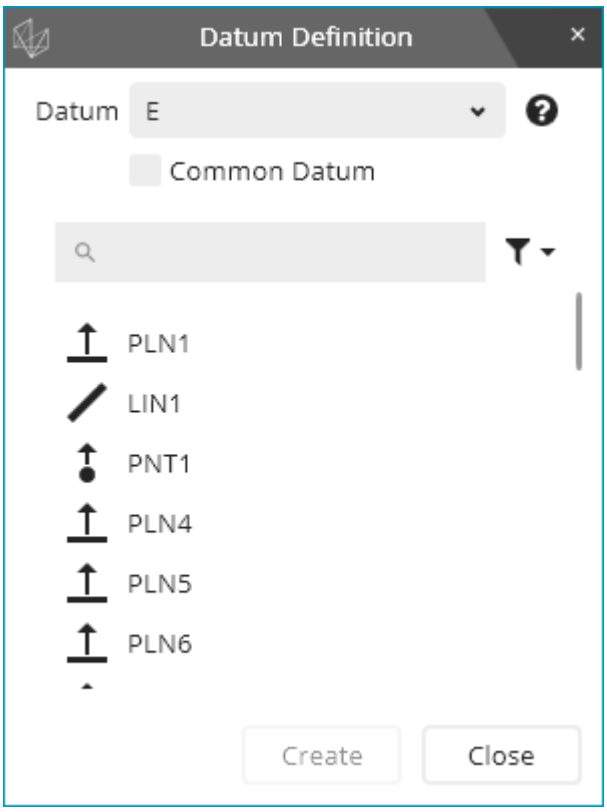
PC-DMIS only allows you to use a single 1D width as a datum. You cannot use patterns of 1D widths as a datum.

A geometric tolerance may reference two or more datum identifiers at once by joining them with a hyphen, such as C-D; this is often called a common datum. PC-DMIS supports many combinations of features as common datums. For more information see “Common Datums”.

Datum Definition Dialog Box Usage and Command Syntax

Before you can use a datum reference, you must define it in your measurement routine. You can do this with the Datum Definition command (`DATDEF`). To define a datum reference and create this command, from the menu, select **Insert | Dimension | Datum Definition**.

The **Datum Definition** dialog box appears:



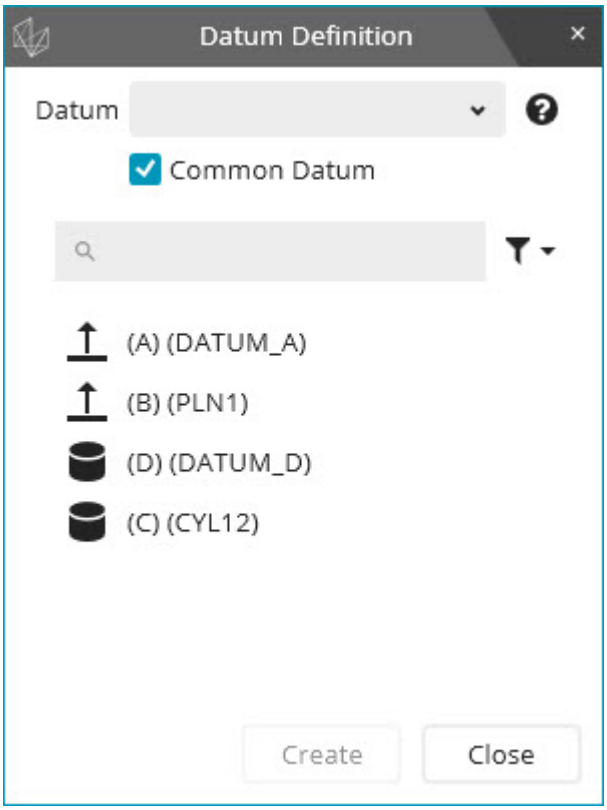
1. In the **Datum** box, type or select the datum name.
2. From the list of datum features, select one or more datum features.
3. If you select one feature, the datum reference represents a single datum. After you select that feature, PC-DMIS filters the feature list so that it only shows other features with the same characteristics. For example, if you select a circle with a diameter of 25 mm, it filters the list by other circles of that same diameter. This makes it easier for you to select additional, similar features so that your datum reference represents a datum pattern.
4. Click **Create**.
5. As needed, continue to use the dialog box to create additional datums and common datums.

The Datum Definition command's Edit window syntax in Command Mode might look like this:

```
DATDEF/A,FEATURES=PLN2,,
```

Datum - This box defines the name of the datum reference. It is typically a single letter such as E or a sequence of a few letters such as BG.

Common Datum - This check box lets you define a common datum. If you mark this check box, the feature list doesn't show features anymore; instead, it shows datums you have already defined, and the features associated with the datums.



Datum Definitions Dialog Box - Common Datum Check Box

For example, to define the common datum A-B, first, define datum A, then, define datum B then select the **Common Datum** check box. From the list of available datums, select datum A and then datum B to define the common datum A-B.

If you use a common datum, the Datum Definition command's Edit window syntax in Command Mode might look like this:

```
DATDEF/A-B,FEATURES=DATUM_A,PLN1,
```

Single Datums

A *single datum* refers to a datum identifier like A or AC that refers to only one datum feature. A single datum may refer to one of the following types of features:

- Plane
- Line
- Point
- Cylinder

- Circle
- Sphere
- Cone
- Width

Most datums in the geometric tolerance command are like the datum features on an inspection gage:

- They have a fixed distance and orientation with respect to each other.
- They engage the part in a fixed order of precedence.

Comparison to past practice:

Under XactMeasure, PC-DMIS treated datum features like alignment features in most cases. They defined the level, rotation, and origin.

The geometric tolerance command is more accurate, because it simulates how the datum reference frame contacts the mating part.

Feature Types that Represent a Planar Surface

A datum feature can represent a planar surface with a plane feature, a line feature measured on a surface, or a point feature measured on a surface.

While you can measure a surface line or surface point feature on a non-planar feature, PC-DMIS always treats them as coming from planar surfaces when referenced as datums in the geometric tolerance command. For more details, see "How PC-DMIS Solves Datums".

Feature Types that Represent a Cylindrical Surface

A datum feature can represent a cylindrical surface with a cylinder feature or a circle feature. While you can measure a circle on a non-cylindrical feature, PC-DMIS always treats circles as coming from cylindrical surfaces when referenced as datums in the geometric tolerance command. For more details, see "How PC-DMIS Solves Datums".

Common Datums

A *common datum* refers to a datum identifier like A-B or A-BC-D. It has one or more hyphens that separate defined datum identifiers. PC-DMIS supports many combinations of common datums described in this section of the documentation.

Common Datums: Best Practices

When you reference features as common datums, we recommend you use 3D features for these reasons:

- 3D features represent feature surface(s) and capture more data to properly evaluate your datums.
- 3D features properly control the applicable Degrees of Freedom (DOF) which you can then use as primary, secondary, and tertiary datums.
- You can only reference 2D and 1D width features as secondary or tertiary datums.

PC-DMIS does not support a pattern of 2D features as part of a common datum. PC-DMIS only allows datum patterns made up of cylinders, spheres, or 3D widths as part of a common datum.

Common Datums: General Guidelines

In this Topic

Common Datums: General Guidelines
Example 1: Plane A and Parallel Plane B as Common Datum A-B
Example 2: External Cylinder B with a Pattern of External Cylinders C as Common Datum B-C
Example 3: External Width A with an External Width B as Primary Common Datum A-B
Example 4: Internal Cylinder C with an Internal Cylinder D as Secondary Common Datum C-D
Example 5: Pattern of Internal Cylinders B and Pattern of External Cylinders C as Common Datum B-C
Example 6: Plane A and External Cylinder B as Common Datum A-B
Example 7: External Cylinder A and Coaxial Internal Cone B as Common Datum A-B
Example 8: External Cylinder A and Coaxial External Cylinder B as Common Datum A-B

For common datums, PC-DMIS allows you to mix the same feature types. You can combine individual features with a pattern or multiple patterns. PC-DMIS ***only*** allows you to mix different feature types in these cases:

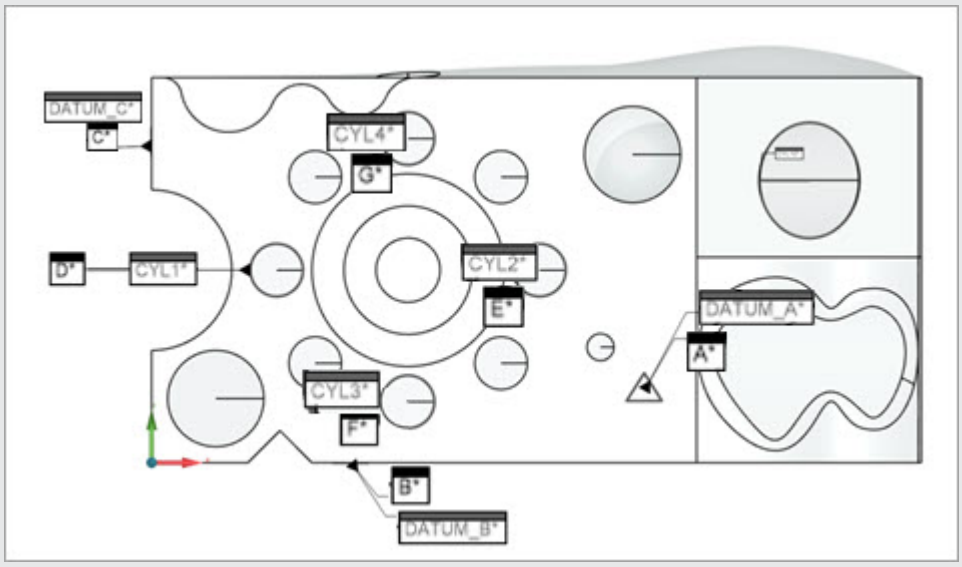
# of Input Features	Common Datum Feature #1	Common Datum Feature #2	Common Datum Features #3-5 (When Selected)	Comments
Minimum of two features	Plane	Plane	Plane	Only multiple parallel planes. See Example 1.
2		Cylinder	-	Only a single Plane & single Cylinder that is perpendicular to the plane. See Example #6.
2	Cylinder	Cone	-	Only a single Co-axial Cone and Cylinder. See Example #7.
2		Plane	-	Only a single Plane and single Cylinder that is perpendicular to the plane. See Example #6.

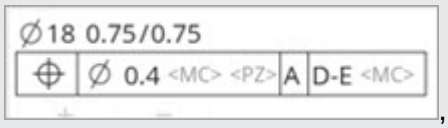
Minimum of two features		Circle	Circle or Cylinder	Only Co-axial Circles and Cylinders may be used for a primary common datum. See Example #8. This co-axiality restriction does not apply to secondary or tertiary common datums.
Minimum of two features		Cylinder or Pattern of Cylinders	Cylinder or Pattern of Cylinders	Each feature or pattern can be internal or external features of size and have different nominal sizes. See Examples #2, 4, and 5.
Minimum of two features	Pattern of Cylinders	Cylinder or Pattern of Cylinders	Cylinder or Pattern of Cylinders	
Minimum of two features	Circle	Circle or Cylinder	Circle or Cylinder	Only Co-axial Circles and Cylinders may be used for a primary common datum. See Example #8. This co-axiality restriction does not apply to secondary or tertiary common datums.

2	Cone	Cylinder	-	Only a single Co-axial Cone and Cylinder. See Example #7.
Minimum of two features	Sphere	Sphere or Pattern of Spheres	Sphere or Pattern of Spheres	Each feature or pattern must be internal or external spheres and have the same nominal size.
	Pattern of Spheres	Sphere or Pattern of Spheres	Sphere or Pattern of Spheres	

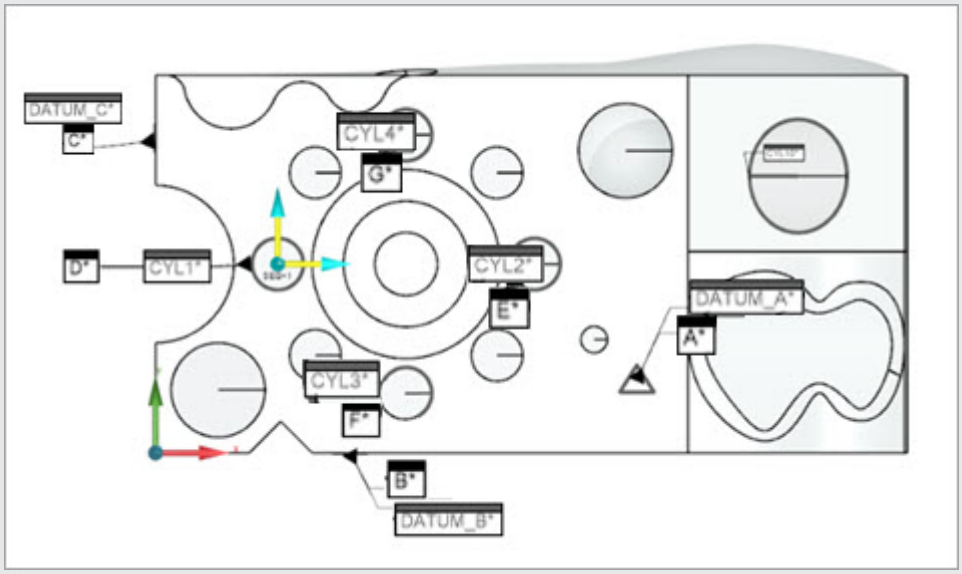
Minimum of two features	3D Width	3D Width or Pattern of 3D Widths	3D Width or Pattern of 3D Widths	Each feature or pattern can be internal or external features of size and have different nominal sizes. See Example #3. When you select multiple widths (or a pattern of widths), they need a common workplane direction to be an acceptable datum. The workplane direction is necessary to simulate the Datum Reference Frame (DRF), similar to a functional gage sliding into various widths.
	Pattern of 3D Widths	3D Width or Pattern of 3D Widths	3D Width or Pattern of 3D Widths	

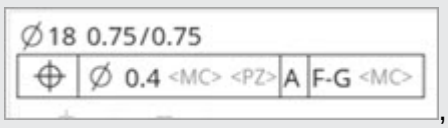
In some cases, the location of the coordinate system trihedron may be different than the axes shown on the drawing for the part. For example, the part below shows four cylindrical datums (D, E, F, and G):



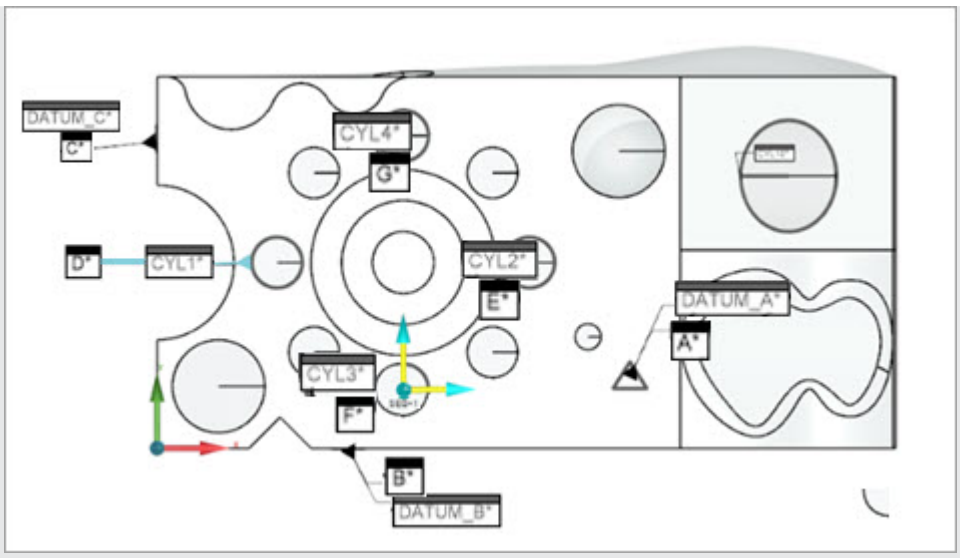
For 

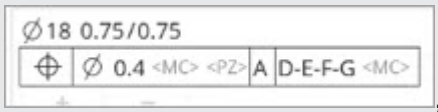
, PC-DMIS would center the trihedron on datum D as shown below:



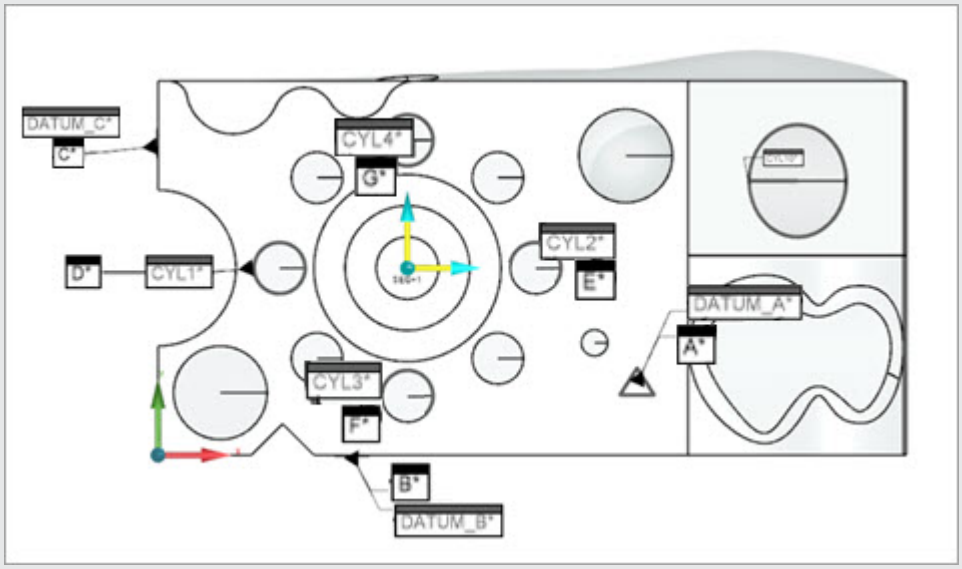
For 

, PC-DMIS would center the trihedron on datum F as shown below:



For 

, PC-DMIS would position the trihedron mid-way between all four datums (the center of the pattern), as shown below:



PC-DMIS allows you to easily change the trihedron's location and orientation. To do this, first create or select an appropriate alignment command and then change the display coordinates of the Geometric Tolerance command from **Datum Reference Frame** to **Current Alignment**. For details, see the "Display Coordinates" section of the "Nominals Tab" topic in the "Using Geometric Tolerances" chapter of the PC-DMIS Core documentation.

Any unsupported combination of features results in an error message when you attempt to create the Feature Control Frame (FCF), such as:

PC-DMIS

Multi-feature datum error. This may be caused by incorrect nominals (x, y, z or i, j, k vector) or an unsupported combination of features.

When you select datum features as a common datum, ensure that they either all have surface data, or that they all do not have surface data.

If you must combine datum features with and without surface data, it is only supported with Material Condition Modifiers (M or L). The datum math is available, but it only applies to datum features with surface data. PC-DMIS does not recalculate datum features with no surface data because they use the math from the feature command that represents the datum.

When you create common datums in the **Datum Definitions (DATDEF)** dialog box, PC-DMIS only performs a limited error check. All validity checks are performed by the Geometric Tolerance command when the FCF is built.

- Once the FCF is completely built, PC-DMIS performs the final verification that the common datum is supported. If the FCF verification fails, PC-DMIS indicates this failure with an error message.
- If the common datum has an error, please refer to the above table and general guidelines regarding Common Datums in order to address and resolve any problems with the Common Datum.
- PC-DMIS currently supports a maximum of five single datums or datum patterns combined to define a common datum (for example: A – B – C – D – E).
- PC-DMIS does not support the mixing of datum features that have surface data with datum features that do not have surface data. For more information, see "Feature Types With and Without Surface Data".

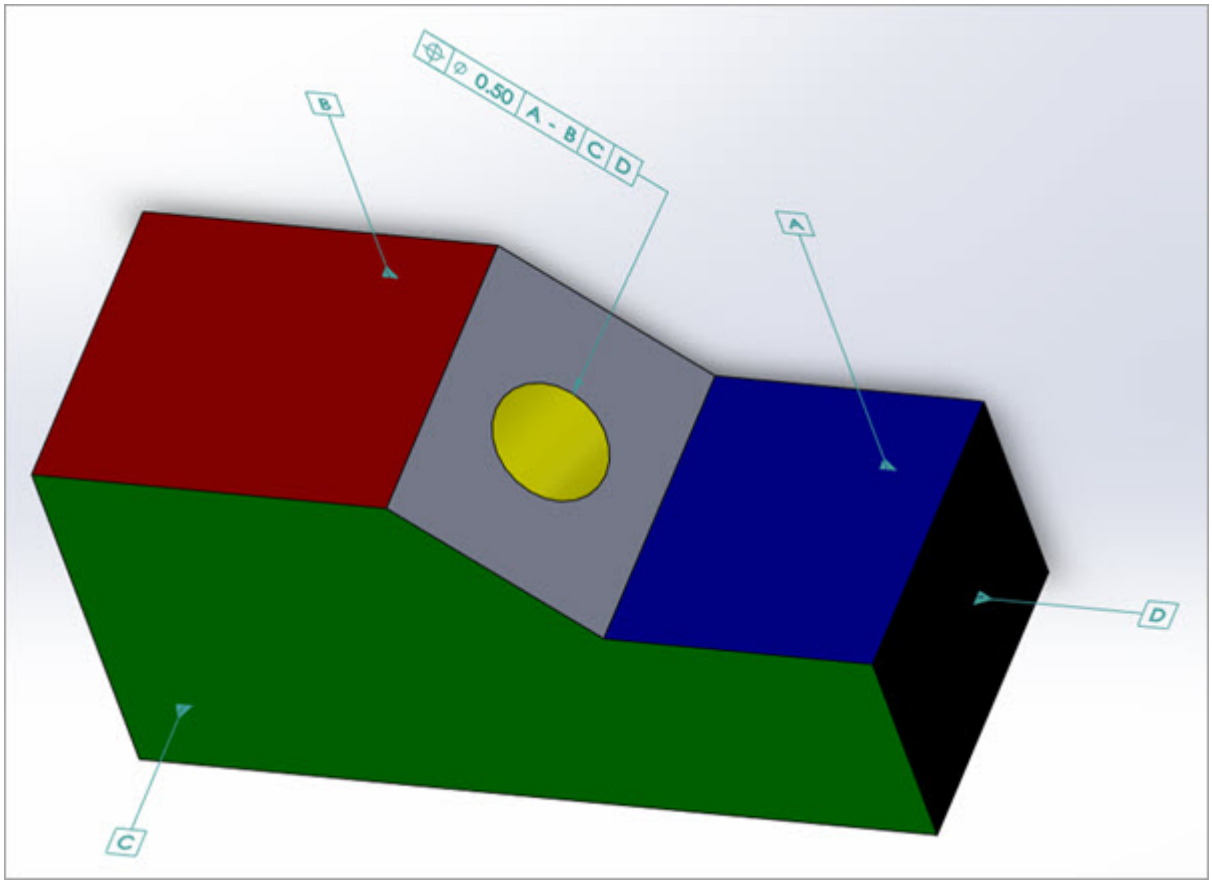
As required by ASME Y14.5 and ISO 5459, the common datum simulators for the pattern are nominally oriented and located with respect to each other. PC-DMIS supports the use of a modifier (MMB or LMB) on the common datum, when ALL the features (within the common datum) are “features of size”. When any of the features are NOT a feature of size, modifiers are not allowed.

In order for PC-DMIS to correctly grow or shrink the simulator sizes simultaneously and correctly calculate the material boundaries (when using MMB or LMB), you must have first toleranced each datum to its higher precedence datums. Also, you must make sure to include their size tolerances before you allow any other geometric tolerances to reference those datums.

That is, the tolerances on the datum must be EARLIER in the measurement routine than the geometric tolerances that reference those datums.

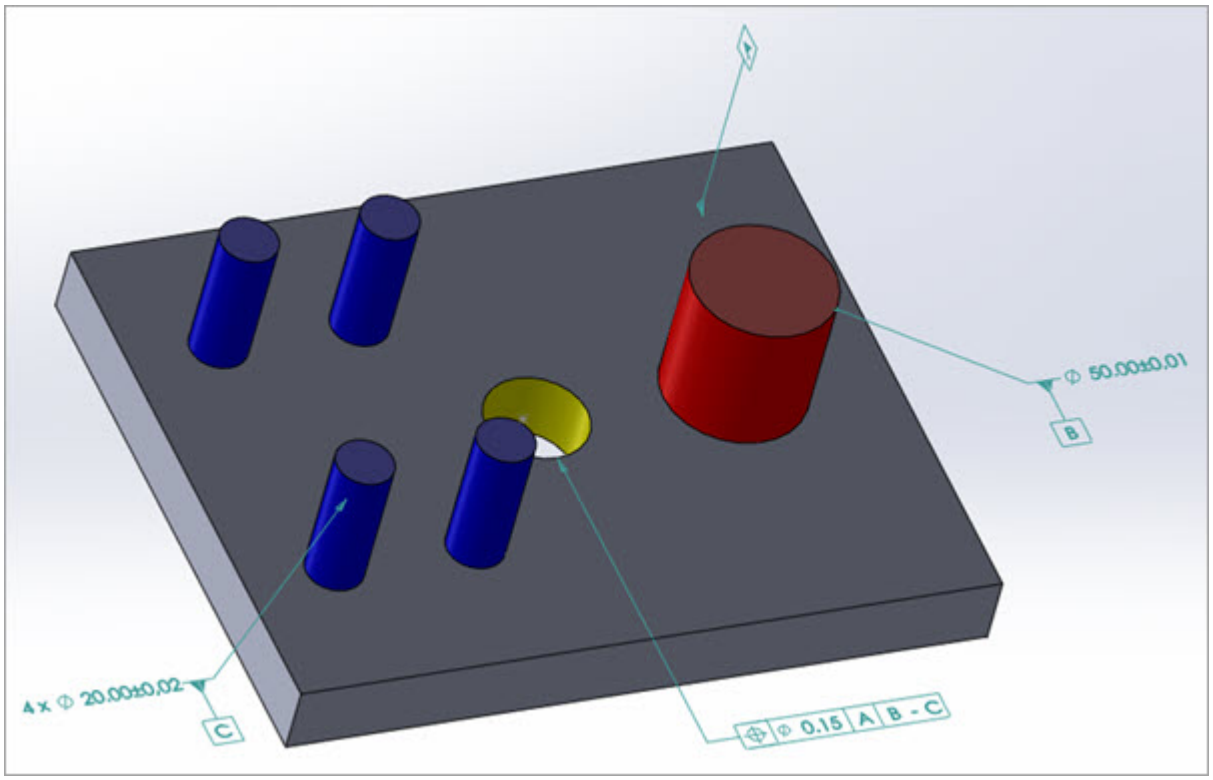
Examples of supported common datum combinations are described in the examples below.

Example 1: Plane A and Parallel Plane B as Common Datum A-B



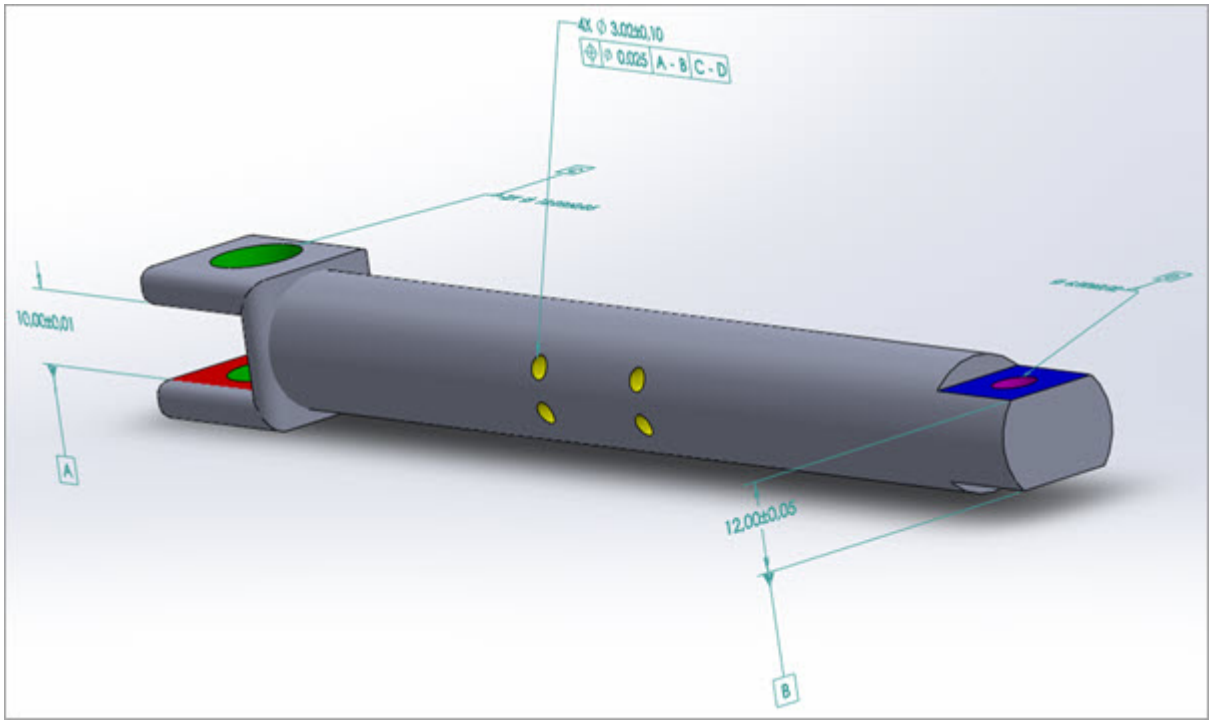
Example 1: Plane A (blue) and Parallel Plane B (red) as Common Datum A-B

Example 2: External Cylinder B with a Pattern of External Cylinders C as Common Datum B-C



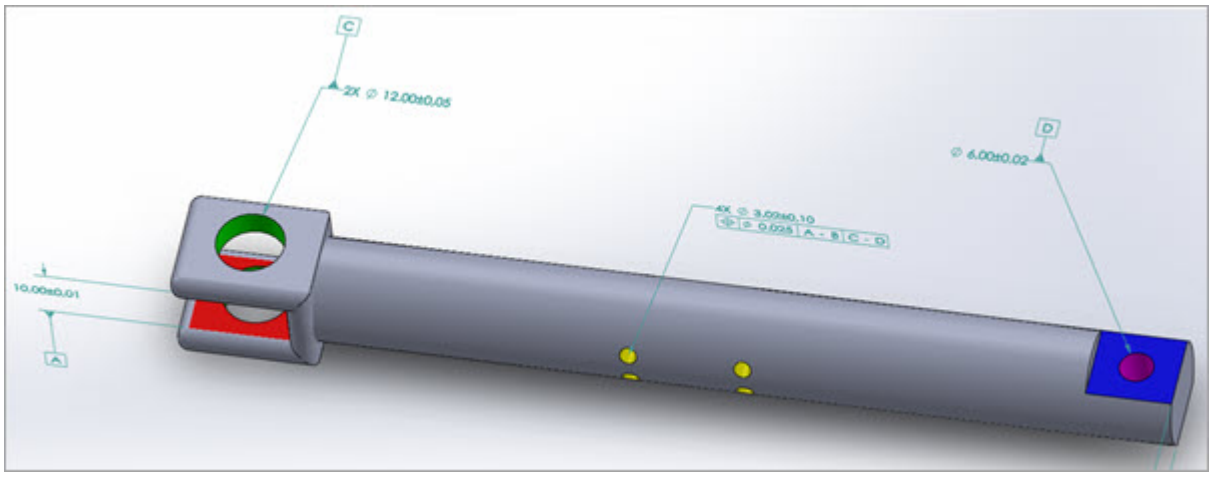
Example 2: External Cylinder B (red) with a Pattern of External Cylinders C (blue) as Common Datum B-C

Example 3: External Width A with an External Width B as Primary Common Datum A-B



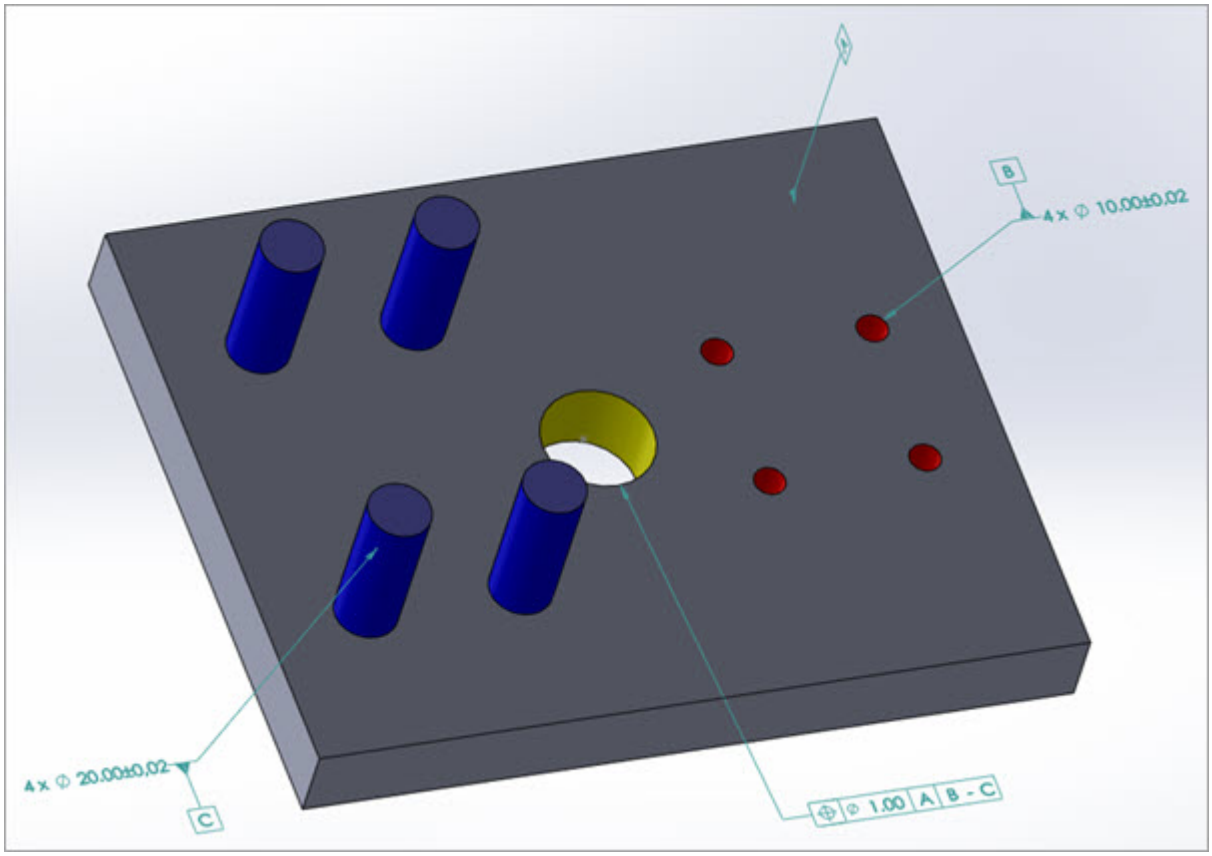
Example 3: External Width A (blue) with an External Width B (red) as Primary Common Datum A-B

Example 4: Internal Cylinder C with an Internal Cylinder D as Secondary Common Datum C-D



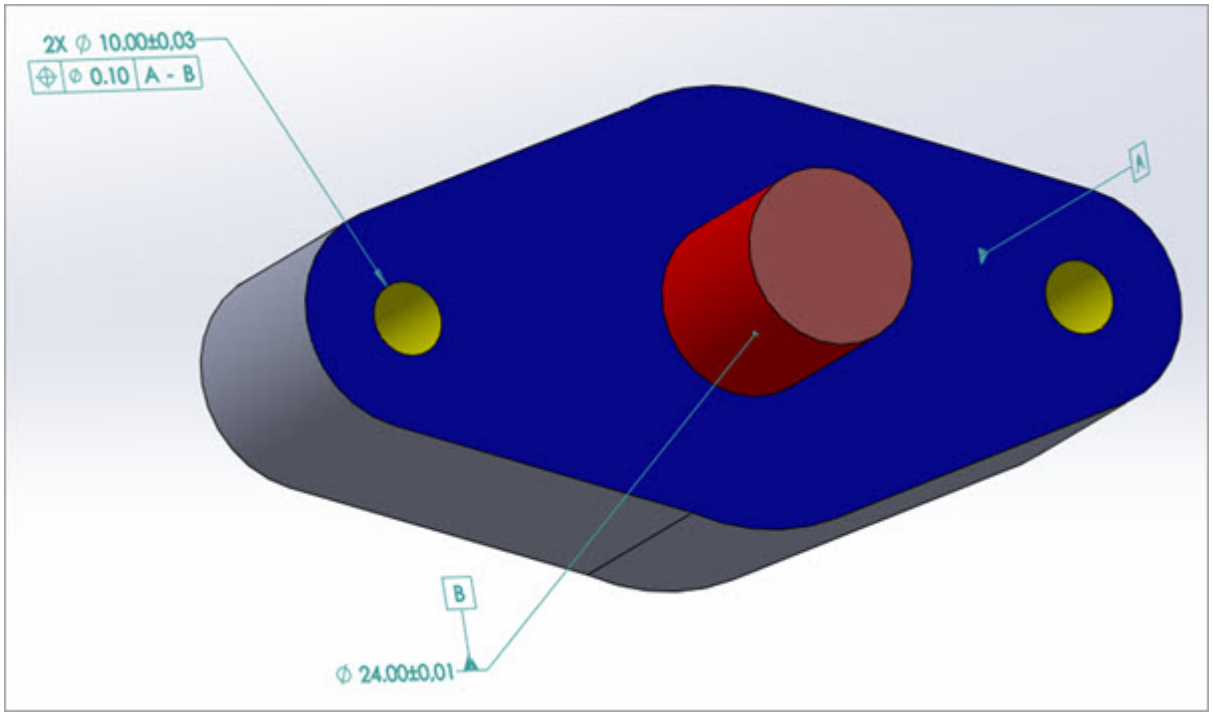
Example 4: Internal Cylinder C (green) with an Internal Cylinder D (magenta) as Secondary Common Datum C-D

Example 5: Pattern of Internal Cylinders B and Pattern of External Cylinders C as Common Datum B-C



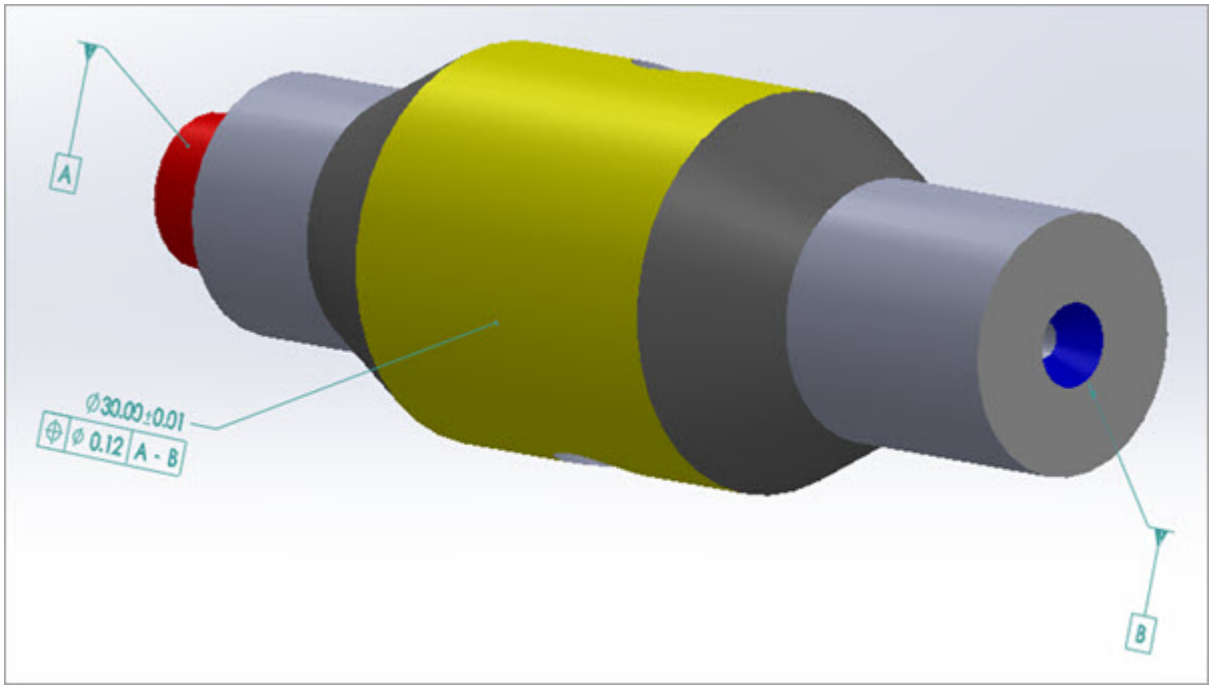
Example 5: Pattern of Internal Cylinders B (red) and Pattern of External Cylinders C (blue) as Common Datum B-C

Example 6: Plane A and External Cylinder B as Common Datum A-B



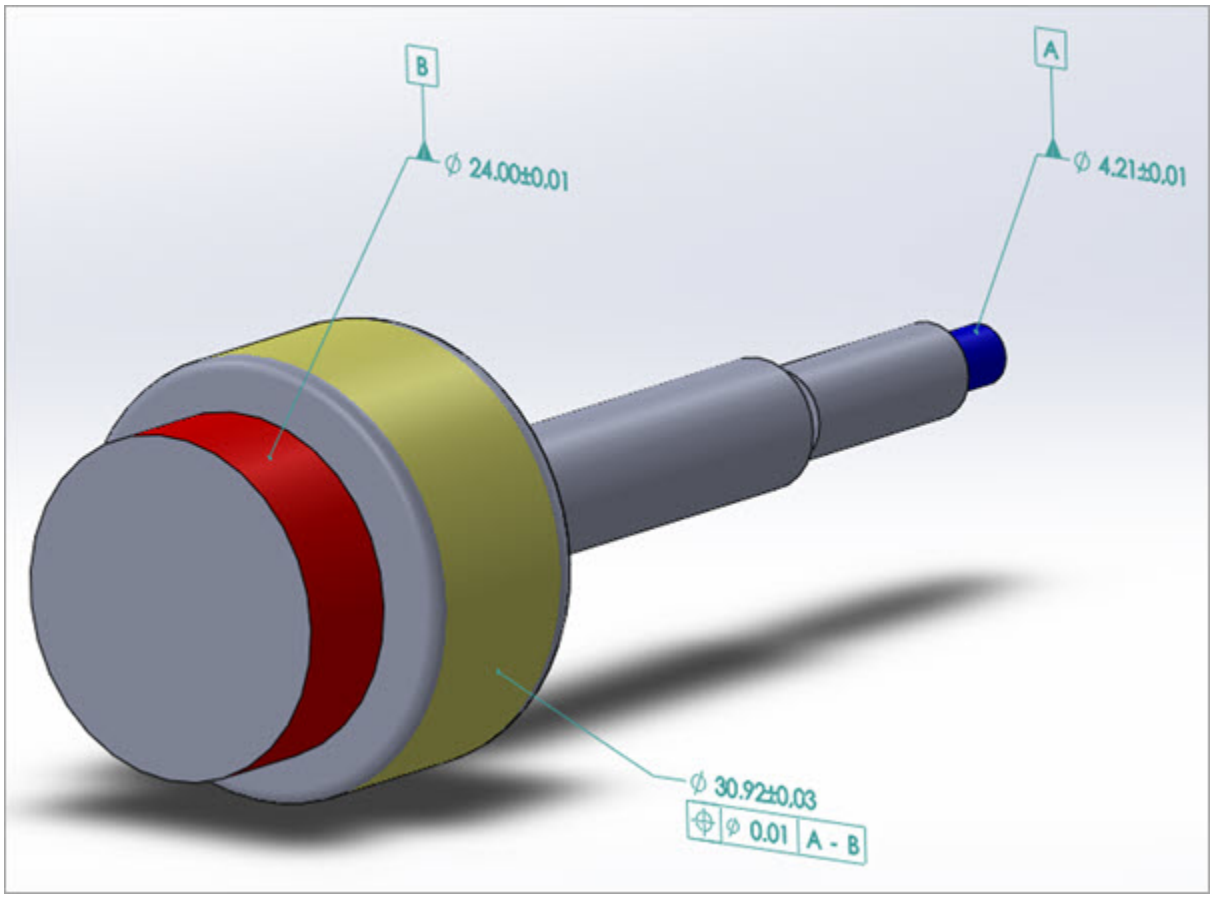
Example 6: Plane A (blue) and External Cylinder B (red) as Common Datum A-B

Example 7: External Cylinder A and Coaxial Internal Cone B as Common Datum A-B



Example 7: External Cylinder A (red) and Coaxial Internal Cone B (blue) as Common Datum A-B

Example 8: External Cylinder A and Coaxial External Cylinder B as Common Datum A-B



Example 8: External Cylinder A (blue) and Coaxial External Cylinder B (red) as Common Datum A-B

Datum Patterns

A *datum pattern* refers to a datum identifier like A or AC that refers to multiple similar features of size that all have the same size tolerance. For example, a pattern of holes might be referenced as a datum pattern B. Technically (per the ASME Y14.5 and ISO 5459 standards), a datum pattern is a type of common datum, but this documentation does not use that terminology.

For more information, see "Datum Patterns" in "How PC-DMIS Solves and Uses Datums".

Defining Geometric Tolerances and Controlling Reporting

In PC-DMIS, you can define geometric tolerances with the Geometric Tolerance command. This topic and its related topics describe how to use the **Geometric Tolerance** dialog box to define and control geometric tolerance commands. It also describes the reporting options to control the report. Finally, each geometric tolerance type is discussed in detail.

Command Mode Syntax

The Geometric Tolerance dialog box is the main way to create or modify a geometric tolerance command. For more information, see the "Geometric Tolerance Dialog Box" topic in the PC-DMIS Core documentation. However, you can also create or modify a geometric tolerance command from the Edit window's Command Mode. Below are a couple of examples to help you understand how to use the Edit window command.

Simple Example

In the Edit window's Command Mode, a relatively simple geometric tolerance might look like this:

```
  FCFCYLY1    =GEOMETRIC_TOLERANCE/STANDARD=ISO
1101,SHOWEXPANDED=NO,

            SEGMENT_1,CYLINDRICITY,0.05,,REFERENCE_FEATU
RE_MATH=DEFAULT,
              FEATURES/CYL4,,
```

There are several parts to this Edit window syntax:

- `FCFCYLY1` is the dimension ID.
- `GEOMETRIC_TOLERANCE` identifies the command as a geometric tolerance.
- `STANDARD=ISO 1101` shows that the geometric tolerance is evaluated with respect to the ISO 1101 family of standards.
- `SHOWEXPANDED=NO` indicates that the display in the Edit window is condensed and simplified (doesn't show much detail). If you set this to YES, you'll see that it shows much more information as described in the "Complex Example" topic below.
- `SEGMENT_1` begins the information about the first segment.
- `CYLINDRICITY` indicates the segment is a cylindricity tolerance.
- `0.05` is the tolerance.
- `REFERENCE_FEATURE_MATH=DEFAULT` indicates the tolerance zone math type. For details, see the "Cylindricity" topic in the PC-DMIS Core documentation.
- `FEATURES/CYL4,,` indicates the tolerance applies to the CYL4 feature.

Please notice that `SHOWEXPANDED=NO` . This hides much of the detail.

Complex Example

Here is an example of the Command Mode syntax for a more complex geometric tolerance, with `SHOWEXPANDED=YES` :

```
  FCFLOC1    =GEOMETRIC_TOLERANCE/STANDARD=ISO
1101,SHOWEXPANDED=YES,
              DESCRIPTION=ON,Multi-segment position of
4x Ø8.2mm hole pattern,

            FEATURE_MATH=MODIFIER_SELECTED,SIZE_MATH=MOD
IFIER_SELECTED,DATUM_MATH=DEFAULT,DISPLAY_COORDS=DRF,
              UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
              SIZE/NOMINAL=8.2,TOLERANCE SPECIFICATION
MODE=NOMINAL_WITH_DEVIATIONS,
              UPPER TOLERANCE=0.1,LOWER TOLERANCE=-0.1,
              UPPER_SPECIFICATION_MODIFIER=__,
              LOWER_SPECIFICATION_MODIFIER=(LP),
                CYL4:
                  MAX LOCAL SIZE:8.202,
                  MIN LOCAL SIZE:8.175,
                CYL6:
                  MAX LOCAL SIZE:8.200,
                  MIN LOCAL SIZE:8.160,
                CYL8:
                  MAX LOCAL SIZE:8.203,
                  MIN LOCAL SIZE:8.168,
                CYL10:
                  MAX LOCAL SIZE:8.209,
                  MIN LOCAL SIZE:8.166,
              SEGMENT_1,POSITION,DIAMETER,0.4,
(G),__,<len>,A,D,MMB,__,B,__,

            SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=OFF,REP
ORTGRAPH=OFF,MULT=10,
                MEASURED:
                  CYL4:0.110,
                  CYL6:0.094,
                  CYL8:0.110,
                  CYL10:0.110,
              SEGMENT_2,POSITION,COMPOSITE,DIAMETER,0.2,
(G),__,<len>,A,<dat>,<dat>,

            SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=OFF,REP
ORTGRAPH=OFF,MULT=10,
                MEASURED:
                  CYL4:0.108,
                  CYL6:0.100,
                  CYL8:0.108,
                  CYL10:0.108,
              ADD
              DATUMS/REPORTDATUMSIZE=OFF,
                D(DATUM_D):NOM=30,+Tol=0.2,-Tol=-0.2,
              FEATURES/CYL4,CYL6,CYL8,CYL10,,
```

This expanded view has the following items:

- `FCFLOC1` is the dimension ID.
- `GEOMETRIC_TOLERANCE` identifies the command as a geometric tolerance.
- `STANDARD=ISO 1101` shows that the geometric tolerance is evaluated with respect to the ISO 1101 family of standards.
- `SHOWEXPANDED=YES` indicates that the display in the Edit window is expanded (shows more detail). If you set this to NO, you'll see that it shows much less information as described in the "Simple Example" topic above.
- `DESCRIPTION=ON` indicates that the description text "Multi-segment position of 4x Ø8.2mm hole pattern" appears in the report as explained in the "Description Tab" topic in the PC-DMIS Core documentation.
- `FEATURE_MATH=MODIFIER_SELECTED` indicates that the feature math uses an ISO Associated Tolerance Feature modifier as explained in the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.
- `SIZE_MATH=MODIFIER_SELECTED` indicates that PC-DMIS calculates the size of the feature using an ISO 14405-1 size modifier as explained in the "Evaluating Size with the Geometric Tolerance Command" topic in the PC-DMIS Core documentation.
- `DATUM_MATH=DEFAULT` indicates that the datum math uses the default method — a void filtration and a constrained least squares best fit — as explained in the "How PC-DMIS Solves and Uses Datums" topic.
- `DISPLAY_COORDS=DRF` indicates that the results are reported in datum reference frame coordinates (as opposed to current alignment coordinates).
- `UNITS=MM` indicates the dimension units are millimeters.
- `OUTPUT=BOTH` indicates that the results are sent to both the statistics and the report.
- `ARROWDENSITY=100` is the arrow density used with graphical analysis.

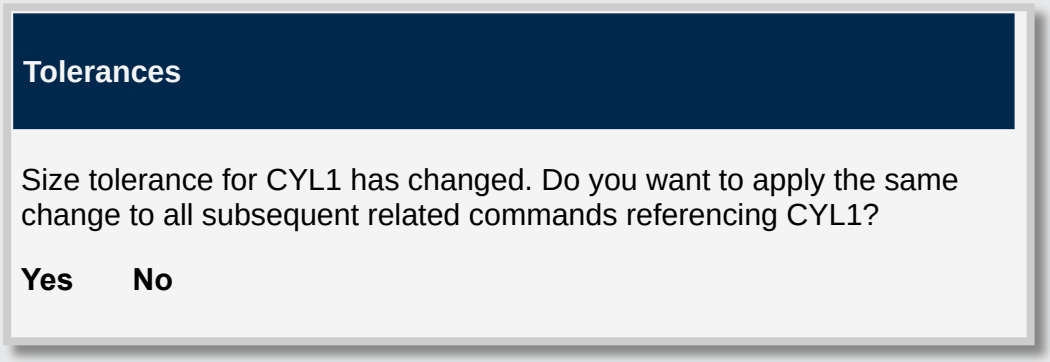
Alternate Command Block 1

```
SIZE/NOMINAL=8.2,TOLERANCE SPECIFICATION
MODE=NOMINAL_WITH_DEVIATIONS,
     UPPER TOLERANCE=0.1,LOWER TOLERANCE=-0.1,
     UPPER_SPECIFICATION_MODIFIER=__,
     LOWER_SPECIFICATION_MODIFIER=(LP),
     CYL4:
       MAX LOCAL SIZE:8.202,
       MIN LOCAL SIZE:8.175,
    CYL6:
       MAX LOCAL SIZE:8.200,
       MIN LOCAL SIZE:8.160,
    CYL8:
       MAX LOCAL SIZE:8.203,
       MIN LOCAL SIZE:8.168,
    CYL10:
       MAX LOCAL SIZE:8.209,
       MIN LOCAL SIZE:8.166,
```

This command block represents the size tolerance, including the nominal size, upper tolerance, lower tolerance, and the upper and lower specification modifiers (if selected). In this case, the (LP) modifier specifies the max and min two-point local size. The respective measured values are then listed below for each of the four features.

If you edit a feature's upper or lower tolerances from the Edit window or from the **Geometric Tolerance** dialog box (**Feature Control Frame** or **Nominals** tab), and the same feature is used as a datum or as a considered feature, PC-DMIS displays a message asking if you want to apply the same changes to all subsequent commands referencing that feature.

For example:



If you click **Yes**, PC-DMIS updates the size tolerances for any Geometric Tolerance commands below the cursor position that references the same feature, either as a considered feature or datum.

If you click **No**, PC-DMIS updates the edited size tolerance only. PC-DMIS does not update any of the respective size tolerances for any related Geometric Tolerance commands below the cursor position that uses the same edited feature either as a considered feature or datum.

Alternate Command Block 2

```
SEGMENT_1,POSITION,DIAMETER,0.4,
(G),__,<len>,A,D,MMB,__,B,__,
SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF
,MULT=10,
    MEASURED:
      CYL4:0.110,
      CYL6:0.094,
      CYL8:0.110,
      CYL10:0.110,
```

This command block represents the first segment, which is a position tolerance with a diametric tolerance zone, 0.4 tolerance, a (G) modifier, and a datum reference frame that consists of A | D MMB | B. Show optimized shift is off, the textual analysis is off, the CAD graphical analysis is off, the report graphical analysis is off, and the arrow multiplier is 10. It also includes the measured position values of each of the four features.

Alternate Command Block 3

```
SEGMENT_2,POSITION,COMPOSITE,DIAMETER,0.2,
(G),__,<len>,A,<dat>,<dat>,

   SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=
OFF,MULT=10,
     MEASURED:
        CYL4:0.108,
        CYL6:0.100,
        CYL8:0.108,
        CYL10:0.108,
```

This command block represents the second segment, which is a lower segment of a composite position tolerance having a diametric tolerance zone, 0.2 tolerance, a (G) modifier, and a datum reference frame that consists of A. Show optimized shift is off, the textual analysis is off, the CAD graphical analysis is off, the report graphical analysis is off, and the arrow multiplier is 10. The command block also includes the measured position values of each of the four features.

Alternate Command Block 4

```
ADD
     DATUMS/REPORTDATUMSIZE=OFF,
        D(CYL2):NOM=30,+TOL=0.2,-TOL=-0.2,
     FEATURES/CYL4,CYL6,CYL8,CYL10,,
```

- `ADD` is a control that you can use to add another segment to the composite position tolerance. To use it, hover over the `ADD` command for couple of seconds, then click once on it, and then click the **ADD** button that appears.
- `DATUMS/REPORTDATUMSIZE=OFF,`

  `D(CYL2):NOM=30,+Tol=0.2,-Tol=-0.2,`

  This part of the command indicates that the measured datum sizes are not included in the report. It also shows the size tolerance on datum D (which is CYL2). The size tolerance on datum features can be important in several situations, such as datums referenced with a material modifier Ⓜ or Ⓛ, and datum patterns with or without a modifier. We recommend that you always make sure that the size tolerances on the datum features are correct. For details, see the "Determining the Size of the Material Boundary" section in the "How PC-DMIS Solves and Uses Datums" topic in the PC-DMIS Core documentation.
- `FEATURES/CYL4,CYL6,CYL8,CYL10,,`

This part of the command indicates the composite position tolerance applies to the CYL4, CYL6, CYL8, and CYL10 features.

Geometric Tolerance Dialog Box

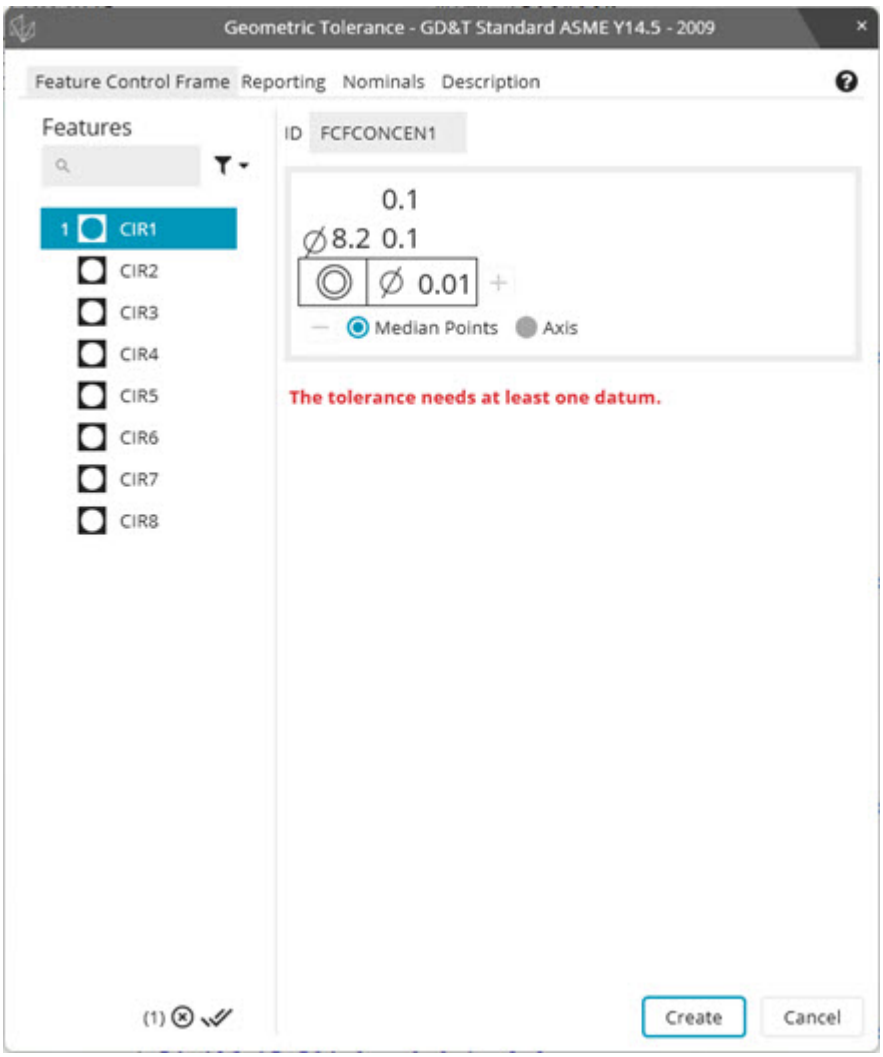
The **Geometric Tolerance** dialog box is the main way to create or modify the geometric tolerance command. To use this dialog box to create a geometric tolerance, select **Insert | Dimension | <geometric tolerance type>** from the menu, or you can select a geometric tolerance type from the **Dimension** toolbar.

The geometric tolerance types are:

- Angularity
- Circular Runout
- Circularity
- Concentricity
- Cylindricity
- Flatness
- Parallelism
- Perpendicularity
- Position
- Profile of a Line
- Profile of a Surface
- Straightness
- Symmetry
- Total Runout

The other tolerance types (Location, Angle, and so on) are not geometric tolerances and so are not handled via the geometric tolerance command.

After you select **Insert | Dimension | <tolerance type>** from the menu (or from the **Dimension** toolbar), PC-DMIS shows the **Geometric Tolerance** dialog box:



This dialog box starts out nearly empty. In the example image above, the dialog box has these properties:

- The default dimension ID is FCFCONCEN2
- The symbol from the selected tolerance type is Concentricity
- The recently-used tolerance value is 0.01
- The red error message indicates that you haven't selected any datum references yet

We recommend this workflow when you use the **Geometric Tolerance** dialog box:

1. Choose your tolerance type from the **Insert | Dimension | <tolerance type>** menu (or from the **Dimension** toolbar) to open the **Geometric Tolerance** dialog box.
2. From the list of features, choose your considered features for the tolerance. The list only shows those features above your pointer's current position in the Edit window.
3. Edit the feature control frame. To do this, add symbols, modifiers, segments, zone orientation specifications, and so forth.
4. Click the **Reporting** tab and make sure all options are set as desired.

5. Click the **Nominals** tab (when present) and make sure all settings are set as desired.
6. Click the **Description** tab and click the **Add** button to add descriptive information as needed. Click the **Show on report** check box if you want to add the text in your reports.
7. Click **Create** to create the geometric tolerance command in your measurement routine.

In the Edit window, you can press F9 at any time on that command to edit that command in the **Geometric Tolerance** dialog box. If you edit an existing geometric tolerance, the **Geometric Tolerance** dialog has an **OK** button instead of a **Create** button.

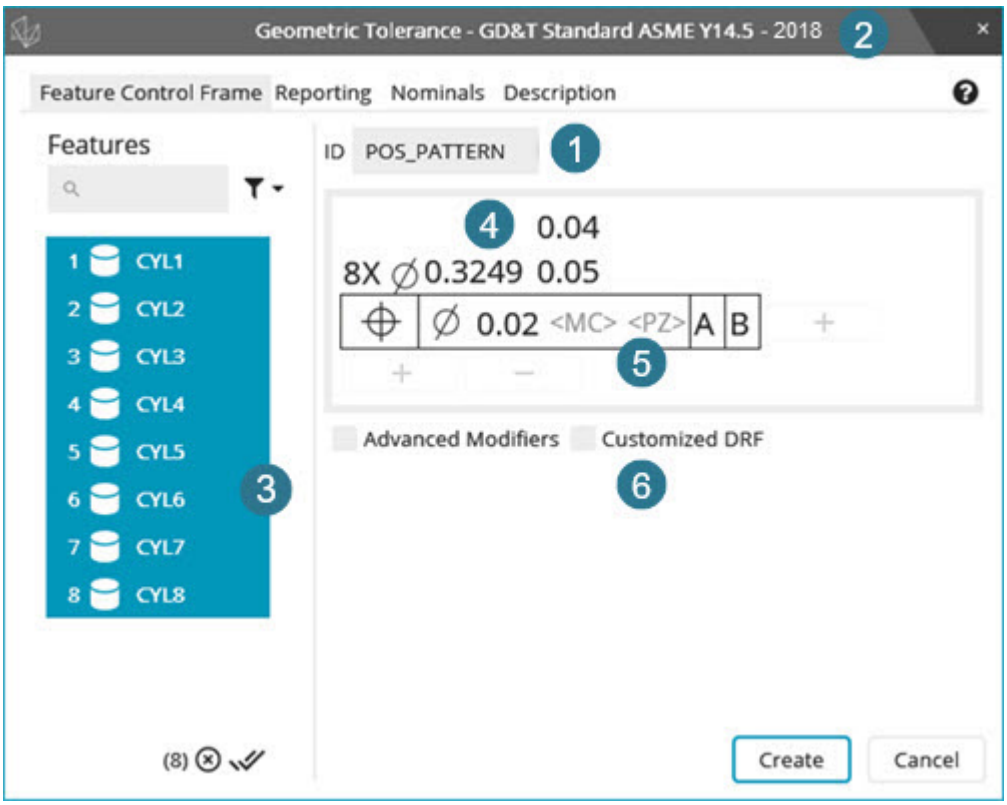
Feature Control Frame Tab

In this Topic

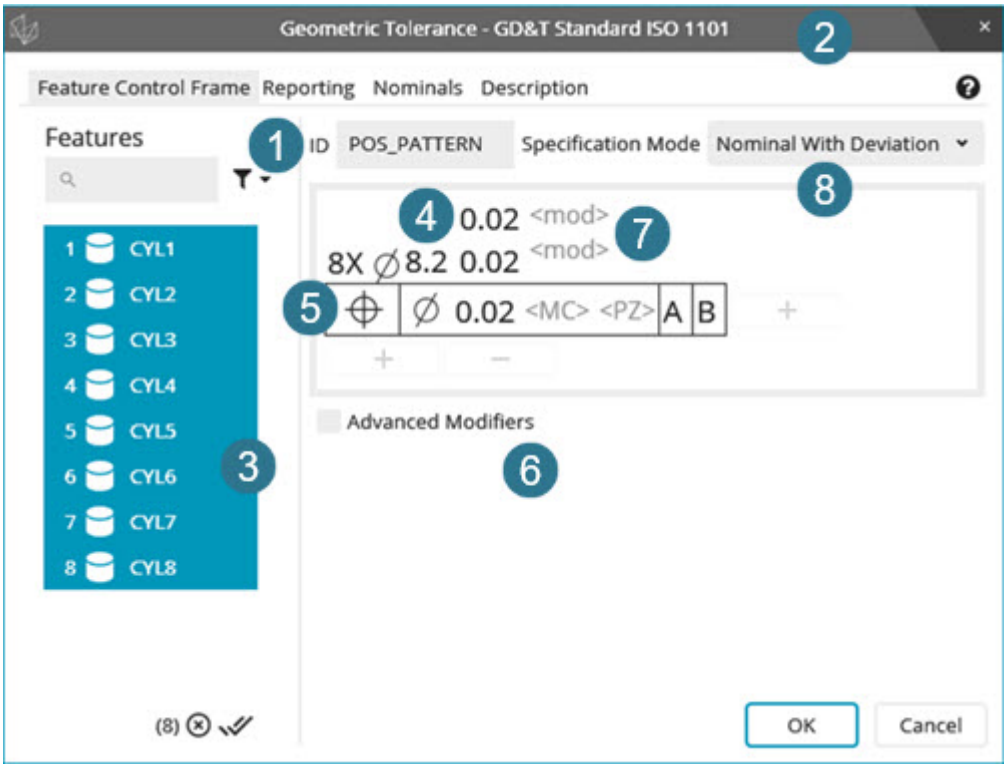
- Feature Control Frame Tab
  - Introduction
  - The Feature List
  - The Size Tolerance Editor
  - The Feature Control Frame Editor
  - The Tolerance Symbol
  - The Zone, Feature, and Characteristic Section
  - Datum Section
  - The Add-Datum Button
  - Modifying a Datum Reference
  - Datum Modifiers
- Additional Options
  - Advanced Modifiers check box
  - Customized DRF check box
  - Zone Orientation
  - Unit Orientation
  - Cone Half Angle
  - Circularity versus Conicity
  - Lengthwise versus Widthwise Slots
  - Median Points versus Axis
  - Adding and Removing Segments
  - The Information Section
  - Specification Mode

Introduction

The **Feature Control Frame** tab of the **Geometric Tolerance** dialog box is where most of your editing takes place. A typical position tolerance might look like this in the dialog box:



The ASME version of the Geometric Tolerance dialog box showing the Feature Control Frame tab



The ISO version of the Geometric Tolerance dialog box showing the Feature Control Frame tab

1. **ID** - This box defines the dimension's ID. In the picture above, it was edited to say POS_PATTERN. For the position tolerance, the **ID** defaults to something like

FCFLOC1, 2, 3... etc. if you do not edit it. For the rules that apply when you type an ID, see "Setup Options: ID Setup tab" in the "Setting Your Preferences" chapter.

2. **GD&T Standard** - This shows the standard to use in the tolerance. It should match the standard used by your print. We support prints based on ASME Y14.5 and ISO 1101; the specific versions of the standards we support (including supporting standards like ISO 5459) are detailed in the "Introduction to Geometric Tolerances and Feature Control Frames" topic.

3. **Feature List** - This list shows the available features for the type of geometric tolerance. For more information, see "The Feature List" below.

4. **Size Tolerance Editor** - The first row in the tolerance editing pane shows the size and plus and minus tolerance information. For more information, see "The Size Tolerance Editor" below.

5. **Feature Control Frame Editor** - The second row in the tolerance editing pane is the main editing area. For more information, see "The Feature Control Frame Editor" below.

6. **Additional Options** - This area of the dialog box contains advanced and other options for your tolerance. For more information, see "Additional Options" below.

7. **Upper and Lower Modifiers** (ISO only) - This area of the dialog box allows you to select the appropriate ISO Size modifiers. For details, see the "ISO Size Modifiers" section below.

8. **Specification Mode** (ISO only) - This list allows you to select the ISO specification mode. The options are **Nominal with Deviation** or **ISO Code**. For more details, see the "Specification Mode" section below.

The Feature List

The first thing to do when you start to create a geometric tolerance is to select the considered feature or features. When the dialog box opens, no features are selected. Every feature in the measurement routine that is allowed for the tolerance type is shown. If you don't see an expected feature, make sure your pointer in the Edit window is below that feature before you access the **Geometric Tolerance** dialog box.

In large routines with many features, you may find it helpful to search for a feature with the search bar.

After you select one feature, PC-DMIS filters the feature list so that only similar features or features with the same characteristics are shown (such as cylinders with the same diameter). You may then select additional features at that point.

The Size Tolerance Editor

The first row of the tolerance editing pane is the Size Tolerance Editor. It is available when your considered features are features of size (cylinders, circles measured on a surface, spheres, or widths) and your geometric tolerance type allows size tolerances.

The Size Tolerance Editor looks like this:

This line contains several types of information and two controls:

- 8X means there are eight considered features; it is not visible when there is only one considered feature. You cannot edit the symbol.
- ∅ means the features are cylinders or circles measured on a surface. The symbol is different for different kinds of features of size. Spherical features have a S∅ symbol. Width features do not have a symbol. You cannot edit the symbol.
- The first number after the above symbol is the nominal size. You cannot edit this value because the geometric tolerance command requires all features to have correct nominals. The nominal size comes from the feature's THEO (theoretical) size.
- The second number, the one on the far right on top (0.04 in the above example), is the plus tolerance for the feature's size. You can edit it. The upper limit of size is equal to the nominal size plus the plus tolerance.
- The third (and final) number, the one on the far right on bottom (0.05 in the above example), is the minus tolerance for the feature's size. You can edit it. The lower limit of size is equal to the nominal size minus the minus tolerance, when the **Minus tols show negative** check box in the **Dimension** tab of the **Setup Options** dialog box is cleared. If you mark **Minus tols show negative**, then the lower limit of size is equal to the nominal size plus the minus tolerance. You may prefer to mark this check box to make the size tolerance look like your print. For information on this check box, see the "Minus Tols Show Negative" topic in the "Setting Your Preferences" chapter. This check box exists so you don't have to type the minus sign for minus tolerances.

The Feature Control Frame Editor

The most complex part of the tolerance editing pane is the Feature Control Frame Editor. It looks like this:

There are several parts to this editor, which are detailed below.

The Tolerance Symbol

The tolerance symbol shows up in the left-most box of the Feature Control Frame Editor. It shows the symbol for the tolerance type you chose. You can change the symbol to change the tolerance type. You click the tolerance symbol to choose another valid geometric type for the features that you selected; the other symbols appear in a drop-down window.

The Zone, Feature, and Characteristic Section

The zone, feature, and characteristic section of the feature control frame includes the tolerance zone shape, the tolerance value, and any tolerance modifiers. Therefore, this section contains several controls. Their availability varies depending on what tolerance type and what considered features are chosen.

- The tolerance zone shape symbol is first. It is Ø for diametric zones, SØ for spherical zones, and blank for planar zones (and for radial arc and right angle to radial zones). If you click the symbol, you can edit which symbol appears, but the only available symbols are those that make sense for the considered features and the tolerance type you chose.

- The tolerance value is second. It is a number (0.02 in the above picture). You may edit it to any positive value.

- When it makes sense for your considered features and tolerance type, the material condition control is next. It appears as <MC> when there is no material condition modifier. That means the feature is referenced at regardless of feature size (RFS). When there is a maximum material condition (MMC) modifier, it appears as Ⓜ. When there is a least material condition (LMC) modifier, it appears as Ⓛ. Click the <MC>, Ⓜ, or Ⓛ to switch between no material condition modifier, the MMC modifier, or the LMC modifier. ISO orientation tolerances (angularity, parallelism, and perpendicularity) and location tolerances (position, concentricity, and symmetry) allow the selection of an Associated Toleranced Feature Modifier within this control. ISO form tolerances (circularity, cylindricity, flatness and straightness of a surface) allow the selection of a Reference Feature Association modifier within this control.

  > **!** You cannot combine an Associated Toleranced Feature Modifier (ATFM) or a Reference Feature Association Modifier (RFAM) with a material condition modifier (MMC/LMC).

  For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

- When it makes sense for your considered features and tolerance type, the projected zone control is next. It appears as <PZ> when there is no projected zone modifier. When there is a projected zone modifier, it appears as Ⓟ with a

projected length number that immediately follows it, like this: Ⓟ 0.8. Click the projected length number to edit it. Click the <PZ> or Ⓟ to switch between no projected zone modifier or the projected zone modifier.

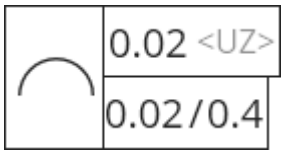
- When it makes sense for your considered features and tolerance type, the tangent plane control is next. It appears as <T> when there is no tangent plane modifier. When there is a tangent plane modifier it appears as Ⓣ. Click the <T> or Ⓣ to switch between no tangent plane modifier or the tangent plane modifier.

> The tangent plane modifier is not available for ASME Y14.5 1994.

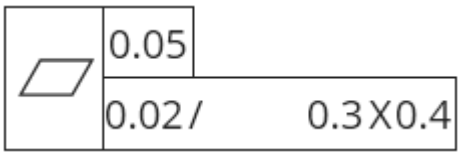
- For profile tolerances, the profile modifier control appears after the tolerance. It appears as <UZ> when there is no profile modifier. When there is an ASME unequal disposition modifier, it appears as Ⓤ with an unequal disposition distance that immediately follows it, like this: Ⓤ 0.02. When there is an ISO unequal disposition modifier, it appears as UZ with an unequal disposition distance that immediately follows it, like this: UZ 0.02. When there is an ASME dynamic profile modifier, it appears as △. When there is an ISO offset zone modifier, it appears as OZ.

> The tangent plane modifier Ⓣ and unequal disposition modifier Ⓤ are not available for ASME Y14.5 1994.
>
> The dynamic profile modifier △ is not available for ASME Y14.5 1994 or ASME Y14.5 2009.

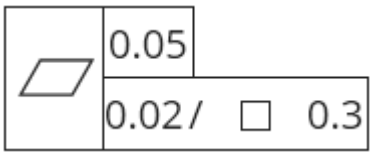
- For per-unit straightness and per-unit profile of a line, you can edit the per-unit length right after the / symbol, as shown in the lower segment below:



- For per-unit flatness, to the right of the / symbol, there is the unit shape control. It appears blank when the unit is rectangular, and it appears as a square □ when the unit shape is square. Rectangular unit shapes give you a length and a width to edit, while square unit shapes give you only a length. Click the blank or the square to switch between the two.



Rectangular Unit Shape



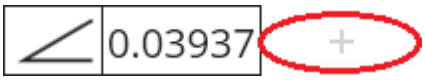
Square Unit Shape

Datum Section

The datum section of a feature control frame (or of a tolerance indicator) is just to the right of the zone, feature, and characteristic section. This consists of zero to three boxes. Each box contains a datum reference (potentially a common datum reference) and zero or more datum modifiers.

The Add-Datum Button

The **+** button to the right of the Feature Control Frame Editor lets you add datums:

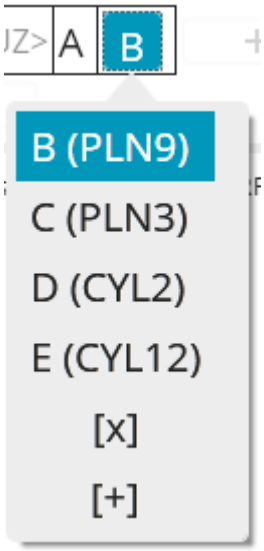


When you click this button, PC-DMIS adds a pre-defined datum to your feature control frame. Each time you click it, it adds a pre-defined datum that has not already been added to this feature control frame.

If no pre-defined datum exists that has not already been added to the feature control frame (or if no pre-defined datums exist at all), then this button shows the **Datum Definition** dialog box. You can use that dialog box to define a datum. (For information on the dialog box, see "Dialog Box Usage and Command Syntax".) This inserts a datum definition command just above your geometric tolerance command. However, the new datum is not automatically selected for you. This means you need to click the add-datum button (**+**) again to select the new datum reference.

Modifying a Datum Reference

To change a datum reference inside a datum box, click the datum reference. A drop-down menu appears from which you can choose a new reference:



From the drop-down menu, you can select a pre-defined datum reference, where the feature name is shown in parentheses next to the datum reference (datum patterns only show one feature name, and common datums don't show any feature names).

If you choose the [x] option, the datum box is deleted.

If you choose the [+] option, the **Datum Definition** dialog box opens up, so that you can define a new datum. (For information on the dialog box, see "Dialog Box Usage and Command Syntax".) However, the new datum is not automatically selected for you. This means you need to click the old datum reference from the datum box and then choose the new datum reference.

Datum Modifiers

To the right of each datum reference, there are the controls for any datum modifiers.

- When it makes sense for your datum and tolerance type, the material boundary control is first. It appears as <MC> when you reference the datum without a material modifier. That means the datum is referenced at regardless of material boundary (RMB). When there is a maximum material boundary (MMB) modifier, it appears as Ⓜ. When there is a least material boundary (LMB) modifier, it appears as Ⓛ. Click the <MC>, Ⓜ, or Ⓛ to switch between no material modifier, the MMB modifier, or the LMB modifier.

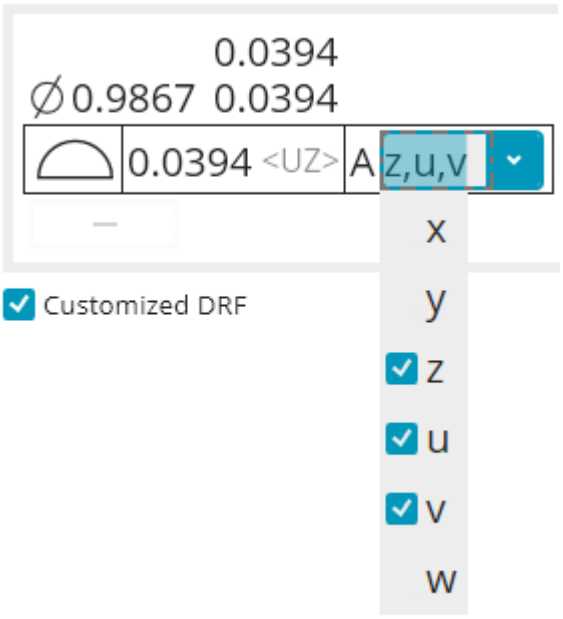
- When a datum is referenced at MMB or LMB, and you marked the **Advanced Modifiers** check box, you can specify a material boundary size. This is not the nominal size of the datum feature, nor is it the maximum or minimum material condition size of the datum feature. For more information about material boundary sizes, see the "Determining the Size of the Material Boundary" sub-topic of the "How PC-DMIS Solves and Uses Datums" topic. For datum cylinders, this modifier looks like this

if you haven't specified a material boundary size: [Ø<size>]. After you specify a material boundary size, the modifier looks like this: [Ø9.2], with whatever size you specified. To remove a specified material boundary size, delete the size value and then press tab to move off the modifier. For more information, see the "Additional Options" section below.

- When the datum is a secondary or tertiary datum, and you marked the **Advanced Modifiers** check box, the translation control appears next. It appears as <TR> when the datum is referenced without a translation modifier. When there is a translation modifier, it appears as ▷. Click the <TR> or ▷ to switch between no translation modifier and the translation modifier. For more information, see the "Additional Options" section below.

> The translation modifier is not available for ASME Y14.5 1994.

- Position and profile ASME tolerances have a **Customized DRF** check box. For more information, see the "Additional Options" section below.



If you mark this check box, PC-DMIS enables access to customized datum reference frames and removes access to advanced datum modifiers. Wherever possible, PC-DMIS will determine the degrees of freedom that each datum would naturally constrain. Where it is not possible for the software to determine which degrees of freedom to constrain, the customization will show <DOF> and you must make a manual selection. The software clears this check box by default because, more often than not, you do not need customized datum reference frames.

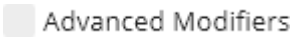
Customized DRF's are not available for ASME Y14.5 1994.

For more information on customized datum reference frames and their associated rules, please refer to ASME Y14.5 2018 section 7.22 and figures 7-55, 7-56 & 7-57.

Additional Options

Advanced Modifiers check box

This check box is not available if you select ASME Y14.5 1994 as your GD&T standard. For all other GD&T standard selections, you can find it under the tolerance editing pane as shown here:



Position and profile tolerances have an **Advanced Modifiers** check box. If you mark this check box, PC-DMIS enables access to translation modifiers (ASME), specified material boundary modifiers (ASME), and the [DF] modifier (ISO). It also removes access to customized datum reference frames. This check box is cleared by default because most users do not need these advanced modifiers.

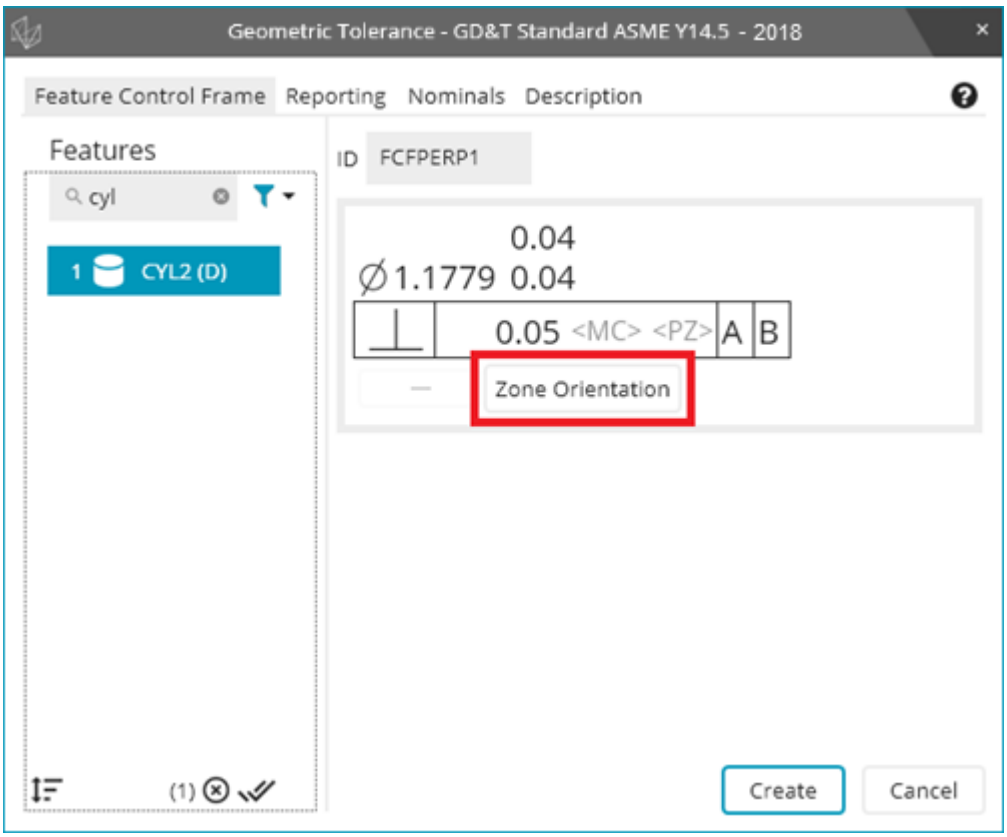
Customized DRF check box

This check box is not available if you select ASME Y14.5 1994 or ISO 1101 2012/2017 as your GD&T standard. For all other GD&T standard selections, you can find it under the tolerance editing pane as shown here:

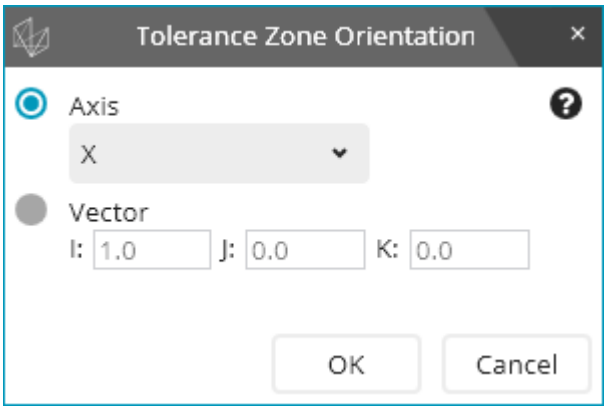
Position and profile ASME tolerances have a **Customized DRF** check box. If you mark this check box, PC-DMIS enables access to customized datum reference frames and removes access to advanced datum modifiers. This check box is cleared by default because most users do not need customized datum reference frames.

Zone Orientation

When it makes sense for the chosen tolerance zone shape symbol, tolerance type, and considered features, the **Zone Orientation** button becomes visible:



If you click the **Zone Orientation** button, the **Tolerance Zone Orientation** dialog box appears. This dialog box allows you to control the orientation of the tolerance zone:



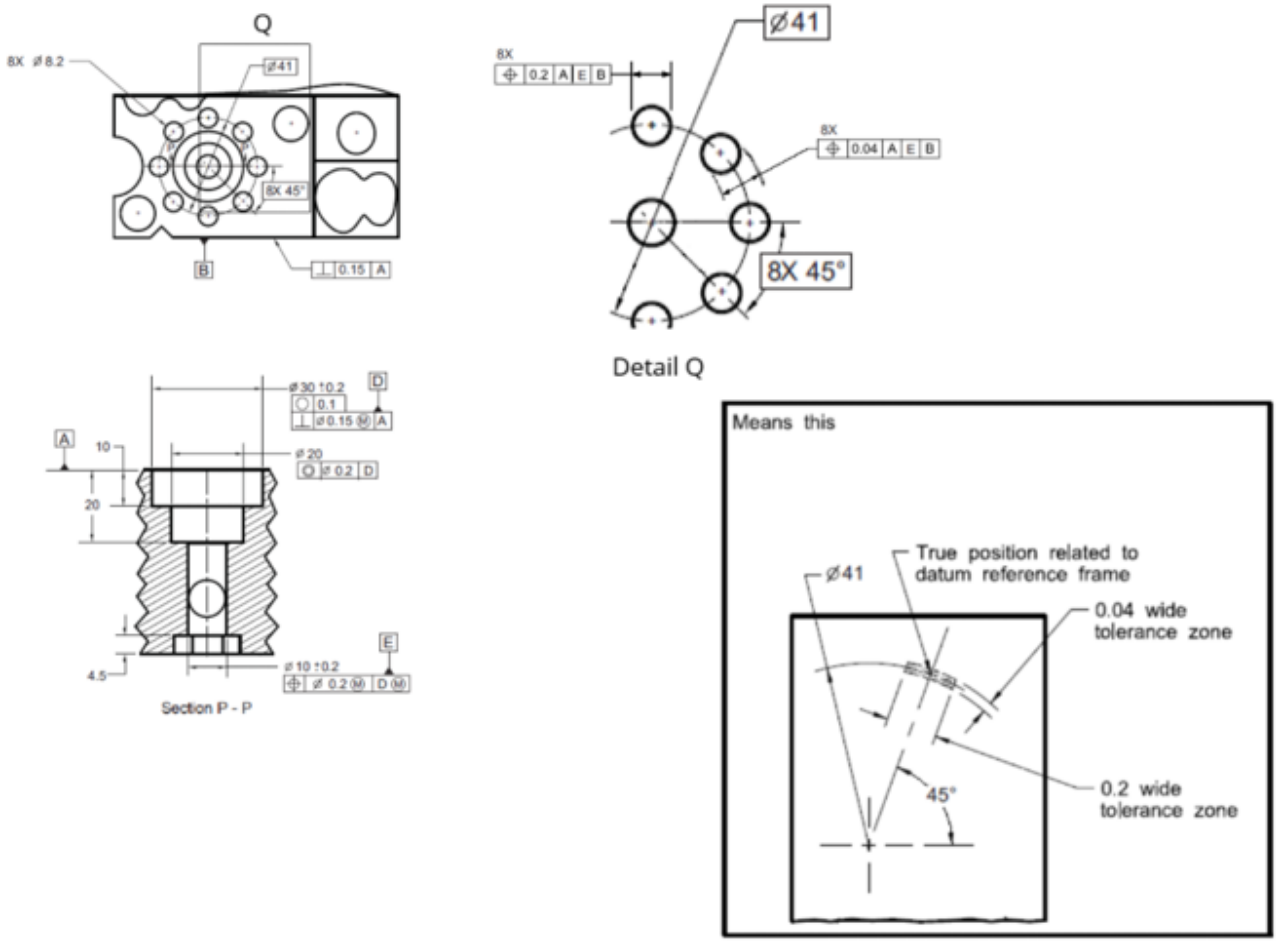
The **Tolerance Zone Orientation** dialog allows you to define the surface normal vector of the planar tolerance zone, or the axis vector of the diametric tolerance zone. The **Axis** drop-down list is for when the vector is along the X, Y, or Z axes (respectively). Alternatively, you can select an arbitrary vector with the **Vector** option and boxes under it.

As an example, if a position tolerance controls the X component of the position (planar tolerance zone), the tolerance zone surface normal vector should be X.

The zone orientation vector is always in part coordinates, never datum reference frame coordinates. It is also always normalized (has length equal to 1) and always compatible with the orientation of the considered feature.

To select a polar tolerance zone (radial arc or right angle to radial), click the **Zone Orientation** button, and from the **Axis** drop-down list, select **Radial Arc** or **Right Angle to Radial**.

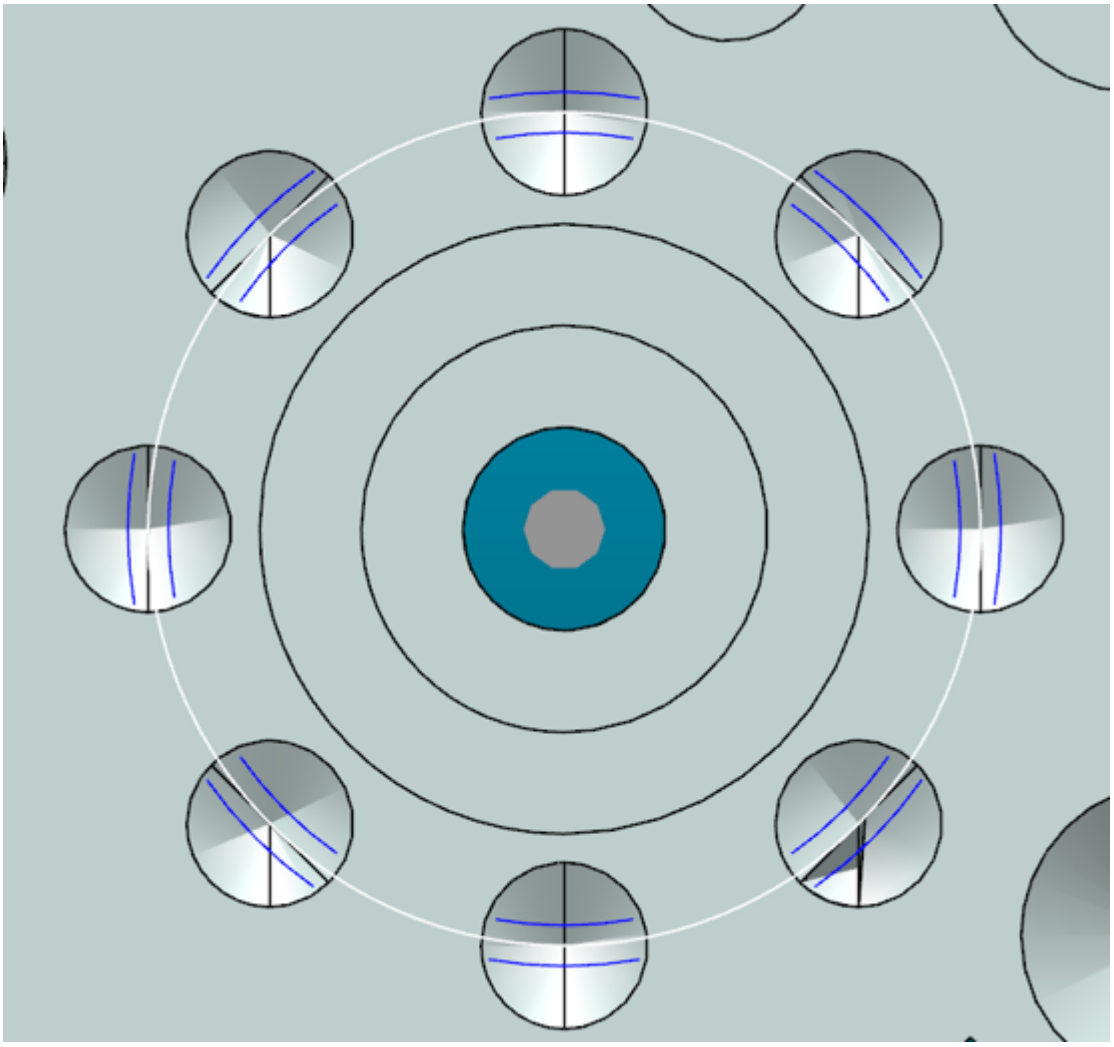
Consider the following requirement on a drawing:



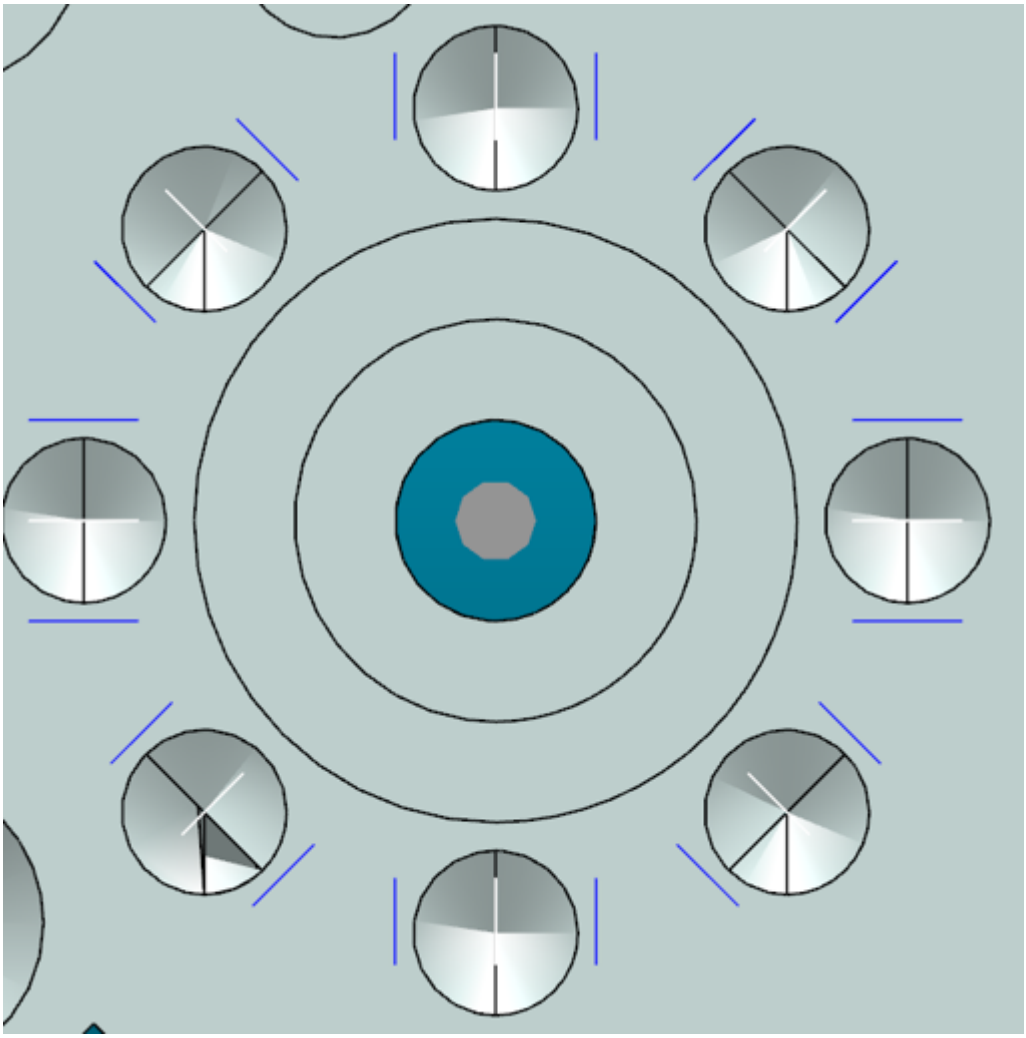
To satisfy this requirement, you would create two position commands, each with a planar tolerance zone. For the 0.04 position, you would click the **Zone Orientation** button and select **Radial Arc** from the **Axis** drop-down list. For the 0.2 position, you would click the **Zone Orientation** button and select **Right Angle to Radial** from the **Axis** drop-down list.

Setting `CADGRAPH=ON` may help you to visualize the tolerance zone shape and orientation more easily in the PC-DMIS graphic window.

```
FCFLOC1     =GEOMETRIC_TOLERANCE/STANDARD=ASME Y14.5 - 2009,SHOWEXPANDED=YES,
             DESCRIPTION=ON,,
             FEATURE_MATH=DEFAULT,DATUM_MATH=DEFAULT,DISPLAY_COORDS=DRF,
             UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
             SEGMENT_1,POSITION,RADIALARC,0.04,__,__,<len>,__,A,E,__,__,B,__,
             SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=ON,REPORTGRAPH=ON,MULT=50,
               MEASURED:
                 CYL1:0.000,
                 CYL2:0.000,
                 CYL3:0.000,
                 CYL4:0.000,
                 CYL5:0.000,
                 CYL6:0.000,
                 CYL7:0.000,
                 CYL8:0.000,
             ADD
             DATUMS/REPORTDATUMSIZE=ON,
               E(DATUM_E):NOM=10,+Tol=0.2,-Tol=-0.2,
             FEATURES/CYL1,CYL2,CYL3,CYL4,CYL5,CYL6,CYL7,CYL8,,
```



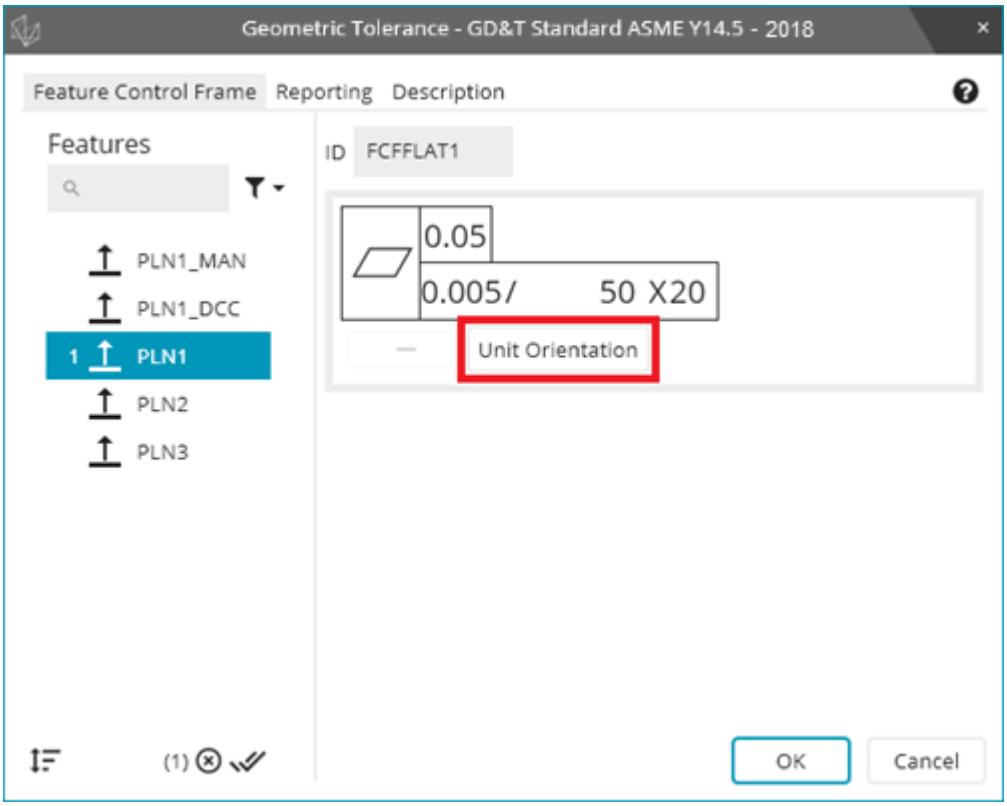
```
FCFLOC2     =GEOMETRIC_TOLERANCE/STANDARD=ASME Y14.5 - 2009,SHOWEXPANDED=YES,
             DESCRIPTION=OFF,,
             FEATURE_MATH=DEFAULT,DATUM_MATH=DEFAULT,DISPLAY_COORDS=DRF,
             UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
             SEGMENT_1,POSITION,PERPTORADIAL,0.2,__,__,<len>,__,A,E,__,__,B,__
             SHOW_OPT_SHIFT=OFF,TEXT=OFF,CADGRAPH=ON,REPORTGRAPH=ON,MULT=50,
               MEASURED:
                 CYL1:0.000,
                 CYL2:0.000,
                 CYL3:0.000,
                 CYL4:0.000,
                 CYL5:0.000,
                 CYL6:0.000,
                 CYL7:0.000,
                 CYL8:0.000,
             ADD
             DATUMS/REPORTDATUMSIZE=ON,
               E(DATUM_E):NOM=10,+Tol=0.2,-Tol=-0.2,
             FEATURES/CYL1,CYL2,CYL3,CYL4,CYL5,CYL6,CYL7,CYL8,,
```



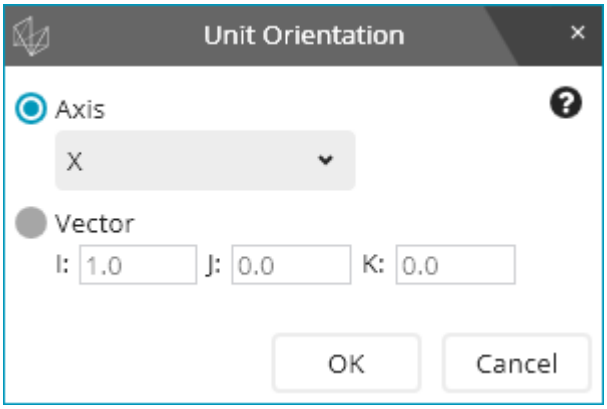
Unit Orientation

Per-unit flatness tolerances have square or rectangular units, as described in the "Flatness" topic. The geometric tolerance command needs to know how to orient the unit on the planar surface. You can enable per-unit flatness with the plus sign that adds the lower segment:

The **Unit Orientation** button becomes visible:



If you click the **Unit Orientation** button, the **Unit Orientation** dialog box appears. This dialog box allows you to control the unit orientation:



The **Unit Orientation** dialog box allows you to define the unit's orientation vector. For more information, see "Flatness Per Unit" in the "Flatness" topic.

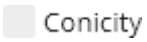
Cone Half Angle

This check box is under the tolerance editing pane, and it looks like this:

For circular runout tolerances on circles, the geometric tolerance command allows you to treat the circle as a cross section of a cone instead of a cross section of a cylinder. To do this, you can select the **Half Cone Angle** check box and then type an angular value for the cone half angle. For more information, especially the meaning of the sign of the cone half angle, see "Circular Runout".

Circularity versus Conicity

The **Conicity** check box is under the tolerance editing pane, and it looks like this:



As discussed in the "Circularity" topic, PC-DMIS can evaluate circularity tolerances on cones as true circularity or as conicity. By default, it is evaluated as true circularity, but you can change it to conicity by selecting this check box.

Lengthwise versus Widthwise Slots

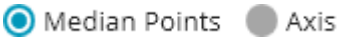
These left and right options appear inside the tolerance editing pane if your considered features are slots:

The left option considers the slot in a widthwise fashion. The size of the slot is its width, and the tolerance zone controls the slot's position in the width direction.

The right option considers the slot in a lengthwise fashion. The size of the slot is its length, and the tolerance zone controls the slot's position in the length direction.

Median Points versus Axis

These options appear inside the tolerance editing pane if you have concentricity or symmetry tolerance with ASME:



For concentricity and symmetry ASME Y14.5 1994 or ASME Y14.5 2009 tolerances, when the considered features have surface data, PC-DMIS can interpret them in terms of the **Median Points** or in terms of the **Axis**. You can choose the desired option to control the interpretation. **Median Points** is the default.

Adding and Removing Segments

Underneath the Feature Control Frame Editor, there are **+** and **-** buttons to add and remove segments, respectively. When it does not make sense to add any more segments, the **+** button is not available.

The buttons look like this:

You can use these buttons to construct composite position tolerances, composite profile tolerances, and per-unit tolerances.

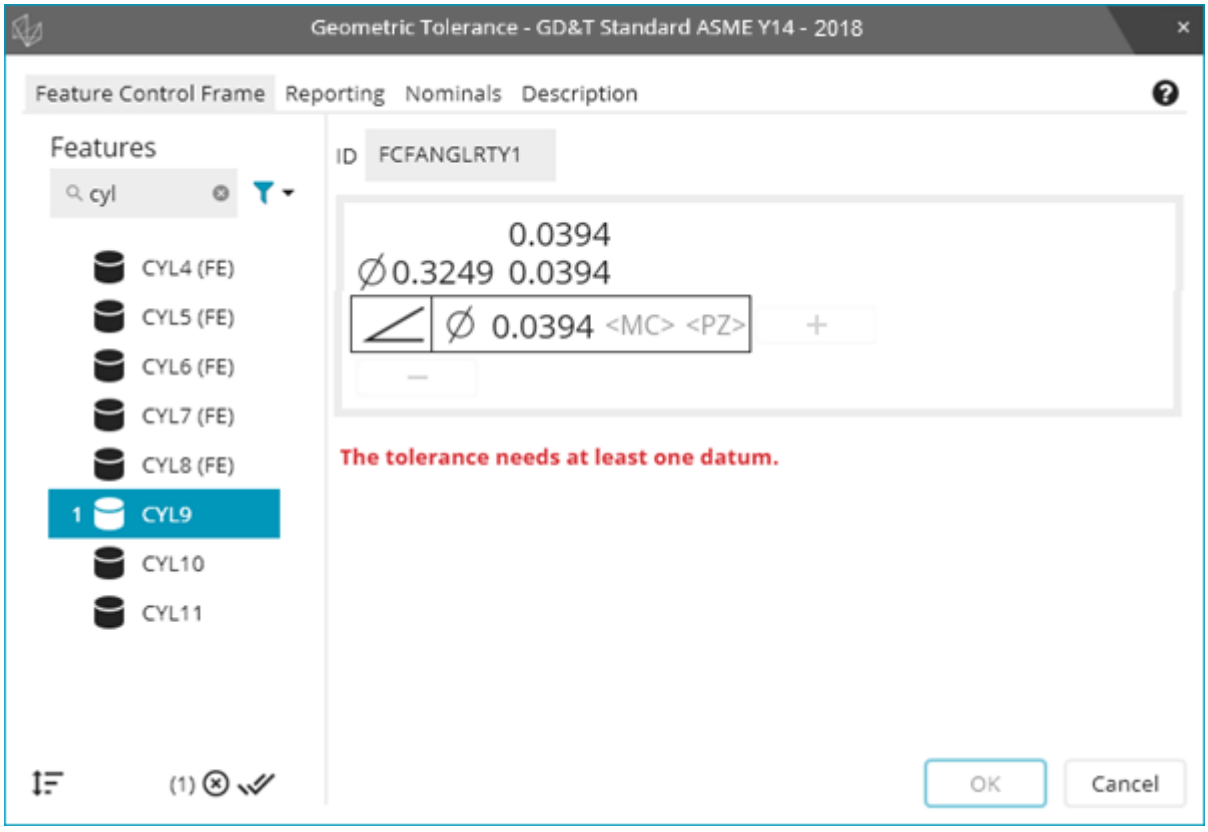
The Information Section

Underneath all of these feature control frame options, PC-DMIS shows error messages, warnings, and other informational messages. For information on how to resolve error or warning messages, see "Troubleshooting Error Messages and Warnings".

Error messages - Error messages show up in **red**.

When there is an error message, you cannot click **Create** or **OK** in the dialog box.

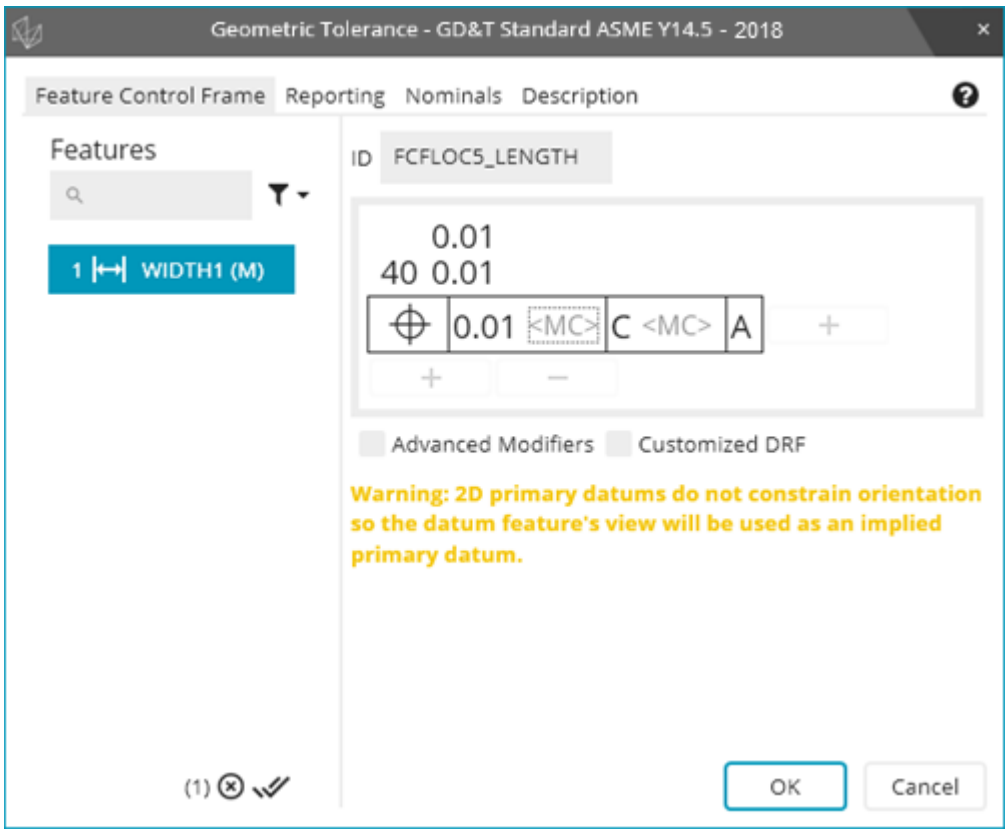
Error messages look like this:



Warning messages - Warning messages show up in **yellow**.

When there is a warning message, you can press **Create** or **OK** in the dialog box.

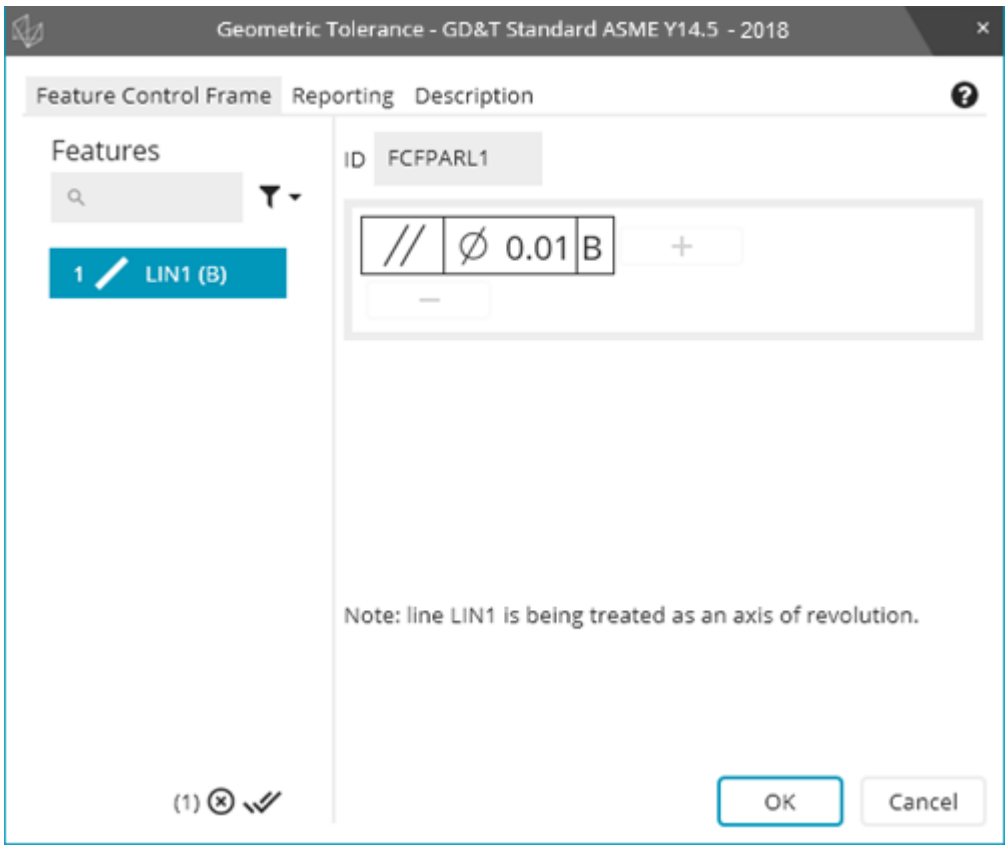
Warning messages look like this:



Interpretation messages - Interpretation messages show up in black.

Interpretation messages appear when you use a constructed line feature as a datum feature or as a considered feature. The message lets you know whether PC-DMIS interprets the feature as a line on a surface (such as a cross section of a planar surface) or as an axis of revolution (such as a surfaceless axis). For more information about what line types are considered a line on a surface, versus what line types are considered an axis of revolution, see "Feature Types With and Without Surface Data".

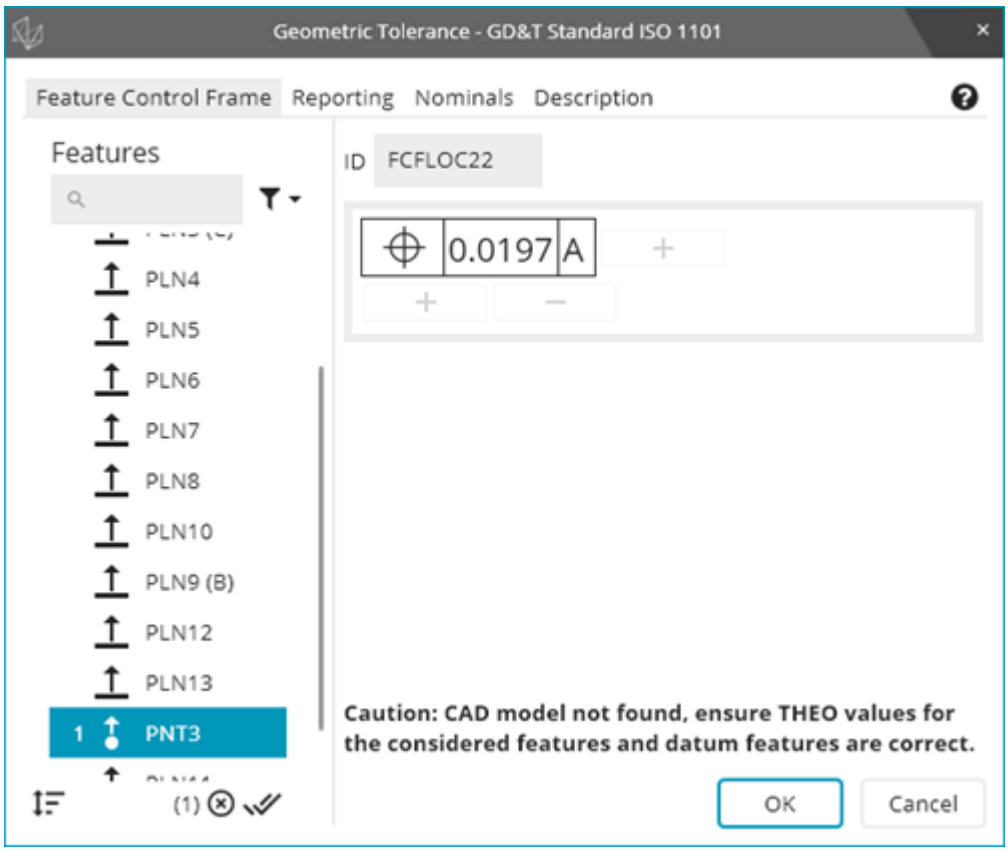
Interpretation messages look like this:



CAD Warning messages - There is an additional type of warning message that appears in black just above the main command buttons on the dialog box.

If you do not have a CAD model, PC-DMIS cannot ensure that your program THEO values are all correct. The **Geometric Tolerance** dialog box shows a warning message that cautions you to ensure your THEO values are all correct. You can still click **Create** or **OK** in the dialog box.

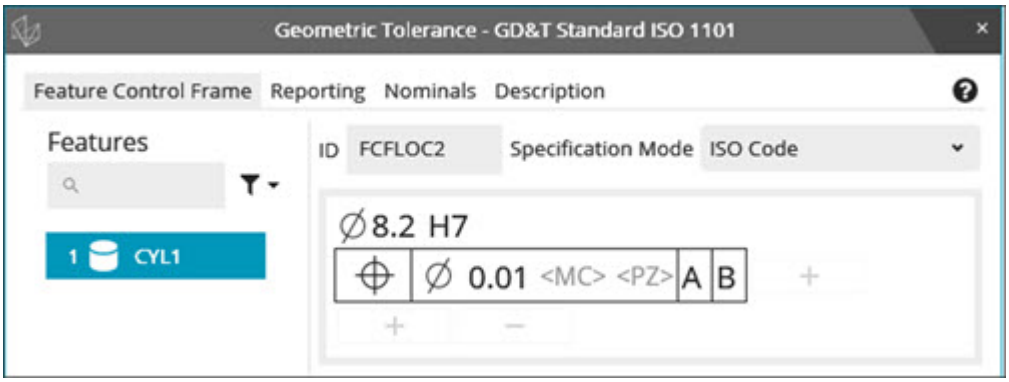
This CAD warning looks like this:



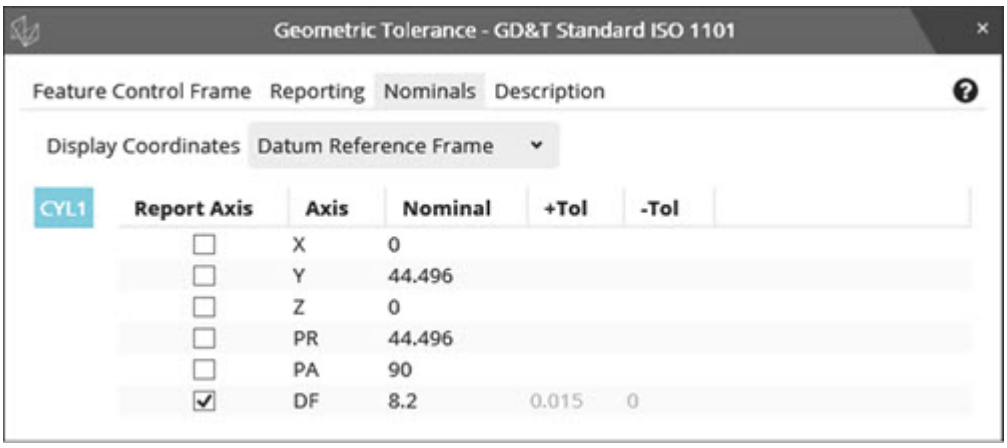
Specification Mode

The **Specification Mode** list controls the **Size Tolerance** editor. For a feature of size, when you select the **Nominals With Deviation** option, tolerances are entered as described in the "The Size Tolerance Editor" section.

When you select the **ISO Code** option, the size tolerance editor displays a default ISO H7 code. Type in the appropriate limit as defined with ISO 286-1. The ISO 286-1 standard defines hundreds of tolerance codes that look somewhat like "E9" and "H7". The ISO 286-1 standard is case-sensitive, for example, holes are uppercase and shafts are lowercase. You can type the standard in either uppercase or lowercase. PC-DMIS determines whether the considered feature is an internal or external feature type and then auto-corrects your entry to the appropriate case if necessary.



In the **Nominals** tab, PC-DMIS shows the appropriate tolerance based on the nominal size.



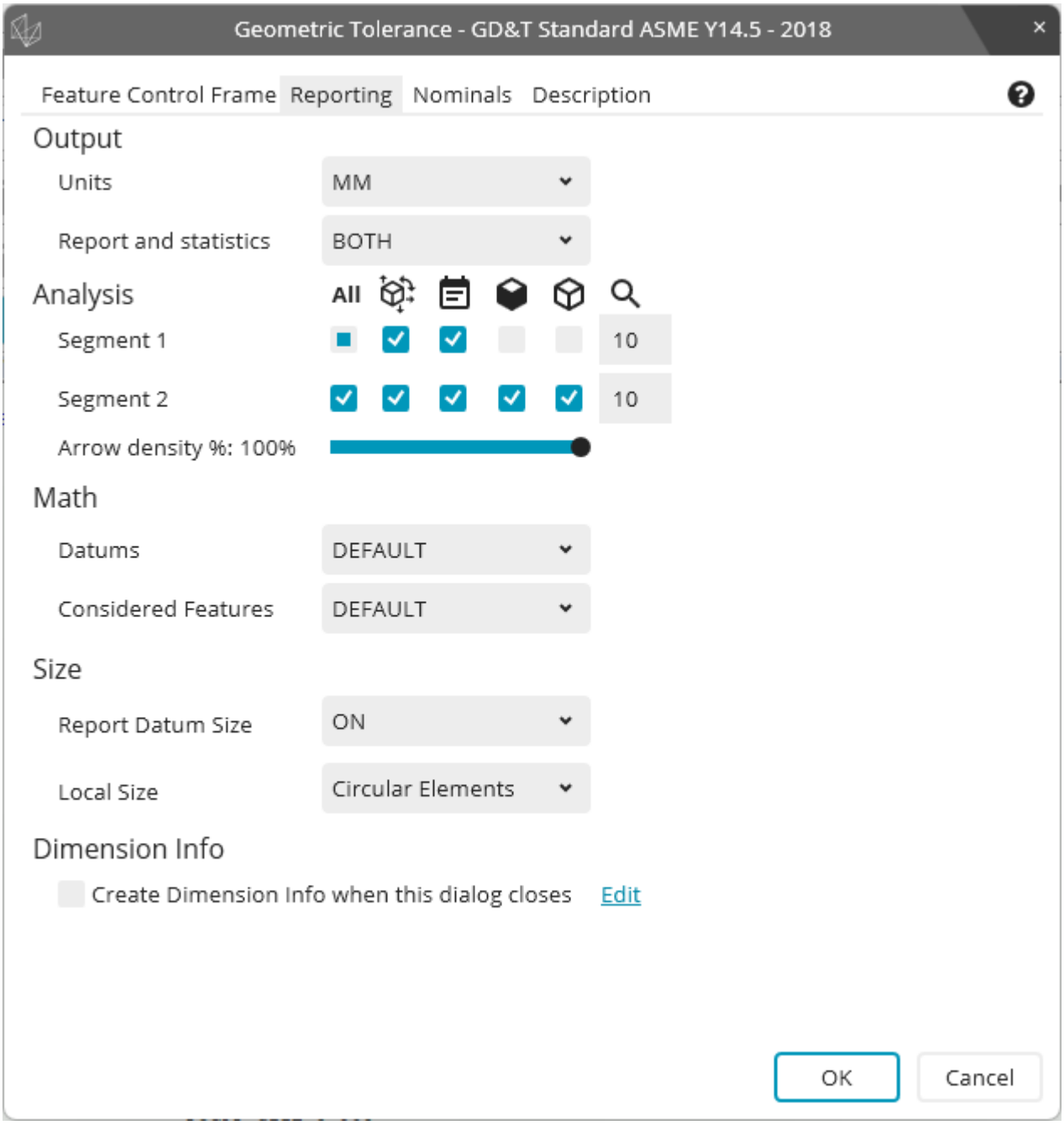
Reporting Tab

In this Topic

Reporting Tab
Introduction
Output
Analysis
Math
Size or Workplane areas
Dimension Info

Introduction

The **Reporting** tab of the **Geometric Tolerance** dialog box has several options to control how PC-DMIS evaluates and reports the calculations. The **Reporting** tab looks like this:



Output

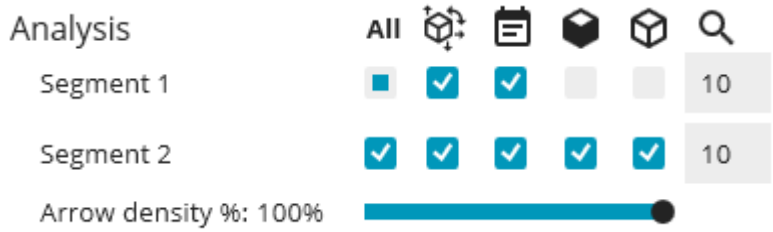
The **Output** area of the **Reporting** tab has these two drop-down lists:

Units - This allows you to report results in either inches or mm, regardless of the dimensions of your measurement routine.

Report and statistics - This allows you to send the results to the **REPORT**, to the **STATS**, to **BOTH**, or to **NONE**.

Analysis

The **Analysis** area of the **Reporting** tab allows you to control the textual and graphical analysis options. There is a row of check boxes for each segment of the geometric tolerance command. For example, with two segments the **Analysis** area looks like this:



The icon above each column of check boxes is a label for which kind of analysis that column of check boxes controls:

All - Selecting or clearing it selects or deselects the entire row of check boxes.

- This shows or hides optimized shift information in the report for position or profile tolerances. You can hover your pointer over the icon to see a tooltip that it is to show optimized shift. For more information, see the "Optimized Shift" section in the "How PC-DMIS Solves and Uses Datums" topic.

- This turns textual analysis on or off. You can hover your pointer over the icon to see a tooltip that it is for report textual analysis.

- This turns CAD graphical analysis on or off. You can hover your pointer over the icon to see a tooltip that it is for CAD graphical analysis.

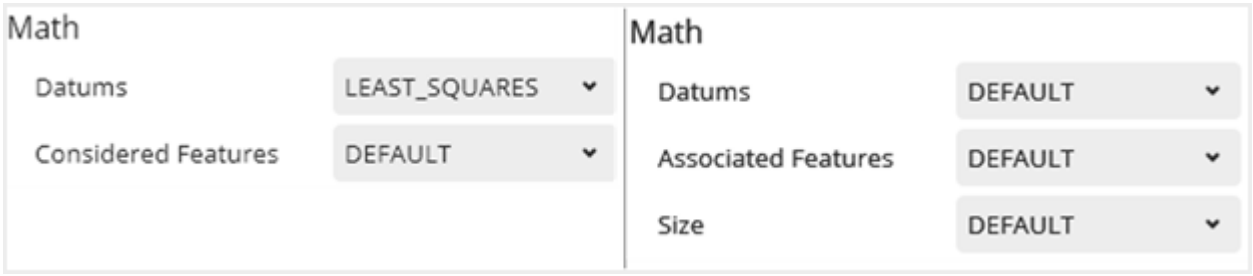
- This turns the report graphical analysis on and off. You can hover your pointer over the icon to see a tooltip that it is for report graphical analysis.

- These boxes show the numeric values of the arrow multipliers for each segment.

Arrow density % slider bar - You can use this slider bar to adjust the arrow density percentage value.

Math

The **Math** area of the **Reporting** tab allows you to control the math options for evaluating the geometric tolerance. The available math type options depend on if you select the ASME or ISO standard.



Math Area showing ASME (left) and ISO (right).

ASME Math Types

The **Considered Features** list controls the math to solve size tolerances and/or create the toleranced feature from the considered feature. It is available when the considered features have surface data and either (a) there is a size tolerance or (b) the toleranced feature is different than the considered feature. For details of what the options mean, see "Deriving the Toleranced Feature" and "Evaluating Size with the Geometric Tolerance Command".

The **Tolerance Zone** list (not pictured) controls the math for optimizing the toleranced feature into the tolerance zone. It appears for form tolerances and for profile tolerances without a datum. For details of the options' meaning, see these topics:

- Circularity
- Cylindricity
- Flatness
- Profile of a Line
- Profile of a Surface
- Straightness

ISO Math Types

The **Associated Features** list controls the math to create the toleranced feature from the considered feature. It is available when the considered features have surface data and either (a) there is a size tolerance or (b) the toleranced feature is different than the considered feature. For details of what the options mean, see the "Deriving the Toleranced Feature" and "Evaluating Size with the Geometric Tolerance Command" topics in the PC-DMIS Core documentation.

The **Size** list controls the math to solve size tolerances. It is available when the considered features have surface data and either (a) there is a size tolerance or (b) the toleranced feature is different than the considered feature. For details of what the options mean, see the "Deriving the Toleranced Feature" and "Evaluating Size with the Geometric Tolerance Command" topics in the PC-DMIS Core documentation.

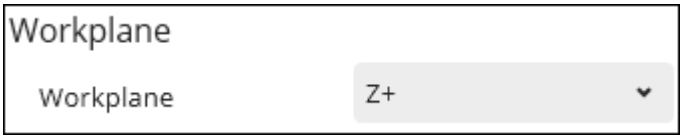
The **Tolerance Zone** list (not pictured) controls the math for optimizing the toleranced feature into the tolerance zone. It appears for all form tolerances and for profile tolerances without datum references.

Size or Workplane areas

The **Size** area of the **Reporting** tab is available for most features. It lets you control whether to report local sizes. It can look like this:

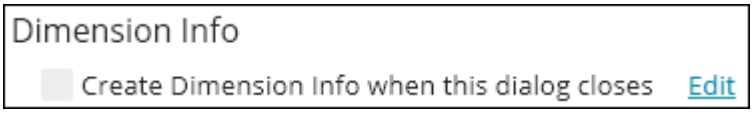
The **Local Size** option is either **ON** or **OFF**. It is available for considered features that have surface data, when there is a size tolerance. For more information, see "Evaluating Size with the Geometric Tolerance Command".

For other features, such as a Line Profile, the **Size** area changes to the **Workplane** area which allows you to select the workplane from the list. It can look like this:



Dimension Info

The **Dimension Info** area of the **Reporting** tab looks like this:



If you mark the check box, PC-DMIS inserts a dimension info command into your measurement routine after the geometric tolerance command. You can click **Edit** to the right of the check box to modify the dimension info command options.

Nominals Tab

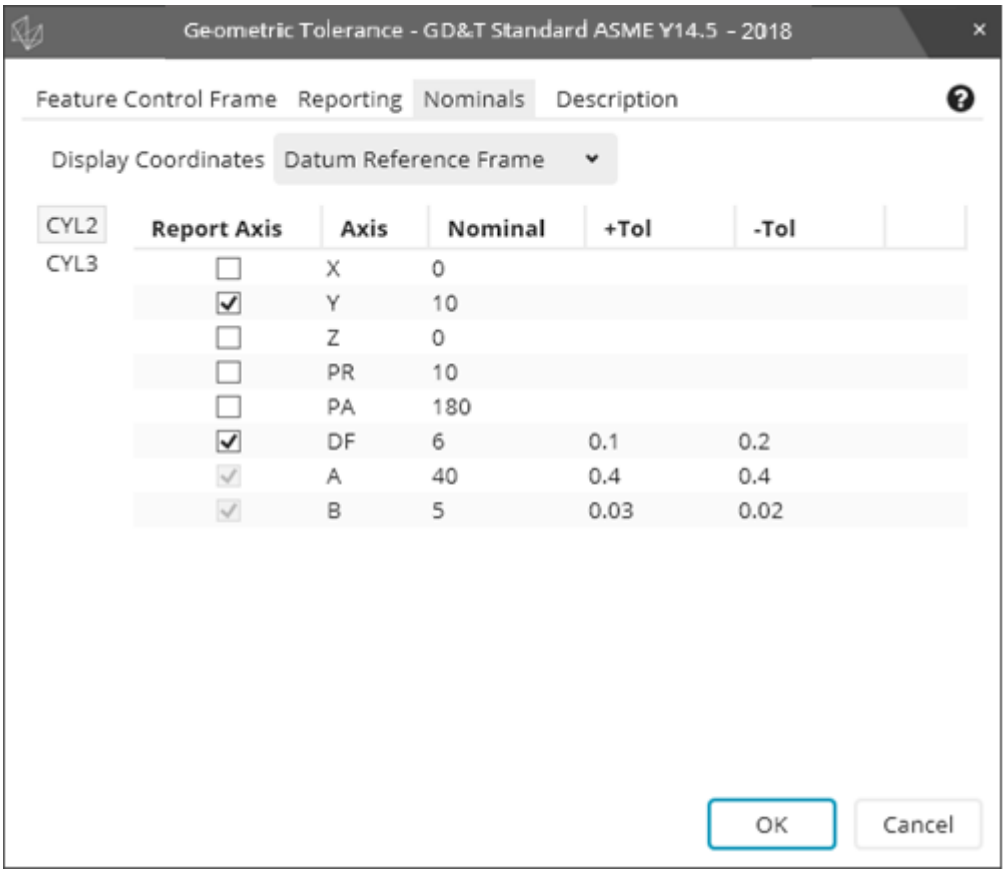
In this Topic

Nominals Tab
Introduction
Display Coordinates
Nominals Table
Select the Considered Feature
Report Axis Column
Axis Column
Nominal Column
+Tol Column
-Tol Column

Introduction

The **Nominals** tab of the **Geometric Tolerance** dialog box is available for position tolerances and for geometric tolerances that have a size tolerance.

It looks like this:

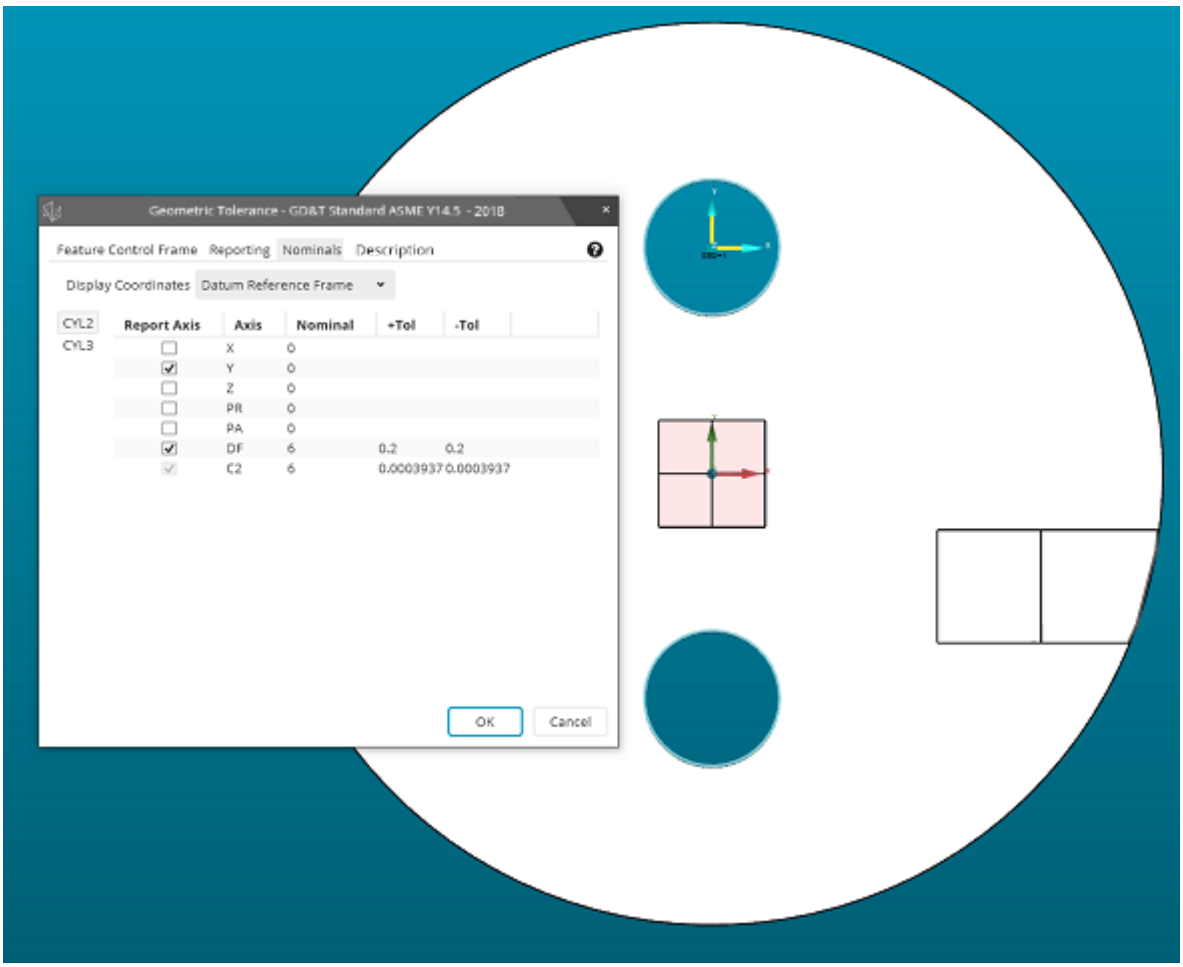


Display Coordinates

This list controls the coordinate system that the nominals are reported in. You can switch between **Datum Reference Frame** and **Current Alignment** to change the nominals that you see.

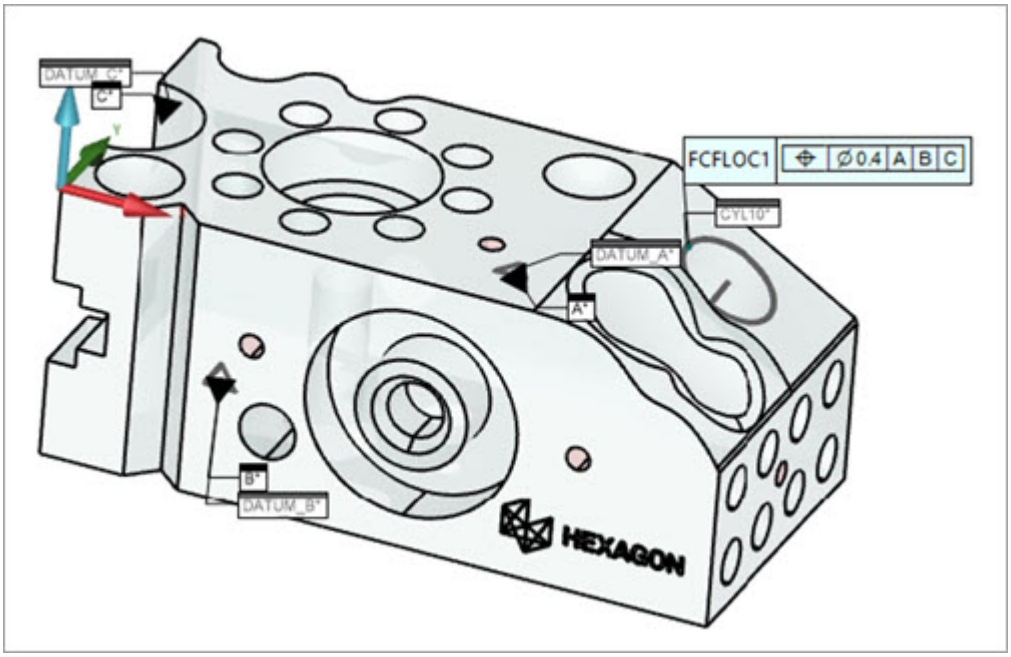
After you click **OK** or **Create**, PC-DMIS changes the coordinate system that the results are reported in.

You can visualize the two coordinate systems in the Graphic Display window, like this:



In the example picture above, the trihedron for the current alignment is at the center of the part. The trihedron for the datum reference frame coordinate system is shown in the upper bore.

The **Current Alignment** option can be useful for cases where the location or orientation of the Datum Reference Frame trihedron does not match the axes depicted on the drawing. It can also serve to provide adjustment information to production. Note the location and orientation of the Datum Reference Frame trihedron for the following position tolerance.

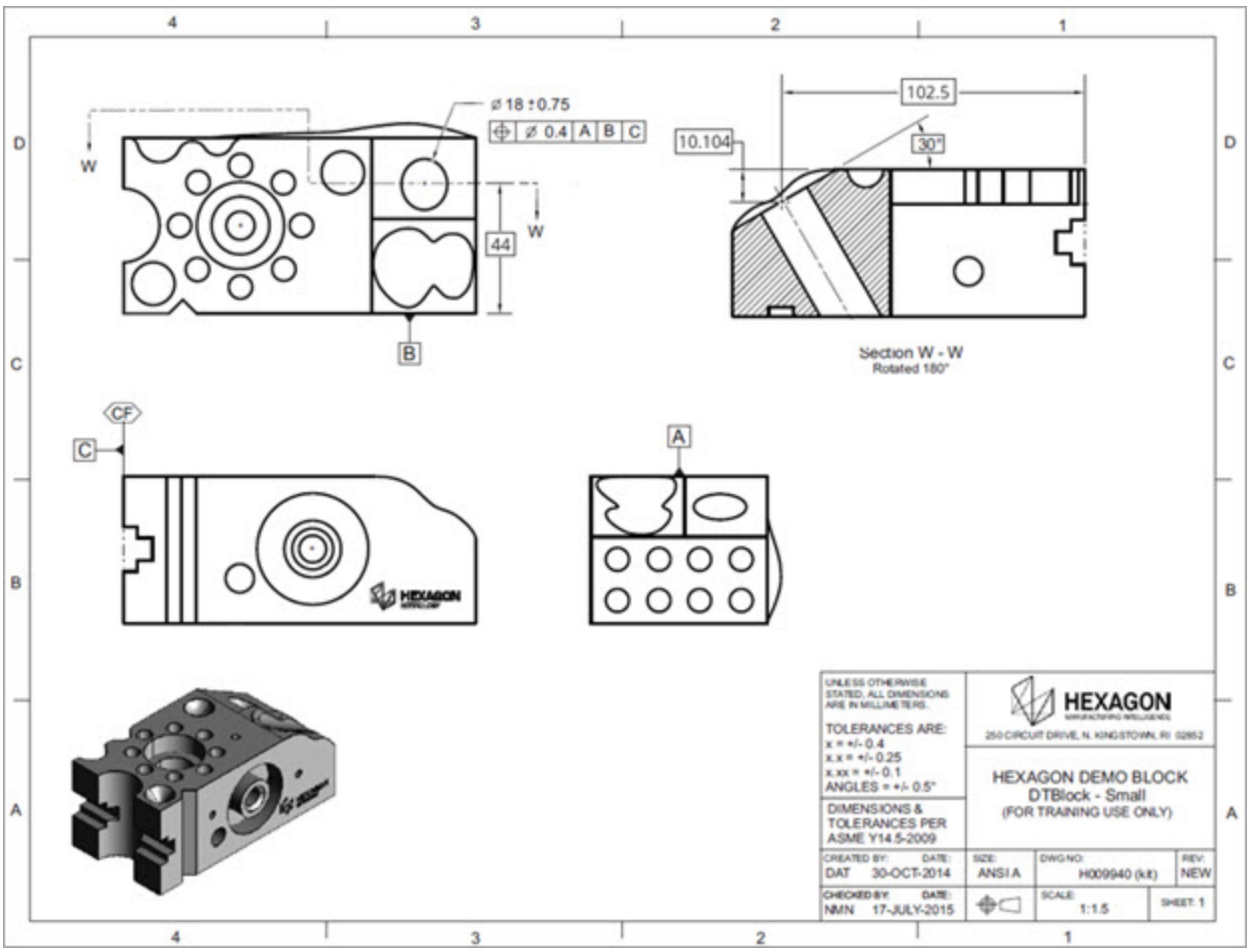


	PART NAME : POSITION_EXAMPLE					April 16, 2024	12:50
	REV NUMBER :		SER NUMBER :			STATS COUNT : 1	
FCFLOC1 Size	MM	Ø 18 +0.75/-0.75			DEFAULT	ASME Y14.5 2018	

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL 10	18.000	0.750	0.750	18.000	0.000	0.000

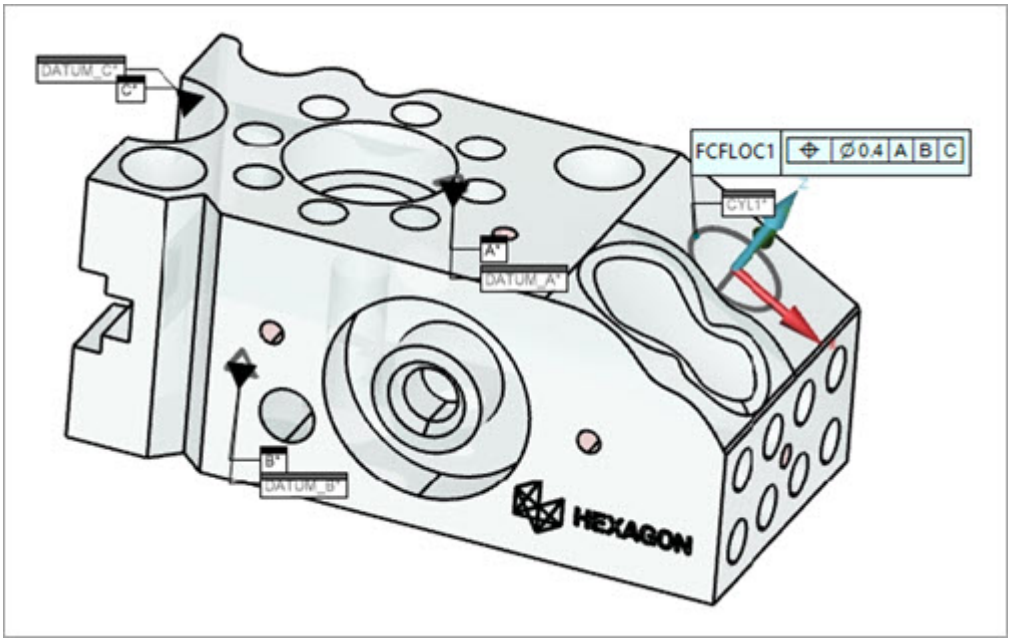
FCFLOC1	MM	Ø0.4 A B C	DEFAULT	ASME Y14.5 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
	X	102.500			102.564	0.064		
	Y	44.000			43.980	-0.020		
CYL 10	Z	-10.104			-10.141	-0.037		
(START PT)	TP	0.000	0.400	0.000	0.154	0.154	0.000	0.000



Although the X, Y, and Z coordinates displayed in the summary table match the basic dimensions on the drawing, it is often more useful to create an alignment that is centered on and orientated to the hole. Once this is done, you should switch the display coordinates from **Datum Reference Frame** to **Current Alignment** as shown below.

```
A1          =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
              ALIGNMENT/TRANS_OFFSET,XAXIS,102.5
              ALIGNMENT/TRANS_OFFSET,YAXIS,44
              ALIGNMENT/TRANS_OFFSET,ZAXIS,-10.104
              ALIGNMENT/ROTATE_OFFSET,30,ABOUT,YPLUS
            ALIGNMENT/END
FCFLOC1     =GEOMETRIC_TOLERANCE/STANDARD=ASME Y14.5-2018,SHOWEXPANDED=YES,
            DESCRIPTION=OFF,,
            FEATURE_MATH=DEFAULT,DATUM_MATH=DEFAULT,DISPLAY_COORDS=CURRENT_ALIGNMENT,
            UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
            SIZE/NOMINAL=18,UPPER TOLERANCE=0.75,LOWER TOLERANCE=0.75,
            REPORT_LOCAL_SIZE=OFF,
              CYL1:
                UAME SIZE:18.000,
            SEGMENT_1,POSITION,DIAMETER,0.4,__,__,<len>,__,A,B,__,C,__,
            TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF,MULT=10,
              MEASURED:
                CYL1:0.154,
            ADD
            FEATURES/CYL1,,
```



	PART NAME : POSITION_EXAMPLE					April 16, 2024	11:38	
	REV NUMBER :		SER NUMBER :			STATS COUNT : 1		
FCFLOC1 Size	MM	Ø 18 +0.75/-0.75			DEFAULT	ASME Y14.5 2018		
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL		
CYL 1	18.000	0.750	0.750	18.000	0.000	0.000		
FCFLOC1	MM	⌖ Ø0.4 A B C			DEFAULT	ASME Y14.5 2018		
Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
	X	-0.000			0.074	0.074		
	Y	0.000			-0.020	-0.020		
CYL 1	Z	0.000			0.000	0.000		
(END PT)	TP	0.000	0.400	0.000	0.154	0.154	0.000	0.000

The X, Y, and Z deviation values are now much more useful to a machinist who can now see how much adjustment is required in each direction to bring the hole position closer to nominal.

Note that in both examples, the MEAS TP value remains the same (0.154 mm). The display coordinates do not affect the datum fitting or geometric tolerance calculation, they only affect the way the data is presented in the report.

The trihedron does not update dynamically while you use the **Geometric Tolerance** dialog box. You need to click **Create** or **OK** to see the updated trihedron.

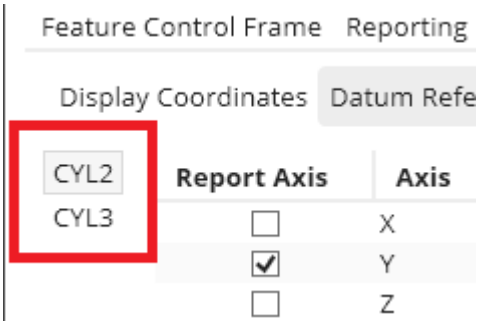
Nominals Table

The table of items allows you to do the following:

- View the nominal locations of the considered features in the chosen display coordinates.
- Control the size tolerances of the considered features and datum features.
- Control which axes to include in the report.

Select the Considered Feature

You can control which considered feature to view with the tabs on the left. Each feature has a small tab that you can click on, like this:



Each feature's tab lets you view the nominal location of that considered feature.

Report Axis Column

The **Report Axis** column of the nominals table determines whether to include that axis in the report. You can mark the check boxes to show the desired information.

Axis Column

The **Axis** column of the nominals table contains the names of the axes you may want to report. These names stand for the following items:

- **X** - This means the X coordinate.
- **Y** - This means the Y coordinate.
- **Z** - This means the Z coordinate.
- **PR** - This means the polar radius.

- **PA** - This means the polar angle.
- **DF** - This means diameter of the feature (or size).

As you can see in the image below, the datum names are also in the axis column:

Report Axis	Axis	Nominal	+Tol	-Tol	
☐	X	0			
☑	Y	10			
☐	Z	0			
☐	PR	10			
☐	PA	180			
☑	DF	6	0.1	0.2	
☑	A	40	0.4	0.4	
☑	B	5	0.03	0.02	

Only datum features of size appear. The **Report Axis** check box is not available for these rows.

Nominal Column

The **Nominal** column of the nominals table contains the nominal values of the reportable axes, and the nominal sizes of the datum features of size.

+Tol Column

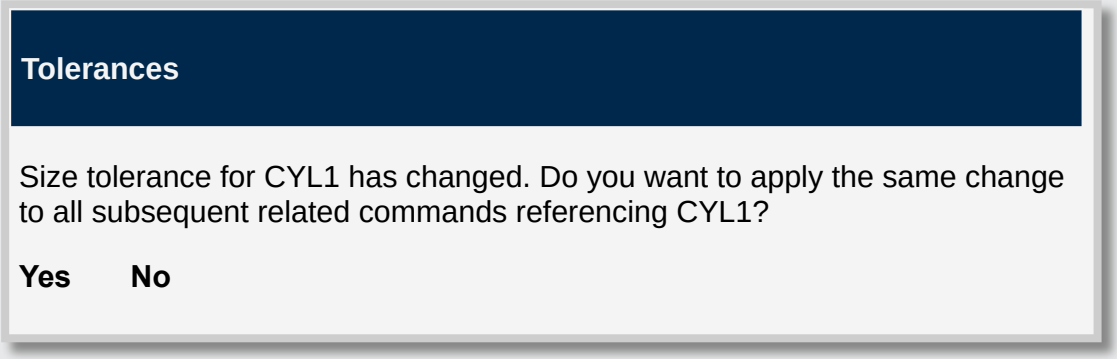
The **+Tol** column of the nominals table contains the plus size tolerances of the considered features and datum features of size. See note below if you make changes to this value.

-Tol Column

The **-Tol** column of the nominals table contains the minus size tolerances of the considered features and datum features of size. See note below if you make changes to this value.

If you edit a feature's upper or lower tolerances from the Edit window or from the **Geometric Tolerance** dialog box (**Feature Control Frame** or **Nominals** tab), and the same feature is used as a datum or as a considered feature, PC-DMIS displays a message asking if you want to apply the same changes to all subsequent commands referencing that feature.

For example:



If you click **Yes**, PC-DMIS updates the size tolerances for any Geometric Tolerance commands below the cursor position that references the same feature, either as a considered feature or datum.

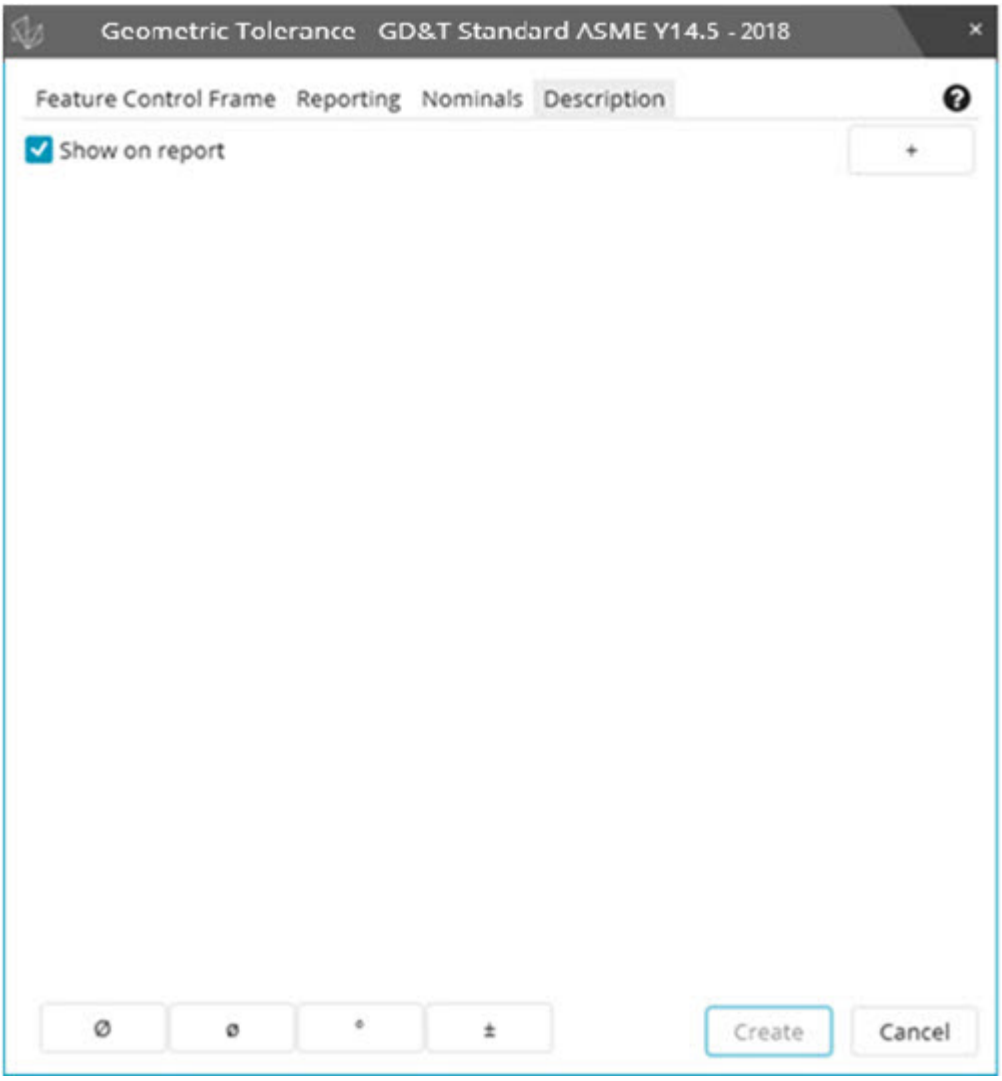
If you click **No**, PC-DMIS updates the edited size tolerance only. PC-DMIS does not update any of the respective size tolerances for any related Geometric Tolerance commands below the cursor position that uses the same edited feature either as a considered feature or datum.

The **Minus Tols Show Negative** check box on the **Dimension** tab of the **Setup Options** dialog box applies the same here as it does in the **Feature Control Frame** tab. For information, see the "Feature Control Frame Tab" topic.

- When it is cleared, the minus tolerances are typically positive.
- When it is selected, the minus tolerances are typically negative.

Description Tab

If you have experience creating comments in XactMeasure, the functionality of the **Description** tab on the **Geometric Tolerance** dialog box may look familiar, but it is more useful than its predecessor. You no longer need to describe features or dimensions in their ID names. This can make them extremely long and cumbersome. The **Description** tab provides a way for you to link dimensions to callouts on prints without the need for lengthy, descriptive features or dimension names.



This tab enables you to create pure text fields that you can use to link dimensions with prints. You can use any symbol or characters since these fields are strictly for text. The symbol buttons at the bottom of the dialog box allow you to insert commonly used symbols.

You must resolve any errors on the **Feature Control Frame** tab before PC-DMIS enables the **Create** button on this dialog box.

To create a **Description** text:

1. Click the Add (**+**) button in the upper-right corner of the dialog box to create and open a new description text field.
2. Enter your text into the field. Use the symbol buttons as needed to insert specific symbols.
3. Press Enter or click the Add (**+**) button again to create another description text field. To delete a field, select it and click the Remove (**-**) button.

4. Select the **Show in report** check box to include the description text in the report.
5. Click the **Create** button to create the description field.

Geometric Tolerance Types

There are fourteen types of geometric tolerances, often grouped into these five categories: Form, Orientation, Location, Profile, and Runout.

The tolerance topics below go through each tolerance type. Those topics provide details about what each type means, its allowed modifiers, and allowed command options.

Form

Form tolerances are the simplest because they do not reference datums:

- Circularity
- Cylindricity
- Flatness
- Straightness

Orientation

- Angularity
- Parallelism
- Perpendicularity

Location

- Concentricity
- Position
- Symmetry

Profile

- Profile of a Line
- Profile of a Surface

Runout

- Circular Runout
- Total Runout

You can find further details in these topics:

- Feature Types With and Without Surface Data

- How PC-DMIS Solves and Uses Datums
- Deriving the Toleranced Feature

Actual and Measured Values

For each geometric tolerance type, we distinguish between “actual values” and “measured values.”

> An *actual value* deals with the specification, and it is defined by a specification standard like ASME Y14.5.1 or ISO 1101. An actual value uses all the points of the surface, with no measurement uncertainty. This value tells us whether the surface conforms to its specification or not. It gives us some idea how close a conforming surface is to non-conformance.
>
> A *measured value* is a measured approximation to an actual value. The measured value uses some subset of the points of the surface. Each measured point of the surface includes measurement uncertainty. The algorithms we use to derive a measured value may or may not be similar to the mathematical definition of the actual value. This is because sometimes the best measured approximation to the actual value uses very different mathematics than the actual value uses.

For more information, refer to "Specification Versus Verification".

Circularity

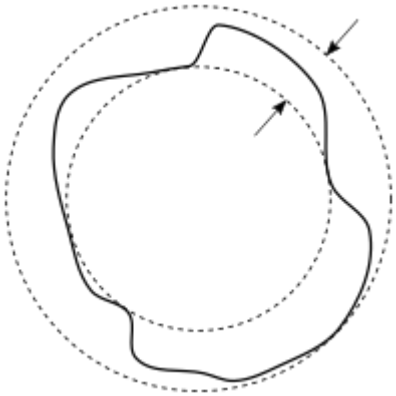
In this Topic

Circularity
- Introduction
- Allowed Feature Types
- Circular Features
- Cylindrical Features
- Conical Features
- Spherical Features
- Allowed Modifiers
- Exposed Options
- Report

Introduction

A circularity specification controls how much the feature's cross sections can deviate from a perfect circle. In other words, circularity evaluates how circular the feature is. Circularity is defined from a feature's cross sections.

> Actual Value:
> This is the minimum distance between two concentric circles that contain the entire cross section between them:



The actual value for circularity of a whole feature is the worst actual value of all possible cross sections.

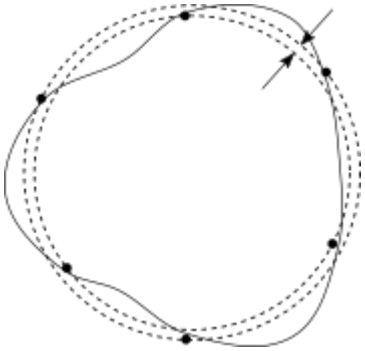
Allowed Feature Types

You can use circular, cylindrical, conical, or spherical features that have surface data. For details on the circles, cylinders, cones, and spheres that have surface data, refer to "Feature Types With and Without Surface Data".

Circular Features

Circular features are interpreted as a single cross section.

Measured Value:
This is the distance between two concentric circles that contain all the measured points between them. A best-fitting routine defines the center-point of the circles. Depending on the measurement uncertainty, how many points you measured, and where you took the points, this can be larger or smaller than the actual value. Here's an illustrated case where too few points were measured, so the measured value is smaller than the actual value:



Cylindrical Features

Circularity tolerances on cylindrical features divide up the data into cross sections. The tolerances evaluate circularity on each cross section. The measured value of the entire feature is the measured value of the worst cross section. To maximize your chances of finding the worst actual cross section, we recommend that you measure

the cylinder with many cross sections. If PC-DMIS is unable to extract cross sections from the measured data, PC-DMIS gives you an error.

Conical Features

Circularity tolerances on conical features divide up the data into cross sections. The tolerances evaluate circularity on each cross section. The measured value of the entire feature is the measured value of the worst cross section. To maximize your chances of finding the worst actual cross section, we recommend that you measure the cone with many cross sections. If PC-DMIS is unable to extract cross sections from the measured data, PC-DMIS gives you an error.

An alternate way to evaluate the form of a conical feature is using conicity. Conicity does not divide the data into cross sections. Instead, the measured value is the distance between two coaxial same-angle cones. These cones contain the measured points between them. A best-fitting routine defines the axis and the cone angle. Conicity includes both circularity and straightness errors. It does not require the measured data to be measured in cross sections.

Spherical Features

Circularity of a sphere is equivalent to sphericity (see ASME Y14.5.1 and ISO 1101). This sphericity tolerance operates on all the data at once. The measured value is the distance between two concentric spheres that contain the measured points between them. A best-fitting routine defines the center-point of the spheres. It does not require the measured data to be measured in cross sections.

Allowed Modifiers

For the ASME standard, this geometric tolerance does not allow modifiers.

For the ISO standard, C or G modifiers are allowed. For details, see the "ISO Reference Feature Association Modifiers (RFAM)" section in the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Exposed Options

For the ASME standard, the tolerance zone math type controls the best fitting routine.

For the ISO standard, the selected reference feature association modifier or tolerance zone math type controls the best fitting.

DEFAULT - This does a minimum-zone best fit (also called min-max). This best fit finds the smallest measured value given the data and the measured-value definition. It is mathematically very similar to the specification, because if you measure the points and cross sections densely and with high accuracy, the measured value closely approximates the actual value. For the ISO standard, if you select the C modifier, it is equivalent to the **DEFAULT** math type.

LSQ - This does a least-squares best fit. It minimizes the sum of the squares of the deviations to a least-squares shape. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly. For the ISO standard, if you select the G modifier, it is equivalent to the **LSQ** math type.

The **CIRCULARITY** and **CONICITY** toggle controls the behavior of circularity of a cone.

CIRCULARITY - Evaluates the circularity of each cross section. In the dialog box, you can clear the **Conicity** check box to use this.

CONICITY - Evaluates the conicity of the entire feature. The conicity interpretation is more conservative than the **CIRCULARITY** option. In the dialog box, you can mark the **Conicity** check box to use this.

Report

Here is an example report for a circularity tolerance:

FCFCIRTY1	MM	○ 0.01	DEFAULT	ASME Y14.5 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
CIR4	0.000000	0.010000		0.002759	0.002759	0.000000	

Cylindricity

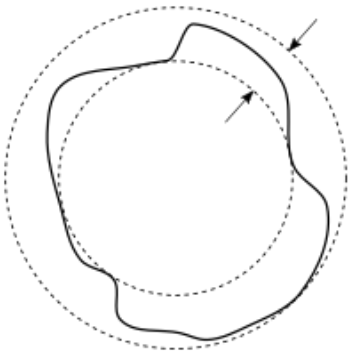
In this Topic

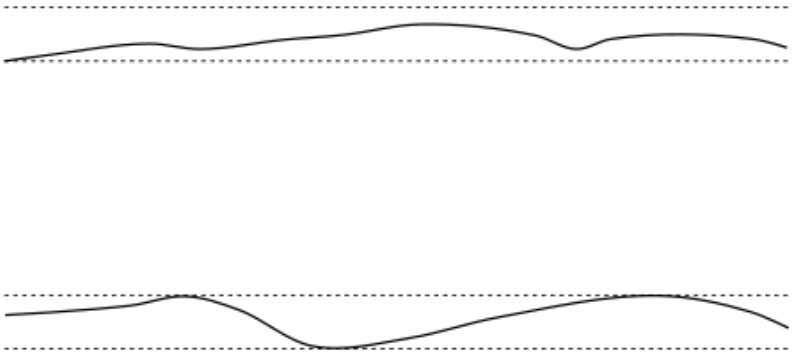
- Cylindricity
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Exposed Options
  - Report

Introduction

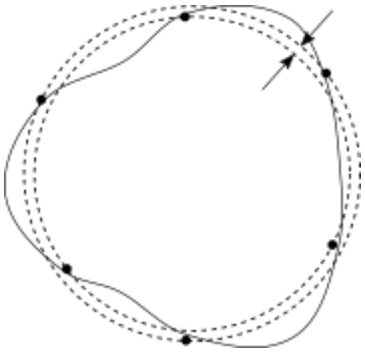
A cylindricity specification controls how much a feature can deviate from a perfect cylinder. In other words, cylindricity evaluates how cylindrical the feature is.

Actual Value:
This is the minimum distance between two concentric cylinders that contain the entire surface between them:





Measured Value:
This is the distance between two concentric cylinders that contain all the measured points between them. A best-fitting routine defines the axis of the two cylinders. Depending on the measurement uncertainty, how many points you measured, and where you took the points, this can be larger or smaller than the actual value. Here's an illustrated case where too few points were measured, so the measured value is smaller than the actual value:



Allowed Feature Types

You can use cylindrical features with surface data. For details on cylinders that have surface data, refer to "Feature Types With and Without Surface Data".

Allowed Modifiers

For the ASME standard, this geometric tolerance does not allow modifiers.

For the ISO standard, C or G modifiers are allowed. For details, see the "ISO Reference Feature Association Modifiers (RFAM)" section in the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Exposed Options

The tolerance zone math type controls how to compute the axis of the two concentric cylinders:

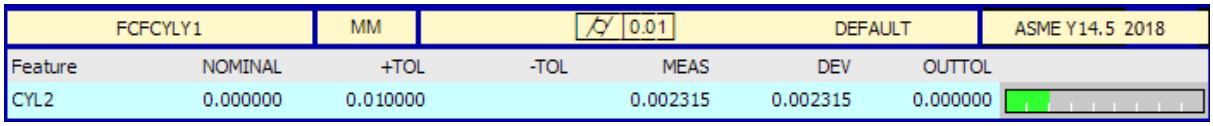
DEFAULT - This does a minimum-zone best fit (also called min-max). This best-fit finds the axis of the two concentric cylinders that contain the points between them that has minimum distance between them. Thus, this option produces the smallest measured value to evaluate cylindricity. It is also mathematically very similar to the

specification, because if you measure points densely and with high accuracy, the measured value closely approximates the actual value. For the ISO standard, if you select the C modifier, it is equivalent to the **DEFAULT** math option.

LSQ - This does a least-squares best fit. It minimizes the sum of the squares of the deviations to a least-squares cylinder. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly. For the ISO standard, if you select the G modifier, it is equivalent to the **LSQ** math option.

Report

Here is an example report for a cylindricity tolerance:



FCFCYLY1	MM	⌭ 0.01		DEFAULT		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	0.000000	0.010000		0.002315	0.002315	0.000000

Flatness

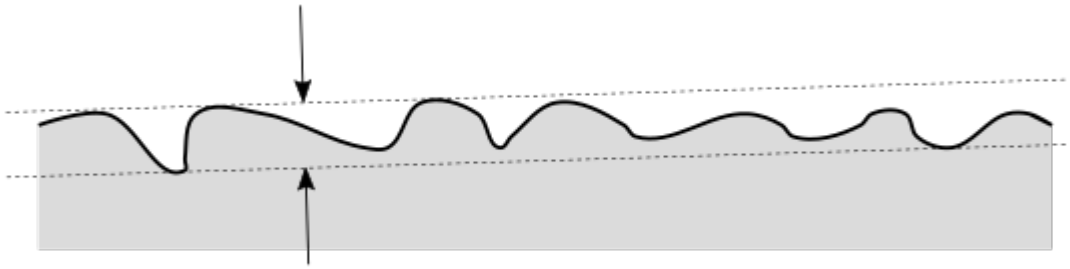
In this Topic

- Flatness
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Report
- Flatness Per Unit-Tolerances
  - Comparison to Past Practice 1
  - Comparison to Past Practice 2
  - Report

Introduction

A flatness specification controls how much the feature can deviate from being perfectly flat. In other words, flatness evaluates how flat the feature is. PC-DMIS only supports flatness specifications on planes.

Actual Value:
This is the minimum distance between two parallel planes that contain the entire surface between them.



Measured Value:
This is the distance between two parallel planes that contain all the measured points between them. A best-fitting routine determines the surface normal of the two planes. Depending on the measurement uncertainty, how many points you measured, and where you took the points, this can be larger or smaller than the actual value. Here's an illustrated case where too few points were measured, so the measured value is smaller than the actual value:

Allowed Feature Types

You can use planar features that have surface data. For details on planes that have surface data, refer to "Feature Types With and Without Surface Data".

Allowed Modifiers

For the ASME standard, this geometric tolerance does not allow modifiers.

For the ISO standard, C or G modifiers are allowed. For details, see the "ISO Reference Feature Association Modifiers (RFAM)" section in the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Exposed Options

For the ASME standard, the tolerance zone math type controls the best fitting routine.

For the ISO standard, the selected reference feature association modifier or tolerance zone math type controls the best fitting.

DEFAULT - This computes a minimum-zone best fit plane (also called min-max). It finds the smallest measured value given the surface data. It is mathematically very similar to the specification, because if the points were measured densely and with high accuracy, the measured value will closely approximate the actual value. For the ISO standard, if you select the C modifier, it is equivalent to the **DEFAULT** math type.

LSQ - This computes a least-squares best fit plane, minimizing the sum of the squares of the deviations to the best fit plane. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general,

this option computes more quickly. For the ISO standard, if you select the G modifier, it is equivalent to the **LSQ** math type.

Report

Here is an example report for a flatness tolerance:

FCFFLAT1	MM	⏥ 0.01		DEFAULT		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
PLN1	0.000000	0.010000		0.000221	0.000221	0.000000

Flatness Per Unit-Tolerances

If you select the **per unit** check box, flatness has two segments. The first (upper) segment is the overall flatness as described above. The lower segment is the per-unit flatness, which defines a unit's size, shape, and orientation. The per-unit tolerances control how flat every possible unit of the toleranced feature is.

You are responsible to do the following:

- Select a square or rectangular unit.
- Choose the size or sizes of each unit.
- Control the orientation of the unit.

Controlling the Unit Orientation

The unit orientation vector controls how the unit is oriented within the planar surface. It is always normalized and always perpendicular to the nominal surface normal of the plane. You can edit the vector with the **Unit Orientation** button in the **Feature Control Frame** tab of the **Geometric Tolerance** dialog box. For more information, see "Unit Orientation" under "Feature Control Frame Tab". With a rectangular unit, the orientation vector represents the direction of the first dimension of the unit area. For example, if the unit area is 5x3, then the unit orientation vector corresponds to the 5. With a square unit, the vector represents the direction of one of the sides of the square.

Actual Value:
Conceptually, the entire toleranced feature is divided into an infinite number of overlapping units. Each unit has the defined unit size, shape, and orientation. Each unit has its own flatness actual value. The flatness actual value for the entire feature is the actual value of the worst unit.

Measured Value:
There are a very large number of overlapping units containing subsets of the measured points. For any given unit, the measured value is the minimum distance between two parallel planes. These planes contain the unit's subset of measured

points between them. This the same as the **DEFAULT** tolerance zone math type. The least squares tolerance zone math type is not available for per-unit tolerances.

The measured value for the entire feature is the measured value of the worst unit.

The algorithm that the geometric tolerance command uses does not check every possible unit. Instead, it does an intelligent search for the worst unit. It always finds the worst unit. It can do this with much less computation time than if it checked every possible unit.

Comparison to Past Practice 1

In PC-DMIS 2020 R2 and later, you can control the unit orientation. The unit orientation vectors are in part coordinates. In earlier versions, with XactMeasure per-unit flatness, you could not control the unit orientation. Also, the units were aligned with the machine coordinate system rather than the part coordinate system.

Comparison to Past Practice 2

In PC-DMIS 2020 R2 and later, the per-unit flatness algorithm is conservative. This means the algorithm always finds the worst unit. In earlier versions, XactMeasure per-unit flatness evaluated a large number of units, but it did not always find the worst unit.

Report

Here is an example report for a flatness per-unit tolerance. The upper label is for the overall flatness, and the lower label is for the per-unit flatness.

FCFFLAT1	MM	⏥ 0.01			DEFAULT	ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
PLN1	0.000000	0.010000		0.000221	0.000221	0.000000
FCFFLAT1	MM	⏥ 0.01/□10			DEFAULT	ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
PLN1	0.000000	0.010000		0.000126	0.000126	0.000000

Straightness

In this Topic

- Straightness
  - Introduction
- Straightness of an Axis
  - Allowed Feature Types
  - Cylindrical Features
  - Conical Features
  - Line Features
  - Allowed Modifiers
  - Exposed Options
  - Report
- Straightness of a Surface

Allowed Feature Types
Allowed Modifiers
Exposed Options
Report
Straightness Per-Unit Tolerances
Exposed Options
Report

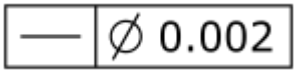
Introduction

A straightness specification controls how much the feature can deviate from being perfectly straight. In other words, straightness evaluates how straight the feature is.

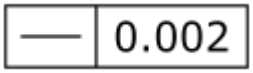
Straightness falls into two broad types:

- Straightness of an axis
- Straightness of a surface

An axis tolerance has a diametric zone symbol before the tolerance value:



A surface tolerance does not have the diametric zone symbol:



Straightness of an Axis

Straightness of an axis operates on a derived median line (or extracted median line in ISO 1101 language). This line represents the straightness form error of a cylinder's or cone's axis.

Actual Value:
This is the diameter of the smallest cylinder that contains the derived median line.

Allowed Feature Types

You can use these features:

- Cylindrical or conical features that have surface data. For details on the cylinders and cones that have surface data, refer to "Feature Types With and Without Surface Data".
- 3D constructed BF lines where the input points are the centers of circles

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Cylindrical Features

On cylindrical features, straightness tolerances of an axis divide up the surface data into cross sections. It then computes the center of each cross section. To maximize your chances of finding the worst cross section, we recommend that you measure the cylinder with many cross sections.

Measured Value:
This is the diameter of a cylinder that contains all the cross-sectional centers. A best-fitting routine determines the axis of the cylinder. If PC-DMIS is unable to extract cross sections from the measured data, PC-DMIS gives you an error.

Conical Features

On conical features, straightness tolerances of an axis divide up the surface data into cross sections. It then computes the center of each cross section. To maximize your chances of finding the worst cross section, we recommend that you measure the cone with many cross sections.

Measured Value:
This is the diameter of a cylinder that contains all the cross-sectional centers. A best-fitting routine determines the axis of the cylinder. If PC-DMIS is unable to extract cross sections from the measured data, PC-DMIS gives you an error.

Line Features

You can use only 3D constructed Best Fit (BF) lines. You cannot use Best Fit Recompensate (BFRE) lines.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

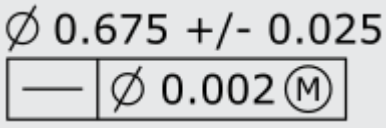
On 3D BF line features, straightness tolerances of an axis assume that the input points represent the centers of circular cross sections. To maximize your chances of finding the worst cross section, we recommend that you measure many cross sections.

Measured Value:
This is the diameter of a cylinder that contains all the input points. A best-fitting routine determines the axis of the cylinder.

Allowed Modifiers

When the feature is a cylinder, straightness tolerances of an axis allow a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, they allow a least material modifier Ⓛ to indicate that the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

This example uses inches. Suppose a cylindrical hole has an axis straightness tolerance 0.002 at MMC:



The size tolerance is 0.675 plus or minus 0.025. This means the range of acceptable sizes is 0.650 to 0.700. The maximum material condition is then 0.650. If the unrelated measured mating envelope size is 0.661, then the bonus tolerance is 0.011, and the total tolerance is 0.013.

Exposed Options

The tolerance zone math type controls the best fitting routine:

DEFAULT - This computes a minimum-zone best fit axis (also called min-max) which finds the smallest measured value given the cross-sectional centers. It is mathematically very similar to the specification, because if the points were measured densely and with high accuracy, the measured value will closely approximate the actual value.

LSQ - This does a least-squares best fit axis. It minimizes the sum of the squares of the deviations to the best fit axis. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly.

Report

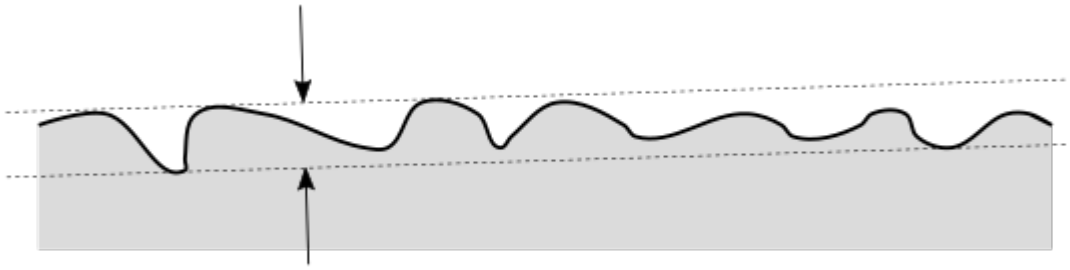
Here is an example report for a straightness of an axis tolerance:

FCFSTRA2	MM	⏤ Ø0.01		DEFAULT		ASME Y14.5
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CON1	0.000000	0.010000		0.000585	0.000585	0.000000

Straightness of a Surface

Straightness of a surface operates on line elements on a surface.

Actual Value:
This is the minimum distance between two parallel lines, containing the entire actual line element between them. The two parallel lines lie in an implicit workplane defined by the view of the drawing. The actual value for an entire surface is the worst actual value of all possible line elements on the surface.



Allowed Feature Types

You must use cone, cylinder, or line features that have surface data. For details on features that have surface data, refer to "Feature Types With and Without Surface Data". To maximize your chances of finding the worst actual cross-section, we recommend that you measure the surface with many lines.

Measured Value:
This is the minimum distance between two parallel lines. The lines contain the surface data between them. A best-fitting routine finds the orientation of the lines. The two parallel lines lie in a temporary (internal) workplane. The temporary workplane's surface normal is perpendicular to the line feature's line vector and to the line feature's surface normal.

Depending on the measurement uncertainty, how many points you measured, how many cross sections you measured, and where you took the points, the measured value can be larger or smaller than the actual value. Here's an illustrated case where too few points were measured, so the measured value is smaller than the actual value:

Allowed Modifiers

For the ASME standard, this geometric tolerance does not allow modifiers.

For the ISO standard, C or G modifiers are allowed. For details, see the "ISO Reference Feature Association Modifiers (RFAM)" section in the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Exposed Options

For the ASME standard, the tolerance zone math type controls the best fitting routine.

For the ISO standard, the selected reference feature association modifier or tolerance zone math type controls the best fitting.

DEFAULT - This computes a minimum-zone best fit line (also called min-max). It finds the smallest measured value given the surface data. It is mathematically very similar to the specification, because if the points and cross sections were measured densely and with high accuracy, the measured value will closely approximate the actual value.

LSQ - This computes a least-squares best fit line. It minimizes the sum of the squares of the deviations to the best fit line. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly.

Report

Here is an example report for a straightness of a surface tolerance:

FCFSTRA6	MM	— 0.01		DEFAULT		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
LIN4	0.000000	0.010000		0.006471	0.006471	0.000000

Straightness Per-Unit Tolerances

If you select the **per unit** check box, straightness has two segments: The first (upper) segment is the overall straightness as described above. The lower segment is the per-unit straightness, which defines a unit length. The per-unit tolerances control how straight every possible unit of the toleranced feature is.

Conceptually, the entire toleranced feature is divided into an infinite number of overlapping unit lengths:

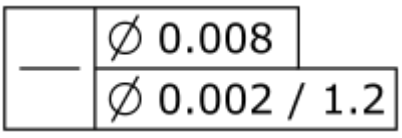
For an axis, the cylinder cross-sectional centers are divided into overlapping unit lengths.

For a surface, the surface cross section is divided up into overlapping unit lengths.

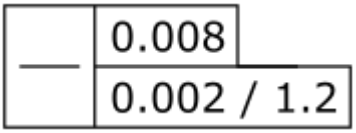
Actual Value:
Each of the infinite units has its own actual value, as defined above. The entire feature's actual value is the actual value of the worst unit.

Measured Value:
There are many overlapping units that contain subsets of the measured points. For any given unit, the measured value is defined the same way as overall straightness, except limited to the subset of the measured points. The measured value for the entire feature is the measured value of the worst unit.

Here is an example of an axis straightness per-unit tolerance. The upper segment is the overall straightness, and the lower segment is the per-unit straightness.



Here is an example of a surface straightness per-unit tolerance. The upper segment is the overall straightness, and the lower segment is the per-unit straightness.



Exposed Options

There are no exposed tolerance zone math options for the **per-unit** segment, it always uses the **DEFAULT** calculations.

Report

Here is an example report for a straightness per-unit tolerance. The upper label is for the overall straightness, and the lower label is for the per-unit straightness.

FCFSTRA1	IN	⏤ 0.008		LSQ		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
LIN1	0.0000	0.0080		0.0007	0.0007	0.0000
FCFSTRA1	IN	⏤ 0.002/1.2		DEFAULT		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
LIN1	0.0000	0.0020		0.0005	0.0005	0.0000

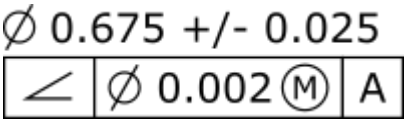
Angularity

In this Topic

- Angularity
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Tolerance Zone Shapes
  - Actual Value and Measured Value
  - Validity Rules
  - Comparison to Past Practice 1
  - Comparison to Past Practice 2
  - Exposed Options
  - Report

Introduction

An angularity specification controls how much the feature can deviate from a specified angle to a datum. Sometimes you can use a secondary datum to control the tolerance zone's orientation.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature".

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes.

Each toleranced feature is optimized independently.

Allowed Feature Types

You can use these feature types:

Cylinders, cones, planes, lines, 3D widths, and 2D widths

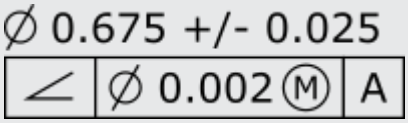
Some feature types have a different toleranced feature than the considered feature's surface data. These include 3D constructed BF lines, cylinders, cones, 3D widths, 2D widths, and planes with a tangent plane modifier. For more information, see "Deriving the Toleranced Feature".

Allowed Modifiers

When the considered feature is a cylinder or a width, this geometric tolerance type allows a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, it allows a least material modifier Ⓛ to indicate the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or the unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

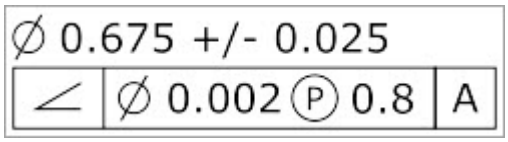
Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Suppose a cylindrical hole has an angularity tolerance 0.002 at MMC, as shown here:



The size tolerance is 0.675 plus or minus 0.025, meaning the range of acceptable sizes is 0.650 to 0.700. The maximum material condition is then 0.650. If the unrelated measured mating envelope size is 0.661, then the bonus tolerance is 0.011, and the total tolerance is 0.013.

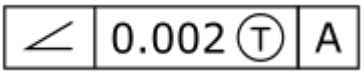
You can use a projected-zone modifier Ⓟ as shown here:



This projects (extrapolates) the measured feature axis as described in "Deriving the Toleranced Feature".

> **!** PC-DMIS only allows you to use projected tolerance zones on Auto Cylinder features. If you try to add a projected-zone modifier for any other feature type, the geometric tolerance command gives an error message stating that the feature type is invalid. This is because the projected tolerance zone needs to start at the nominal end-face of the cylinder. Measured cylinders and constructed cylinders typically do not place the nominal start point at the nominal end-face.

When the considered feature is a plane with surface data, you can use a tangent-plane modifier Ⓣ, as shown here:



This makes the toleranced feature be a perfect-form plane tangent to the actual surface, as described in "Deriving the Toleranced Feature".

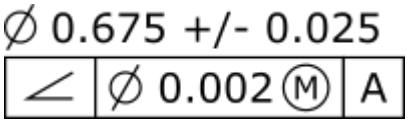
Tolerance Zone Shapes

When the considered feature has an axis, the tolerance zone shape can be diametric (with the tolerance zone shape symbol ∅) or planar (no tolerance zone shape symbol). These are the axial considered features:

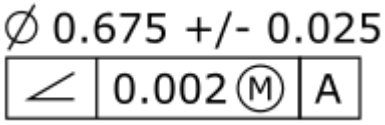
- Cylinder
- Cone
- Surfaceless axis

For information on the feature command types that correspond to these feature types, see "Feature Types With and Without Surface Data".

Here is an example of an angularity on a cylinder with a diametric tolerance zone:



Here is an example of an angularity on a cylinder with a planar tolerance zone:



Planar tolerance zones on axial considered features need a specified tolerance zone orientation. This is because the feature does not have enough information to orient the zone correctly. In these cases, the **Zone Orientation** button in the **Geometric Tolerance** dialog box becomes visible. For information on how to use this button to change the zone orientation, see "Zone Orientation" in the "Feature Control Frame Tab" topic.

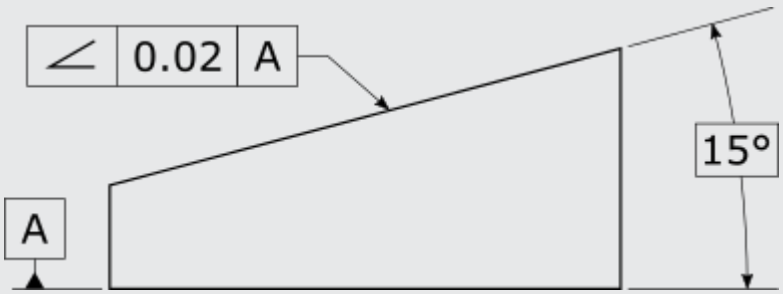
If a tolerance controls the X component of the orientation, the tolerance zone surface normal vector should be X.

Actual Value and Measured Value

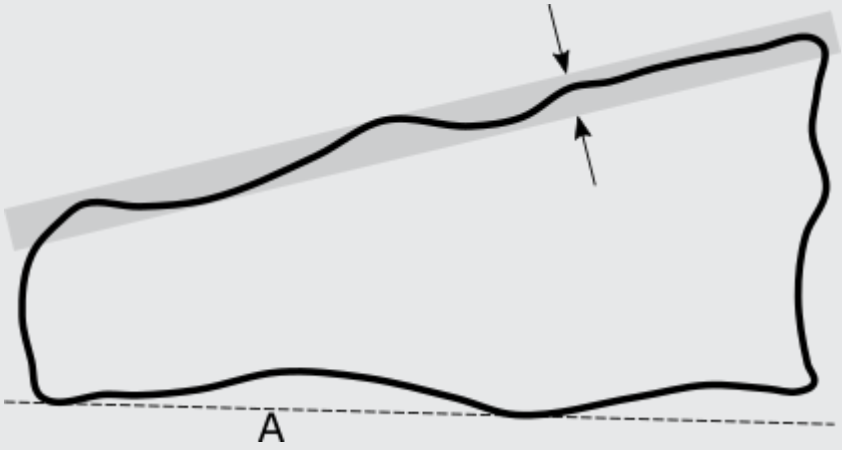
Actual Value:
This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented to the actual datum or datums.

Measured Value:
This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented to the measured datum or datums.

Suppose you had this angularity specification:

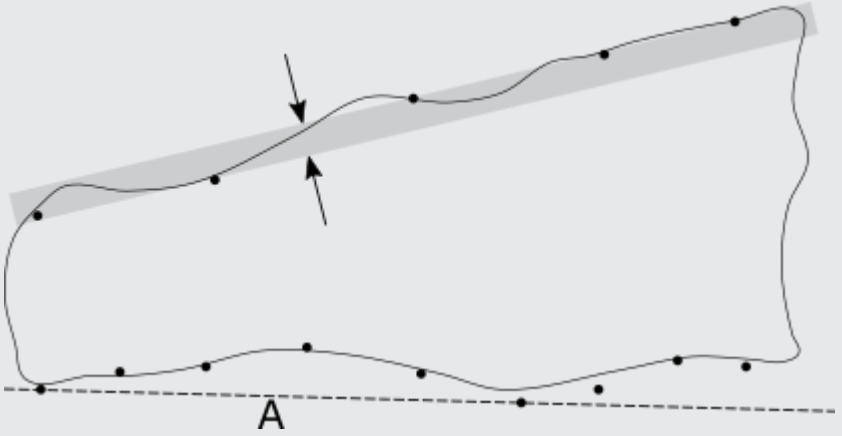


In the above specification, the actual value looks like this:



In the above image, the actual part surface uses the solid line, the actual datum uses the dashed line, and the smallest tolerance zone containing the actual toleranced feature is shown in the shaded area. The tolerance zone is exactly 15° to the actual datum.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



In the above image, the measured tolerance zone is exactly 15° to the measured datum. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Validity Rules

The considered feature or features are specified at some nominal angle to the datum feature or features.

Because of this, all input features (considered and datum) must have the correct specified nominal values. This also ensures that the tolerance command correctly identifies the optimizable degrees of freedom.

For a planar zone on axial features, the datum reference frame must fully constrain the tolerance zone orientation. The planar tolerance zone's surface normal must be perpendicular to each considered feature's axis vector.

Comparison to Past Practice 1

In PC-DMIS version 2020 R2 and later, you can no longer type in the nominal angle from the considered feature to the primary datum. Instead, you need to ensure that the features (considered and datum) have the correct nominal values.

Comparison to Past Practice 2

In PC-DMIS version 2020 R2 and later, material modifiers on datums are no longer allowed.

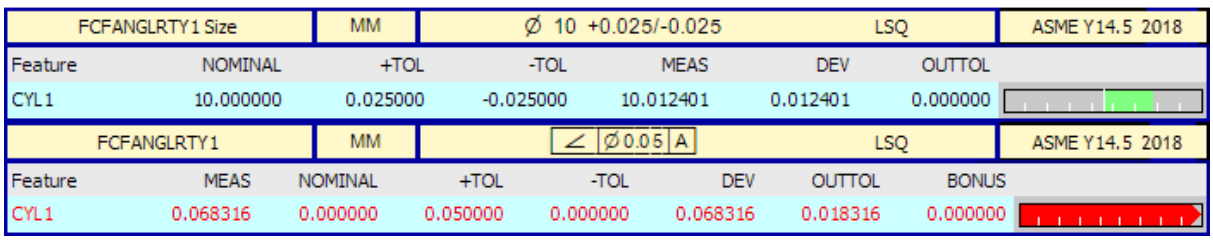
Exposed Options

When the considered feature has surface data, and the toleranced feature differs from the considered feature's surface data (cones, cylinders, and widths), the feature math type controls how to compute the toleranced feature from the considered feature's surface data. For information, see "Deriving the Toleranced Feature".

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for an angularity tolerance. The cylinder's size tolerance is in the upper label, and the diametric-zone angularity is in the lower label.



FCFANGLRTY1 Size	MM	Ø 10 +0.025/-0.025			LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL1	10.000000	0.025000	-0.025000	10.012401	0.012401	0.000000

FCFANGLRTY1	MM	∠ Ø0.05 A			LSQ	ASME Y14.5 2018
Feature	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL	BONUS
CYL1	0.068316	0.000000	0.050000	0.000000	0.068316	0.018316	0.000000

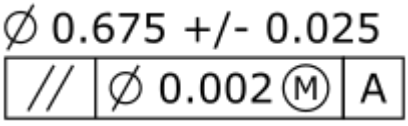
Parallelism

In this Topic

Parallelism
Introduction
Allowed Feature Types
Allowed Modifiers
Tolerance Zone Shapes
Actual Value and Measured Value
Validity Rules
Comparison to Past Practice
Exposed Options
Report

Introduction

A parallelism specification controls how much the feature can deviate from being perfectly parallel to a datum. Sometimes you can use a secondary datum to control the tolerance zone's orientation.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature".

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes.

Each toleranced feature is optimized independently.

Allowed Feature Types

You can use these feature types:

Cylinders, cones, planes, lines, 3D widths, and 2D widths

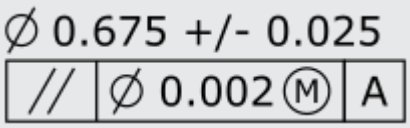
Some feature types have a different toleranced feature than the considered feature's surface data. These include 3D constructed BF lines, cylinders, cones, 3D widths, 2D widths, and planes with a tangent plane modifier. For more information, see "Deriving the Toleranced Feature".

Allowed Modifiers

When the considered feature is a cylinder or a width, this geometric tolerance type allows a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, it allows a least material modifier Ⓛ to indicate the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or the unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

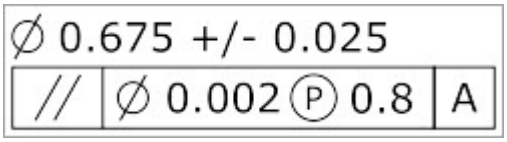
Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Suppose a cylindrical hole has a parallelism tolerance 0.002 at MMC, as shown here:



The size tolerance is 0.675 plus or minus 0.025, meaning the range of acceptable sizes is 0.650 to 0.700. The maximum material condition is then 0.650. If the unrelated measured mating envelope size is 0.661, then the bonus tolerance is 0.011, and the total tolerance is 0.013.

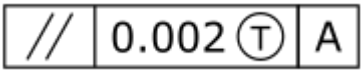
You can use a projected-zone modifier Ⓟ as shown here:



This projects (extrapolates) the measured feature axis as described in "Deriving the Toleranced Feature".

PC-DMIS only allows you to use projected tolerance zones on Auto Cylinder features. If you try to add a projected-zone modifier for any other feature type, the geometric tolerance command gives an error message stating that the feature type is invalid. This is because the projected tolerance zone needs to start at the nominal end-face of the cylinder. Measured cylinders and constructed cylinders typically do not place the nominal start point at the nominal end-face.

When the considered feature is a plane with surface data, you can use a tangent-plane modifier Ⓣ, as shown here:



This makes the toleranced feature be a perfect-form plane tangent to the actual surface, as described in "Deriving the Toleranced Feature".

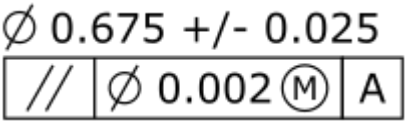
Tolerance Zone Shapes

When the considered feature has an axis, the tolerance zone shape can be diametric (with the tolerance zone shape symbol ⌀) or planar (no tolerance zone shape symbol). These are the axial considered features:

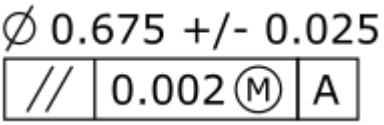
- Cylinder
- Cone
- Surfaceless axis

For information on the feature command types that correspond to these feature types, see "Feature Types With and Without Surface Data".

Here is an example of a parallelism on a cylinder with a diametric tolerance zone:



Here is an example of a parallelism on a cylinder with a planar tolerance zone:



Planar tolerance zones on axial considered features need a specified tolerance zone orientation. This is because the feature does not have enough information to orient the zone correctly. In these cases, the **Zone Orientation** button in the **Geometric Tolerance** dialog box becomes visible. For information on how to use this button to

change the zone orientation, see "Zone Orientation" in the "Feature Control Frame Tab" topic.

If a tolerance controls the X component of the orientation, the tolerance zone surface normal vector should be X.

When the considered feature is a plane, surface line, 3D width, or 2D width, the tolerance zone shape is always planar. It is oriented parallel to the nominal surface or surfaces.

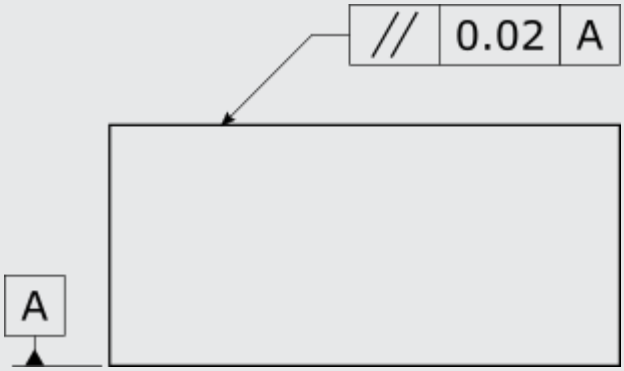
You can have more than one considered feature, but those features must all be the same type.

Actual Value and Measured Value

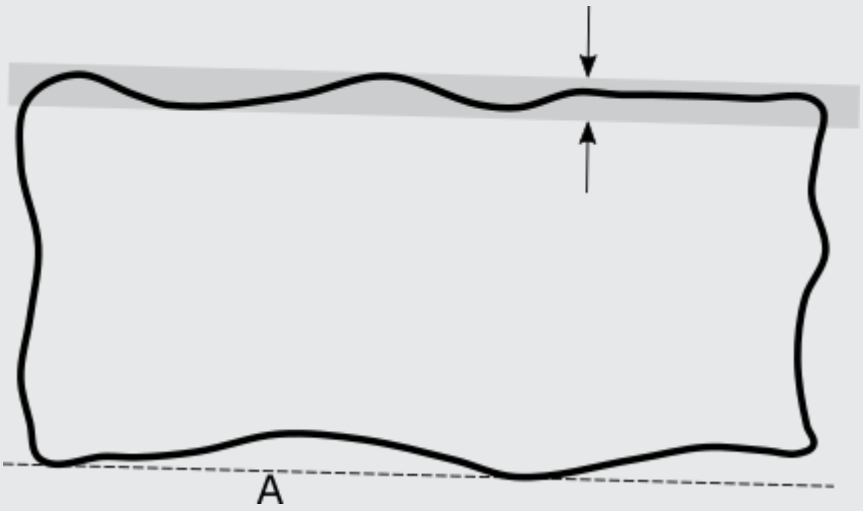
Actual Value:
This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented to the actual datum or datums.

Measured Value:
This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented to the measured datum or datums.

Suppose you had this parallelism specification:

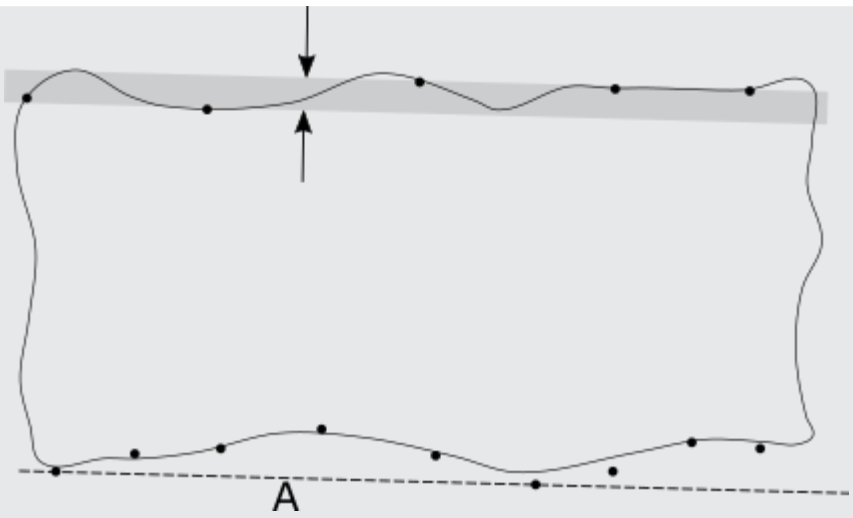


For the above specification, the actual value looks like this:



In the above image, the actual part surface uses the solid line, the actual datum uses the dashed line, and the smallest tolerance zone containing the actual toleranced feature is shown in the shaded area. The tolerance zone is exactly parallel to the actual datum.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone is exactly parallel to the measured datum. In this case, because the measured points were not measured densely enough, the measured value is smaller than the actual value.

Validity Rules

Each considered feature must be nominally parallel to the primary datum.

Because of this, all input features (considered and datum) must have the correct specified nominal values. This ensures that the tolerance zone orients correctly based on the datum features. It also ensures that the geometric tolerance command correctly identifies the optimizable degrees of freedom.

For a planar zone on axial features, the datum reference frame must fully constrain the tolerance zone orientation. The planar tolerance zone’s surface normal must be perpendicular to each considered feature's axis vector.

Comparison to Past Practice

In PC-DMIS version 2020 R2 and later, material modifiers on datums are no longer allowed.

Exposed Options

When the considered feature has surface data, and the toleranced feature differs from the considered feature’s surface data (cones, cylinders, and widths), the feature math type controls how to compute the toleranced feature from the considered feature’s surface data. For information, see "Deriving the Toleranced Feature”.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for a parallelism tolerance. The cylinder's size tolerance is in the upper label, and the diametric-zone parallelism is in the lower label.

FCFPARL1 Size	MM	Ø 15 +0.025/-0.025		LSQ		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	15.000000	0.025000	-0.025000	15.421644	0.421644	0.396644

FCFPARL1	MM	// Ø0.2 D		LSQ		ASME Y14.5 2018
Feature	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL	BONUS
CYL2	0.132449	0.000000	0.200000	0.000000	0.132449	0.000000	0.000000

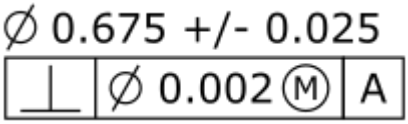
Perpendicularity

In this Topic

- Perpendicularity
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Tolerance Zone Shapes
  - Actual Value and Measured Value
  - Validity Rules
  - Comparison to Past Practice
  - Exposed Options
  - Report

Introduction

A perpendicularity specification controls how much the feature can deviate from a perfect 90 degree angle to a datum. Sometimes you can use a secondary datum to further control the tolerance zone’s orientation.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature”.

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes.

Each toleranced feature is optimized independently.

Allowed Feature Types

You can use these feature types:

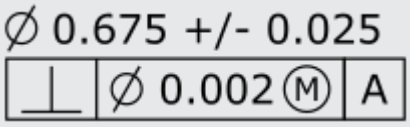
Cylinders, cones, planes, lines, 3D widths, and 2D widths

Some feature types have a different toleranced feature than the considered feature's surface data. These include 3D constructed BF lines, cylinders, cones, 3D widths, 2D widths, and planes with a tangent plane modifier. For more information, see "Deriving the Toleranced Feature".

Allowed Modifiers

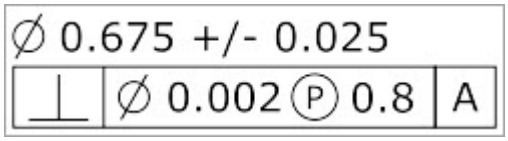
Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Suppose a cylindrical hole has a perpendicularity tolerance 0.002 at MMC, as shown here:



The size tolerance is 0.675 plus or minus 0.025, meaning the range of acceptable sizes is 0.650 to 0.700. The maximum material condition is then 0.650. If the unrelated measured mating envelope size is 0.661, then the bonus tolerance is 0.011, and the total tolerance is 0.013.

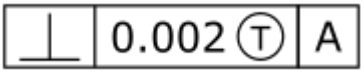
You can use a projected-zone modifier Ⓟ as shown here:



This projects (extrapolates) the measured feature axis as described in "Deriving the Toleranced Feature".

PC-DMIS only allows you to use projected tolerance zones on Auto Cylinder features. If you try to add a projected-zone modifier for any other feature type, the geometric tolerance command gives an error message stating that the feature type is invalid. This is because the projected tolerance zone needs to start at the nominal end-face of the cylinder. Measured cylinders and constructed cylinders typically do not place the nominal start point at the nominal end-face.

When the considered feature is a plane with surface data, you can use a tangent-plane modifier Ⓣ, as shown here:



This makes the toleranced feature be a perfect-form plane tangent to the actual surface, as described in "Deriving the Toleranced Feature".

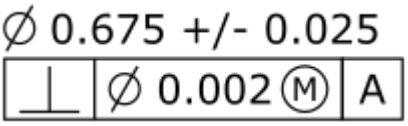
Tolerance Zone Shapes

When the considered feature has an axis, the tolerance zone shape can be diametric (with the tolerance zone shape symbol ⌀) or planar (no tolerance zone shape symbol). These are the axial considered features:

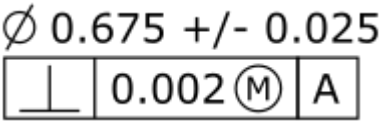
- Cylinder
- Cone
- Surfaceless axis

For information on the feature command types that correspond to these feature types, see "Feature Types With and Without Surface Data".

Here is an example of a perpendicularity on a cylinder with a diametric tolerance zone:



Here is an example of a perpendicularity on a cylinder with a planar tolerance zone:



Planar tolerance zones on axial considered features need a specified tolerance zone orientation. This is because the feature does not have enough information to orient the zone correctly. In these cases, the **Zone Orientation** button in the **Geometric**

Tolerance dialog box becomes visible. For information on how to use this button to change the zone orientation, see "Zone Orientation" in the "Feature Control Frame Tab" topic.

If a tolerance controls the X component of the orientation, the tolerance zone surface normal vector should be X.

When the considered feature is a plane, surface line, 3D width, or 2D width, the tolerance zone shape is always planar. It is oriented parallel to the nominal surface or surfaces.

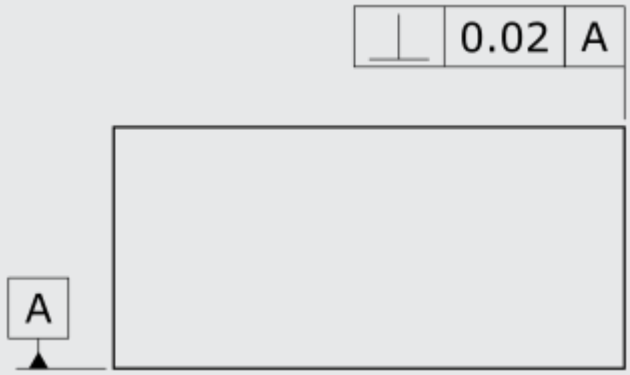
You can have more than one considered feature, but those features must all be the same type.

Actual Value and Measured Value

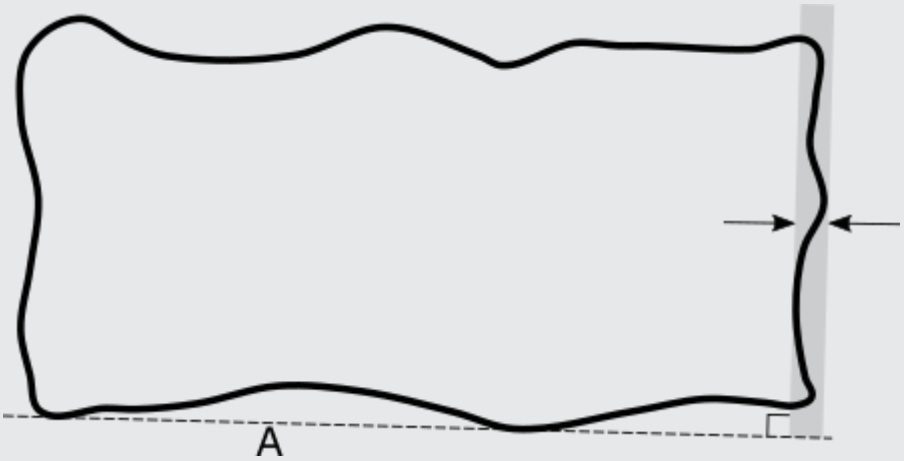
Actual Value:
This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented to the actual datum or datums.

Measured Value:
This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented to the measured datum or datums.

Suppose you have this perpendicularity specification:

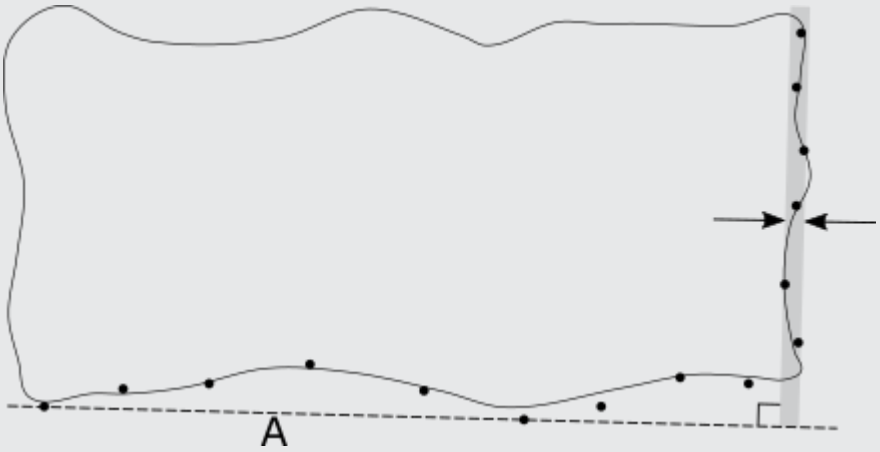


For the above specification, the actual value looks like this:



In the above image, the actual part surface uses the solid line, the actual datum uses the dashed line, and the smallest tolerance zone containing the actual toleranced feature is shown in the shaded area. The tolerance zone is exactly perpendicular to the actual datum.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



In the above image, the measured tolerance zone is exactly perpendicular to the measured datum. In this case, because the measured points were not measured densely enough, the measured value is smaller than the actual value.

Validity Rules

The considered feature or features must be nominally perpendicular to the primary datum.

Because of this, all input features (considered and datum) must have the correct specified nominal values. This ensures that the tolerance zone orients correctly based on the datum features. It also ensures that the geometric tolerance command correctly identifies the optimizable degrees of freedom.

For a planar zone on axial features, the datum reference frame must fully constrain the tolerance zone orientation. The planar tolerance zone’s surface normal must be perpendicular to each considered feature's axis vector.

Comparison to Past Practice

In PC-DMIS version 2020 R2 and later, material modifiers on datums are no longer allowed.

Exposed Options

When the considered feature has surface data, and the toleranced feature differs from the considered feature’s surface data (cones, cylinders, and widths), the feature math type controls how to compute the toleranced feature from the considered feature’s surface data. For information, see "Deriving the Toleranced Feature”.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for a perpendicularity tolerance. The cylinder's size tolerance is in the upper label, and the diametric-zone perpendicularity is in the lower label.

FCFPERP3 Size	MM	Ø 60.5 +0.025/-0.025		LSQ		ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
D_CB	60.500000	0.025000	-0.025000	60.786416	0.286416	0.261416

FCFPERP3	MM	⊥ Ø0.05 A		LSQ		ASME Y14.5 2018
Feature	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL	BONUS
D_CB	0.021472	0.000000	0.050000	0.000000	0.021472	0.000000	0.000000

Concentricity

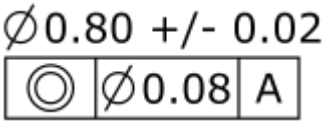
In this Topic

- Concentricity
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Tolerance Zone Shapes
  - Actual Value and Measured Value
  - Validity Rules
  - Exposed Options
  - Report

Introduction

> When you select ASME Y14.5 – 2018 as the GDT standard, two past practices, use of concentricity and use of symmetry symbols, are no longer supported. For more detail, see the main foreword and sections A-5.3, A-8.4, D-3 and D4 of ASME Y14.5 – 2018.

A concentricity specification controls how much the feature can deviate from being concentric to one or more datums.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature".

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes.

Allowed Feature Types

You can use these feature types:

3D constructed BF lines, cast lines, generic lines, cylinders, circles, spheres, and cones

PC-DMIS constructs the toleranced feature based on the standard you use (ASME Y14.5 or ISO 1101).

ISO 1101 (or with a feature that does not have surface data):

PC-DMIS constructs the toleranced feature in the same way as position tolerances do.

ASME Y14.5 with cylinders, circles, spheres, and cones with surface data:

PC-DMIS provides an option to switch between MEDIAN_POINTS or AXIS:

AXIS - The software constructs the toleranced feature as the axis of the unrelated mating envelope (this is the same as position tolerances).

MEDIAN_POINTS - The software constructs the toleranced feature from all the median points of the feature. It does this according to paragraph 7.6.4.2.2 of ASME Y14.5 2009.

Allowed Modifiers

When the considered feature is a cylinder, circle, or sphere, concentricity tolerances referencing ISO 1101 allow a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, they allow a least material modifier Ⓛ to indicate the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or the unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Tolerance Zone Shapes

For ISO, the tolerance zone shape is always diametric. It is oriented parallel to the datum axis.

For ASME, the tolerance shapes are usually diametric, but sphere features can have a spherical or a diametric zone.

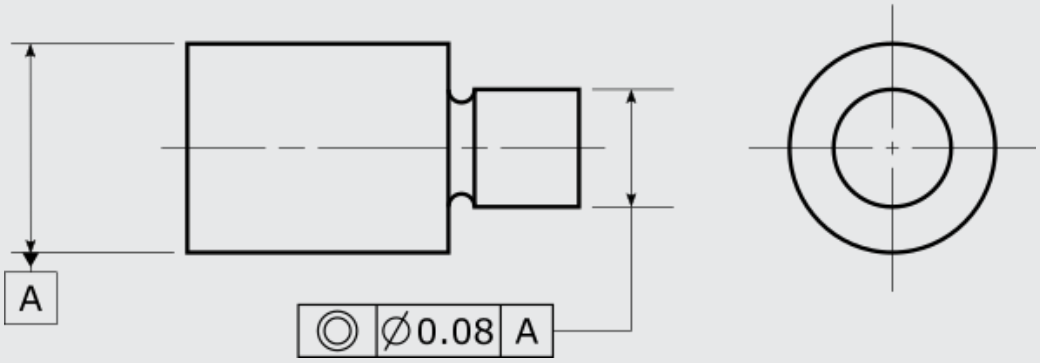
Actual Value and Measured Value

There is one special case to consider. If you have a spherical-zone concentricity of two or more spheres (thus, ASME concentricity) it is not clear from the standard whether the spheres should be considered simultaneously or independently. The PC-DMIS geometric tolerance command considers them simultaneously, since this is the more conservative choice.

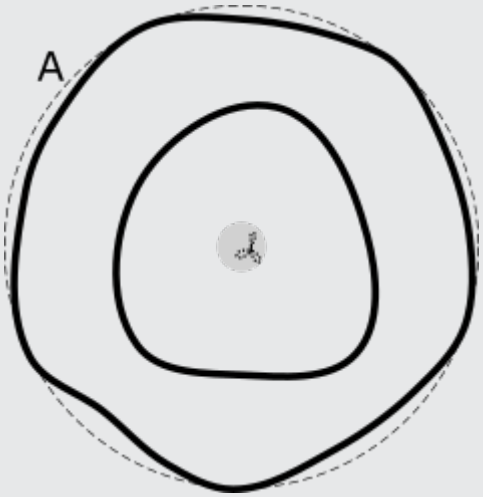
Actual Value:
This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented and located to each actual datum.

Measured Value:
This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented and located to each measured datum.

Suppose you have this concentricity specification:

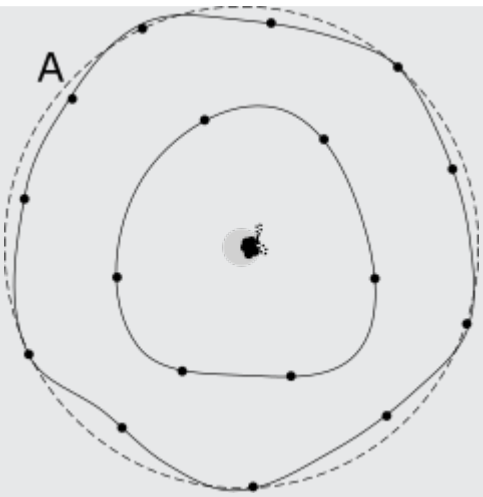


With the above specification, the actual value looks like this:



The actual part surface uses the solid line, the actual datum uses the dashed line, the toleranced feature uses the dotted line, and the smallest tolerance zone that contains the actual toleranced feature is shown in the shaded area. The tolerance zone is exactly concentric to the axis of the actual datum.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone is exactly concentric to the axis of the measured datum. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Validity Rules

All input features (considered and datum) must have the correct specified nominal values. This ensures that the measured values are calculated correctly, and that the tolerance command correctly identifies the optimizable degrees of freedom.

The datum reference frame must be axial, and the considered feature surface or surfaces must be nominally concentric with the datum axis.

Exposed Options

Concentricity tolerances have a feature math type when the considered feature has surface data.

This math type controls how to compute the toleranced feature from the considered feature's surface data. For more information, see "Deriving the Toleranced Feature".

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for a concentricity tolerance of a cylinder. The cylinder's size tolerance is in the upper label, and the concentricity is in the lower label.

FCFCONCEN1 Size		IN	Ø 0.8 +0.02/-0.02		LSQ	ASME Y14.5 2009
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	0.8000	0.0200	-0.0200	0.8027	0.0027	0.0000
FCFCONCEN1		IN	⌾ Ø0.08 A AXIS		LSQ	ASME Y14.5 2009
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	0.0000	0.0800	0.0000	0.0357	0.0357	0.0000

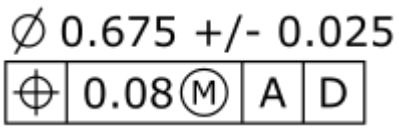
Position

In this Topic

- Position
  - Introduction
  - Allowed Feature Types
  - Allowed Modifiers
  - Tolerance Zone Shapes
  - Actual Value and Measured Value
  - Validity Rules
  - Exposed Options
  - Lower Segments of Composite Position
  - Report

Introduction

A position specification controls how much the considered feature or features can deviate from a specified position relative to zero or more datums.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features (if any are referenced)

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature".

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes. When there are multiple considered features, the optimization process simultaneously considers all features, so that all toleranced features are fitted into their tolerance zones at once. This process is similar to a physical gage, where all of the gage pins must fit into the workpiece's holes at the same time.

Allowed Feature Types

You can use these feature types:

Spheres, 3D surfaceless points, cylinders, circles, cones, widths, slots, notches, constructed mid planes, constructed mid lines, and constructed mid points.

ASME position tolerances also allow 3D constructed BF lines. ISO position tolerances also allow planes, lines, and surface points.

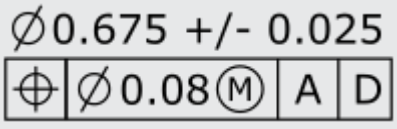
3D constructed BF lines, spheres, cylinders, circles, cones, widths, slots, and notches have a toleranced feature that is different than the considered feature's surface data. For information, see "Deriving the Toleranced Feature".

Allowed Modifiers

When the considered feature is a cylinder, sphere, or width, position tolerances allow a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, they allow a least material modifier Ⓛ to indicate the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

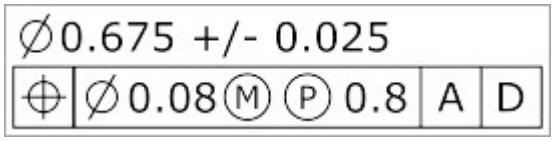
Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

This example is in inches. Suppose a cylindrical hole has a position tolerance 0.08 at MMC, as shown here:



The size tolerance is 0.675 plus or minus 0.025, meaning the range of acceptable sizes is 0.650 to 0.700. The maximum material condition is then 0.650. If the unrelated measured mating envelope size is 0.661, then the bonus tolerance is 0.011, and the total tolerance is 0.091.

You can use a projected-zone modifier Ⓟ as shown here:



This projects (extrapolates) the measured feature axis as described in "Deriving the Toleranced Feature".

> **!** PC-DMIS only allows you to use projected tolerance zones on Auto Cylinder features. If you try to add a projected-zone modifier for any other feature type, the geometric tolerance command gives an error message stating that the feature type is invalid. This is because the projected tolerance zone needs to start at the nominal end-face of the cylinder. Measured cylinders and constructed cylinders typically do not place the nominal start point at the nominal end-face.

Tolerance Zone Shapes

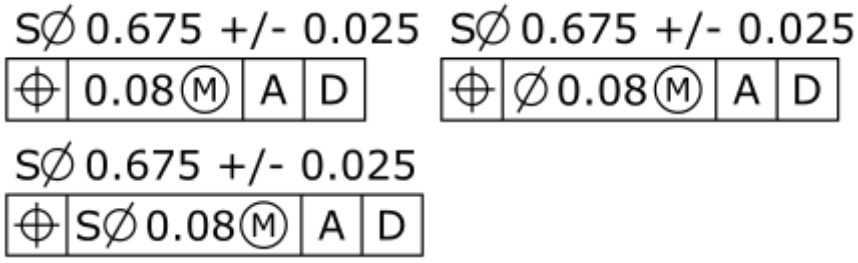
Different tolerance zone shapes are allowed for different types of features. For information about the feature command types that refer to different feature types, see "Feature Types With and Without Surface Data".

Point-like considered features

When the considered feature is point-like, the tolerance zone shape may be planar, diametrical, or spherical. These are point-like considered features:

- Sphere or 3D surfaceless point

From left to right, the pictures below show FCFs with a planar, diametric, and spherical tolerance zone when the considered feature is a sphere:



Planar and diametric tolerance zones on point-like considered features need a specified tolerance zone orientation, because the feature does not have enough information to orient the zone correctly. In these cases, the **Zone Orientation** button in the **Geometric Tolerance** dialog box becomes visible. For information on how to use this button to change the zone orientation, see "Zone Orientation" in the "Feature Control Frame Tab" topic.

If a position tolerance controls the X component of the position (planar tolerance zone), the tolerance zone surface normal vector should be X.

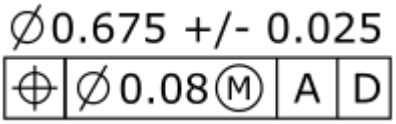
If a position tolerance controls the X and Y components of the position (diametric tolerance zone), the tolerance zone axis vector should be Z.

Axial Considered Features

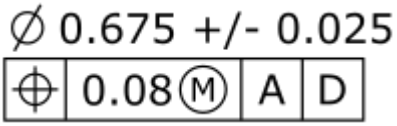
When the considered feature is axial, the tolerance zone may be planar, diametrical, radial arc, or right angle to radial. These are axial considered features:

Cylinder, circular cross section of a cylinder, cone, or surfaceless axis

A diametric tolerance zone is shown below, using the diametric zone symbol.



Planar zones, radial arc zones, and right angle to radial zones do not use any tolerance zone shape symbol and are shown below.



Planar tolerance zones on axial considered features need a specified tolerance zone orientation, because the feature does not have enough information to orient the zone correctly. In these cases, the **Zone Orientation** button in the **Geometric Tolerance** dialog box becomes visible. For information on how to use this button to change the zone orientation, see "Zone Orientation" in the "Feature Control Frame Tab" topic. This button also allows users to specify a radial arc or right angle to radial zone.

If a position tolerance controls the X component of the position, the tolerance zone surface normal vector should be X.

Plane-Like Considered Features

When the considered feature is plane-like, the tolerance zone is always planar and oriented parallel to the nominal surfaces. These are plane-like considered features:

Plane, surface line, width, slot, notch, surface point, or mid-point

You can have more than one considered feature, but those features must all be the same type.

Be careful with slots and notches.

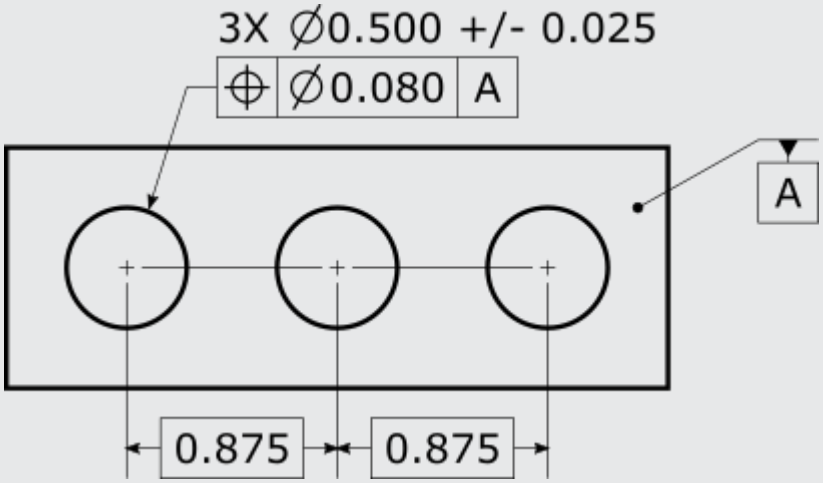
You should only use them if you already know that the form of the features is very good. If you suspect that the manufactured form error might be significant, do not use a slot or notch command. Instead, measure a scan around the perimeter of the feature and then tolerance the form, orientation, and location of the feature with a profile of a line tolerance.

Actual Value and Measured Value

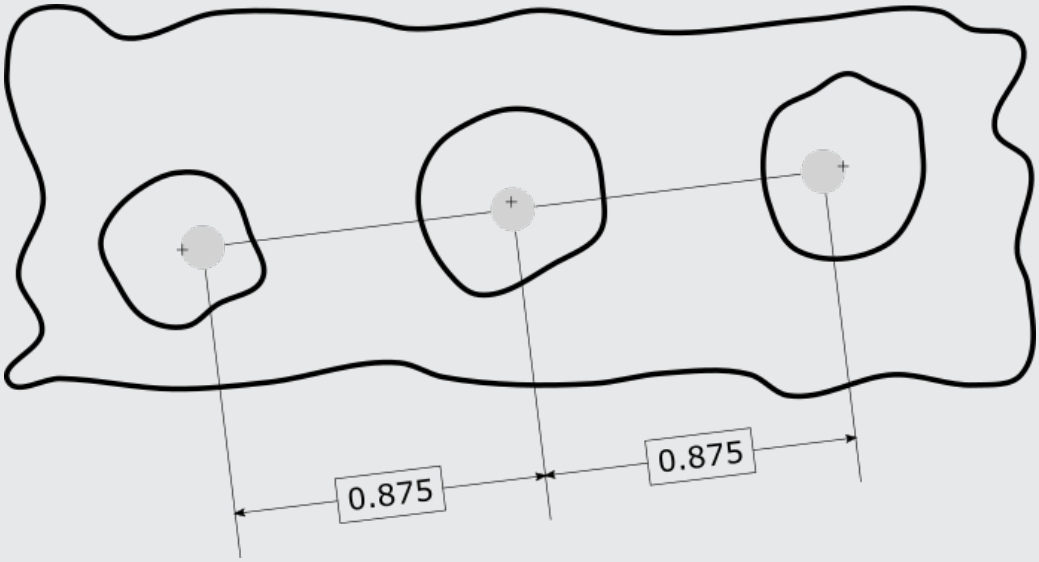
Actual Value:
Each considered feature has its own actual value. This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented and located to each actual datum, with some exceptions detailed in "How PC-DMIS Solves Datums". When the position tolerance contains more than one considered feature, and the datum reference frame is not fully constrained, an optimization procedure must simultaneously fit all toleranced features in their respective tolerance zones if possible.

Measured Value:
Each considered feature has its own measured value. This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented and located to each measured datum, with some exceptions detailed in "How PC-DMIS Solves Datums". When the position tolerance contains more than one considered feature, and the datum reference frame is not fully constrained, the PC-DMIS optimization procedure simultaneously fits all toleranced features into their respective tolerance zones in a proportional way, which guarantees all toleranced features will fit in their respective tolerance zones if at all possible.

Suppose you have this position specification:



With the above specification, the actual value looks like this:



The actual part surface uses the solid line, the actual toleranced features are the small crosses, and the smallest tolerance zones containing the actual toleranced features are shown in the shaded areas. The tolerance zones are nominally located and oriented with respect to each other and with respect to the datum.

Validity Rules

Considered features must have some nominal location and orientation to each datum feature.

All input features (considered and datum) must have the correct specified nominal values. This ensures that the measured values are calculated correctly, and that the tolerance command correctly identifies the optimizable degrees of freedom.

For a planar zone on axial features, the datum reference frame must fully constrain the tolerance zone orientation. The planar tolerance zone’s surface normal must be perpendicular to each considered feature's axis vector.

For radial arc and right angle to radial tolerance zones on axial features, follow these requirements:

- The datum reference frame must establish a clear polar origin and polar axis.
- The axial features must be nominally parallel to the polar axis.

Exposed Options

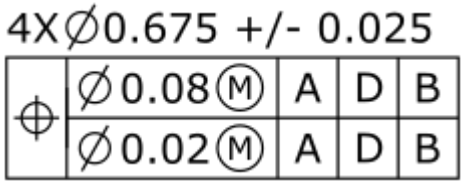
When the considered feature has surface data and the toleranced feature differs from the considered feature's surface data (spheres, cones, cylinders, circles, widths), the feature math type controls how to compute the tolerance feature from the considered feature's surface data. For information, see "Deriving the Toleranced Feature”.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Slots can be considered in either a lengthwise or a widthwise fashion, as described in "Lengthwise versus Widthwise Slots".

Lower Segments of Composite Position

A position tolerance with multiple segments is called a “composite position”. Composite position tolerances are typically specified on a pattern of features. The first (or upper) segment of a composite position is the same as a single segment position as described in the preceding sections of this page. All lower segments of a composite position are subtly different. This is because the tolerance zones of the pattern have unlocked translation compared to the datum reference frame. However, the tolerance zones remain nominally located and oriented to each other.



The datum reference frames for the lower segments of a composite position follow these rules:

- Each datum reference frame must only use the same datums as the reference frame above it.
- The datums must be in the same order.

- The datums must have the same modifiers.
- A lower segment can have fewer datums than the segment above.

Suppose the upper segment has datums ABC. The lower segment then could reference no datums, datum A, datums AB, or datums ABC. But it could not reference datums BA nor AC nor ABD.

Here are some examples of allowed composite position tolerances:

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	A	B	C

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	A	B	

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	A		

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ			

Here are some examples of not-allowed composite position tolerances:

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	B	A	

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	A	C	

4X∅0.675 +/- 0.025

⌖	∅0.08Ⓜ	A	B	C
	∅0.02Ⓜ	A	B	D

Report

Here is an example report for a position tolerance of two cylinders. The cylinders' size tolerance is in the upper label, and the diametric-zone position is in the lower label. The lower label includes information for the YZ positions of the cylinders, in the optimized frame (not in the current alignment).

FCFLOC1 Size	MM	Ø 8 +0.1/-0.1	DEFAULT	ASME Y14.5 2018

Feature	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL
CYL1	7.995	8.000	0.100	0.100	-0.005	0.000
CYL2	7.990	8.000	0.100	0.100	-0.010	0.000

FCFLOC1	MM	⌖ Ø0.2 Ⓜ A D E	DEFAULT	ASME Y14.5 2018

Feature	AX	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL	BONUS
CYL1 (END PT)	Y	-6.912	-7.000			0.088		
	Z	-27.992	-28.000			0.008		
	TP	0.176	0.000	0.200	0.000	0.176	0.000	0.095
CYL2 (END PT)	Y	7.092	7.000			0.092		
	Z	-27.989	-28.000			0.011		
	TP	0.184	0.000	0.200	0.000	0.184	0.000	0.090

In the lower label of the above illustration, the Y, Z, and TP rows for CYL1 and CYL2 summarize the position evaluation for each considered feature. The bottom row of each, labeled TP, behaves just like all of the reports for other geometric tolerances. For CYL2, the TP row includes the tolerance 0.200, the bonus 0.090, and the measured value 0.184.

In the lower label of the example report above, there are only three rows for each feature (Y, Z, and TP). For your reports, the axis column (AX) for your features might have some combination of several rows labeled X, Y, Z, PR, PA, and TP. When present, the X, Y, Z, PR, and PA rows give supplementary information as described below.

The Y and Z rows for CYL1 and CYL2 give supplementary information about the position evaluation. Those rows are intended to give a simplified representation of how the feature deviates from the ideal.

- The NOMINAL column of the supplementary information rows shows the nominal start point of the feature in question.
- The DEV column of those rows shows the worst deviation vector, among all of the toleranced feature's points.
- The MEAS column of those rows is the NOMINAL column plus the DEV column. In other words, it is the toleranced feature's worst point, after being projected to be as close as possible to the nominal start point, while preserving the worst deviation vector.

- Underneath the feature name, the report indicates which end or level of the cylinder was the worst. For example, it could say "(START PT)", "(END PT)", or "LEVEL#3".

The representation is simplified, because, as described in Deriving the Toleranced Feature, most feature types have more than one point in the toleranced feature. For example, cylinders typically have at least two points in the toleranced feature, the measured start point and the measured end point. By contrast, in some cases ISO cylinders have several points in the toleranced feature, one for each measured cross section.

If you turn on the textual analysis or graphical analysis, you can see all the points in the toleranced feature, together with their respective deviations. The supplementary information rows in the report has a MEAS column that does not necessarily appear in the textual analysis, because the supplementary information has been simplified to show deviation with respect to the nominal start point.

Symmetry

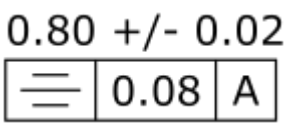
In this Topic

Symmetry
- Introduction
- Allowed Feature Types
- Allowed Modifiers
- Tolerance Zone Shapes
- Actual Value and Measured Value
- Validity Rules
- Exposed Options
- Comparison to Past Practice
- Report

Introduction

When you select ASME Y14.5 – 2018 as the GDT standard, two past practices, use of concentricity and use of symmetry symbols, are no longer supported. For more detail, see the main foreword and sections A-5.3, A-8.4, D-3 and D4 of ASME Y14.5 - 2018.

A symmetry specification controls how much the feature can deviate from being symmetric about one or more datums.



For this geometric tolerance, these three aspects work together:

- Each considered feature and each resulting toleranced feature
- Each tolerance zone
- The datum features

To evaluate this tolerance, PC-DMIS converts each considered feature to a toleranced feature. You can find this described in "Deriving the Toleranced Feature".

PC-DMIS then optimizes each toleranced feature into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes.

Allowed Feature Types

You can use these feature types:
widths, constructed mid planes, constructed mid lines, and constructed mid points

PC-DMIS constructs the toleranced feature differently based on the standard you use (ASME Y14.5 or ISO 1101).

ISO 1101 (or with a constructed mid feature or a 1D width):

PC-DMIS constructs the toleranced feature in the same ways as position toleranced features do.

ASME Y14.5 with a 2D or 3D width:

PC-DMIS provides an option to switch between MEDIAN_POINTS or AXIS:

AXIS - The software constructs the toleranced feature as the axis (center-plane) of the unrelated mating envelope (this is the same as position tolerances).

MEDIAN_POINTS - The software constructs the toleranced feature from all median points of the width. It does this according to paragraph 7.7.2 of ASME Y14.5 2009.

Allowed Modifiers

When the considered feature is a width, symmetry tolerances referencing ISO 1101 allow a maximum material modifier Ⓜ to indicate the specification is at the maximum material condition (MMC). Alternatively, they allow a least material modifier Ⓛ to indicate the specification is at the least material condition (LMC). This means that as the unrelated mating envelope size (or the unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For more information on this bonus tolerance, see "Evaluating Size with the Geometric Tolerance Command".

Tolerances referencing ISO 1101 allow additional Associated toleranced feature specification modifiers Ⓒ and Ⓖ for non-features of size. For features of size, Ⓒ, Ⓖ, Ⓝ, and Ⓧ are available. For details, see the "Deriving the Toleranced Feature" topic in the PC-DMIS Core documentation.

Tolerance Zone Shapes

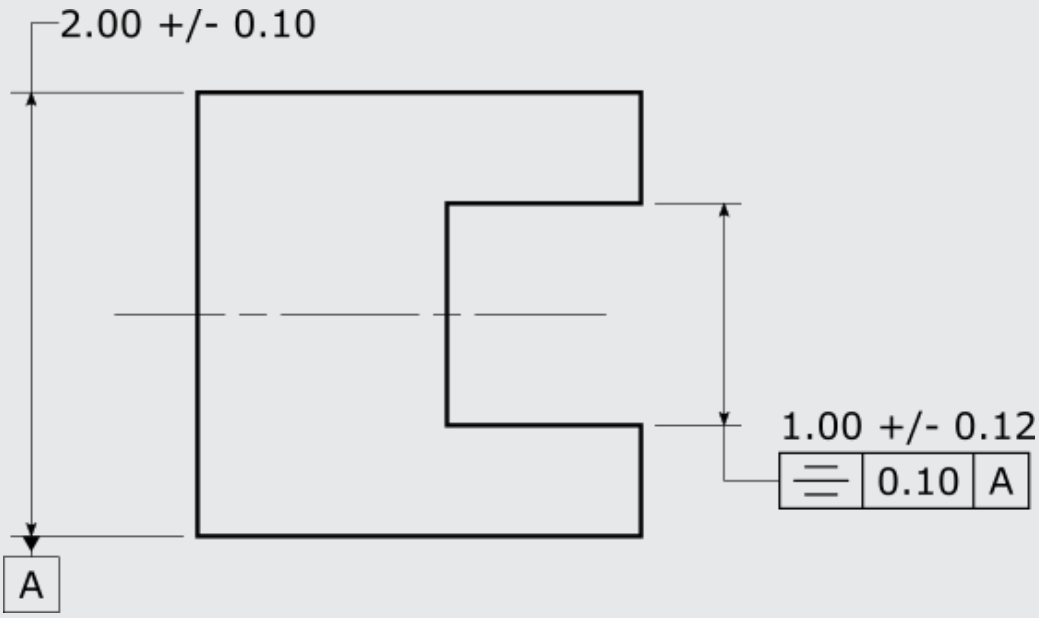
The tolerance zone shape is always planar. It is oriented parallel to the nominal surface or surfaces.

Actual Value and Measured Value

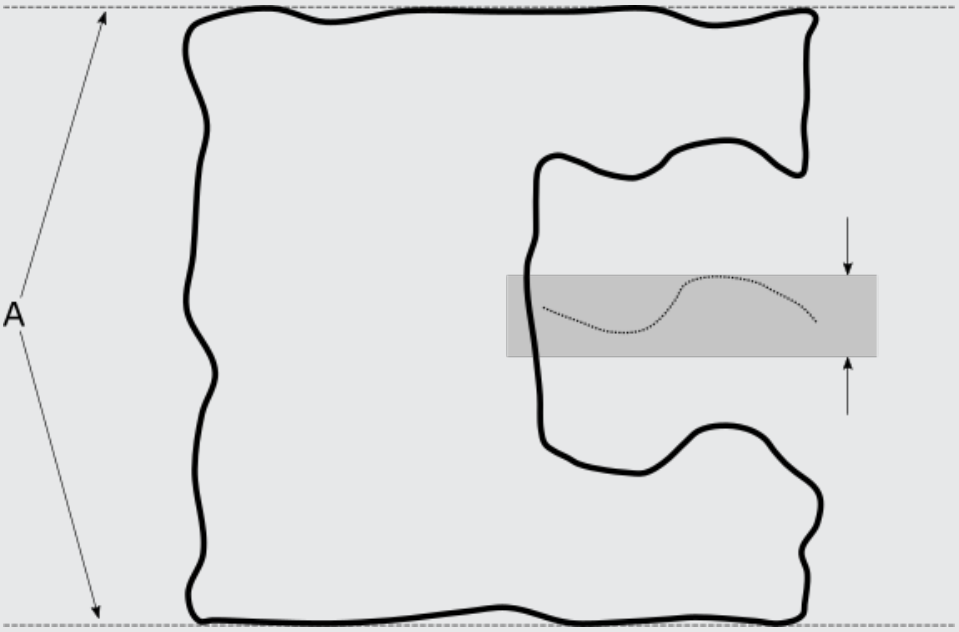
Actual Value:
This is the size of the smallest tolerance zone that contains the actual toleranced feature. The zone is nominally oriented and located to each actual datum.

Measured Value:
This is the size of the smallest tolerance zone that contains the measured toleranced feature. The zone is nominally oriented and located to each measured datum.

Suppose you have this symmetry specification:

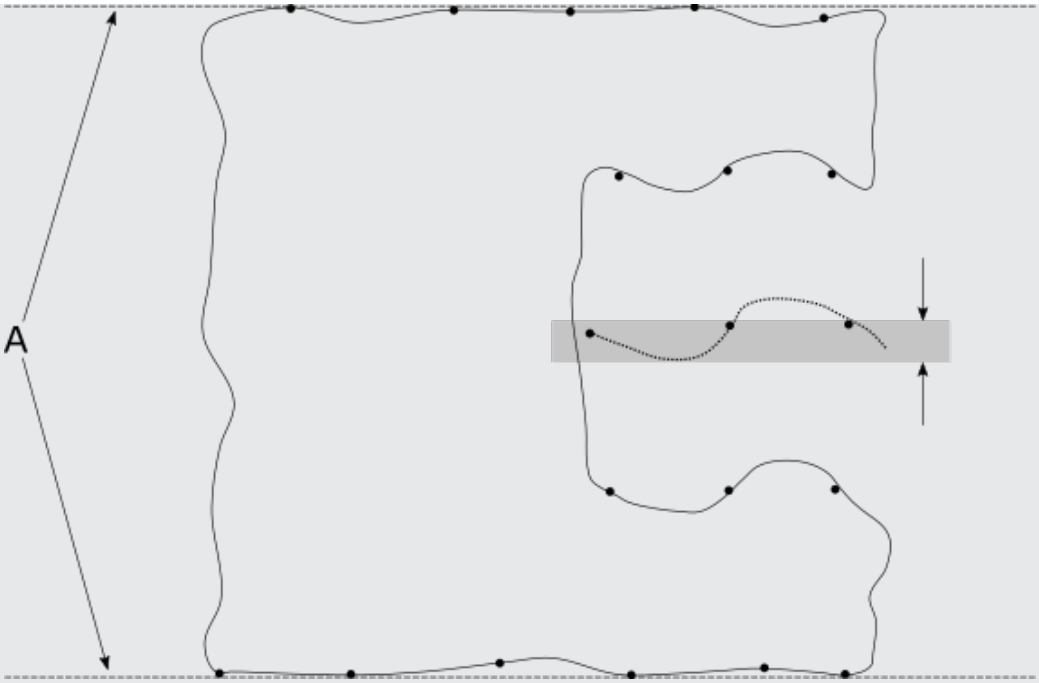


With the above specification, the actual value looks like this:



The actual part surface uses the solid line, the actual datum uses the dashed line, the toleranced feature uses the dotted line, and the smallest tolerance zone containing the actual toleranced feature is shown in the shaded area. The tolerance zone is exactly symmetric to the center-plane of the actual datum.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone is exactly symmetric to the center-plane of the measured datum. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Validity Rules

All input features (considered and datum) must have the correct specified nominal values. This ensures that the measured values are calculated correctly, and that the tolerance command correctly identifies the optimizable degrees of freedom.

The considered feature surface or surfaces must be nominally symmetric with the datum reference frame.

Exposed Options

Symmetry tolerances have a feature math type when the considered feature is a width.

This math type controls how to compute the toleranced feature from the considered feature's surface data. For more information, see "Deriving the Toleranced Feature".

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Comparison to Past Practice

For many years, with PC-DMIS symmetry tolerances, you could input pairs of planes, pairs of lines, pairs of points, or pairs of sets. Originally, this was because PC-DMIS did not have a width command. Starting with PC-DMIS 2020 R2, these kinds of considered feature pairs are no longer allowed. Each considered feature has its own measured value, and that means the best way to use a symmetry command is with a width feature.

Report

Here is an example report for a symmetry tolerance of a mid-line.

FCFSYM1		MM	⌯ 0.5 A B AXIS		LSQ	ASME Y14.5 2009
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
LINE3	0.000000	0.500000	0.000000	0.007845	0.007845	0.000000

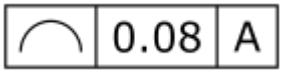
Profile of a Line

In this Topic

- Profile of a Line
  - Introduction
  - Recommended Usage
  - Regarding Simultaneous Tolerance Command
  - Allowed Feature Types
  - Tolerance Zones and Allowed Modifiers
  - Profile In Between and Profile All Around
  - Actual Value and Measured Value
  - Validity Rules
  - Exposed Options
  - Lower Segments of Composite Profile of a Line
  - Report
- Profile of a Line Per Unit Tolerances
  - Report

Introduction

A profile of a line specification controls how much the cross sections of features' surfaces can deviate from being the nominal shapes. These cross sections are located and oriented to zero or more datums.



For this geometric tolerance, these three aspects work together:

- Each considered feature's surface data
- Each considered feature's nominal shape and each resulting tolerance zone
- The datum features (if any are referenced)

To evaluate this tolerance, PC-DMIS optimizes each feature's surface data into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes. With multiple considered features, the optimization process simultaneously considers those features. In this way it fits all the toleranced features into their tolerance zones at once.

Recommended Usage

Profile of a line specifications apply to surfaces. This means each cross section of the surface must have an actual value that is less than the specified tolerance. Each cross section should be considered separately from the others (rather than simultaneously).

For each surface that has a profile of a line specification, we recommend that you measure several cross sections:

- You need to have enough cross sections to adequately capture the entire surface's behavior.
- Place each cross section in a separate geometric tolerance command, so that the cross sections are considered and optimized separately. (If you put the cross sections into just one geometric tolerance command, they are considered simultaneously and your measured values will be overly large.)

Regarding Simultaneous Tolerance Command

Strictly speaking, it does not make sense to include a profile of a line specification in a simultaneous tolerance command. If you do this, the behavior of the profile of a line specification changes. Instead of considering each cross section individually, it considers all cross sections simultaneously. This is equivalent to profile of a surface. However, PC-DMIS does permit profile of a line specifications that have at least one datum to participate in a simultaneous tolerance command. In this case, PC-DMIS shows a warning message to indicate that the behavior has changed to match that of profile of a surface.

Allowed Feature Types

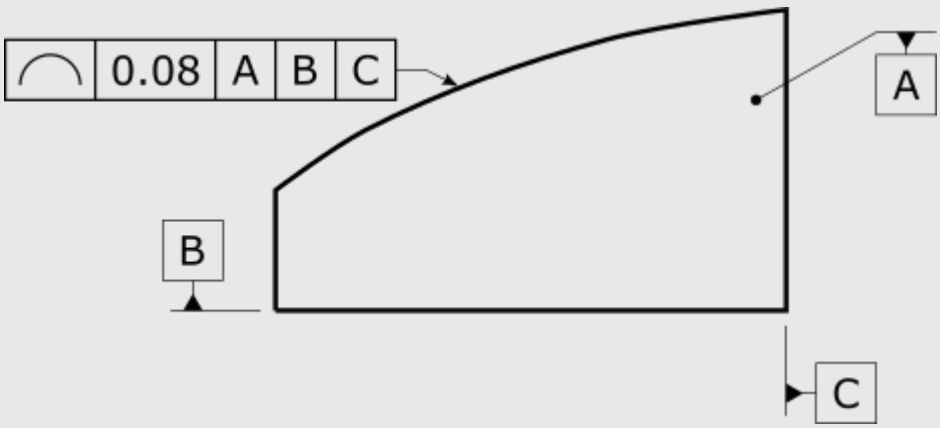
You can use these feature types to represent surface cross sections if they have surface data:

lines, circles, 2D and 1D widths, scans, ellipses, notches, slots, and sets.

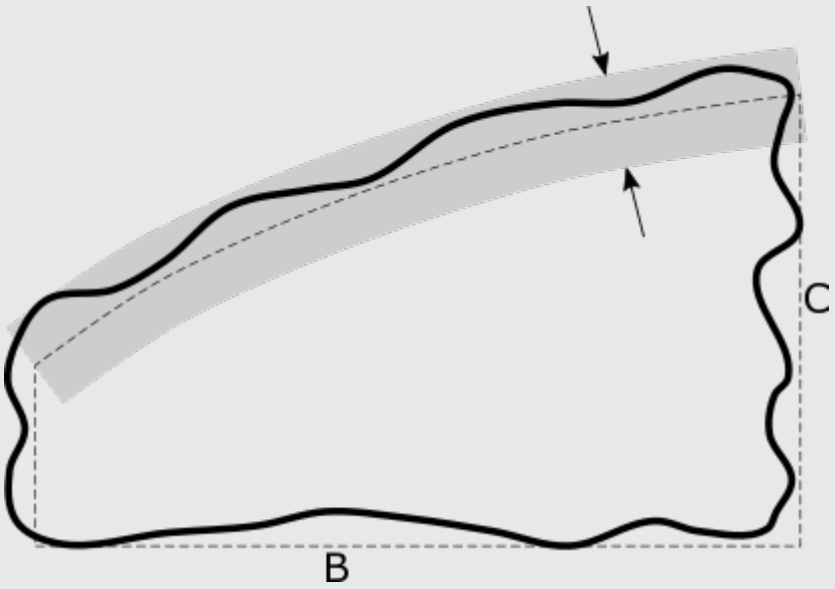
Tolerance Zones and Allowed Modifiers

The tolerance zone is based on the nominal surface of the feature. By default (without any modifiers), the tolerance zone is equal bilateral. This means that half of the tolerance value is on each side of the nominal surface:

Suppose you have this profile of a line specification:

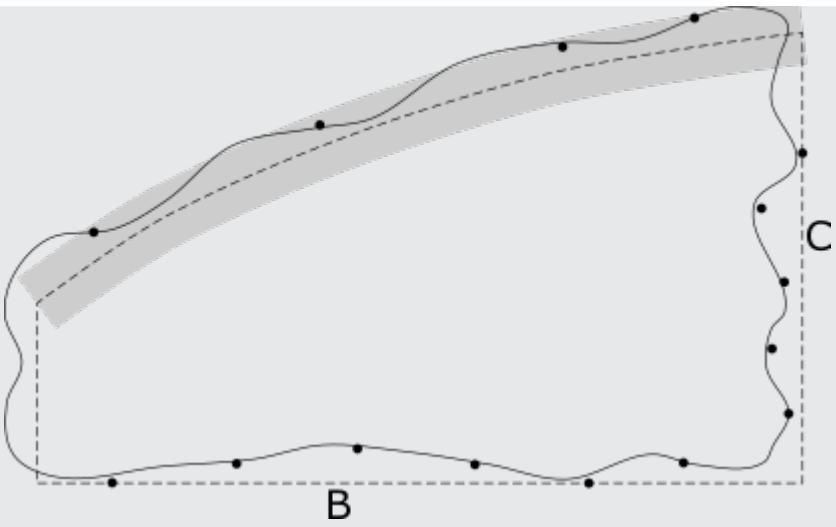


With the above specification, the actual value looks like this:



Since there are no modifiers, the tolerance zone is centered on the nominal surface, which is nominally oriented and located to each actual datum. The solid line indicates the actual surface, the dashed lines represent the nominal surfaces (including the actual datums) and the gray shaded area represents the minimum-size tolerance zone centered on the nominal surface containing the actual surface.

The measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone center remains the nominal surface, which is nominally oriented and located to each measured datum. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Modifiers may change the nature of the tolerance zone. Starting in ASME Y14.5 2009, PC-DMIS supports the Ⓤ modifier (unequally disposed profile) and, starting in ASME Y14.5 2018, PC-DMIS supports the △ modifier (dynamic profile). Under ISO 1101, PC-DMIS supports the UZ modifier (specified tolerance zone offset) and the OZ modifier (unspecified linear tolerance zone offset). While they are not equivalent, the Ⓤ and UZ modifiers have similar functionality. They offset the center of the tolerance zone from the nominal surface. Likewise, the △ and OZ modifiers have similar functionality. They allow the center of the tolerance zone to progress in the plus-material or minus-material direction. These modifiers did not exist in ASME Y14.5 1994 but it did support the concept of an unequally disposed tolerance zone. This was usually specified graphically on the print by dotted lines showing the extent of tolerance into and out of the material. Therefore, when selecting ASME Y14.5 1994 as the GDT standard for your PC-DMIS measurement routine, you must enter both the upper and lower tolerances when creating the feature control frame. PC-DMIS will then report the maximum and minimum deviations which are compared against the corresponding upper and lower tolerance values to assess conformance.

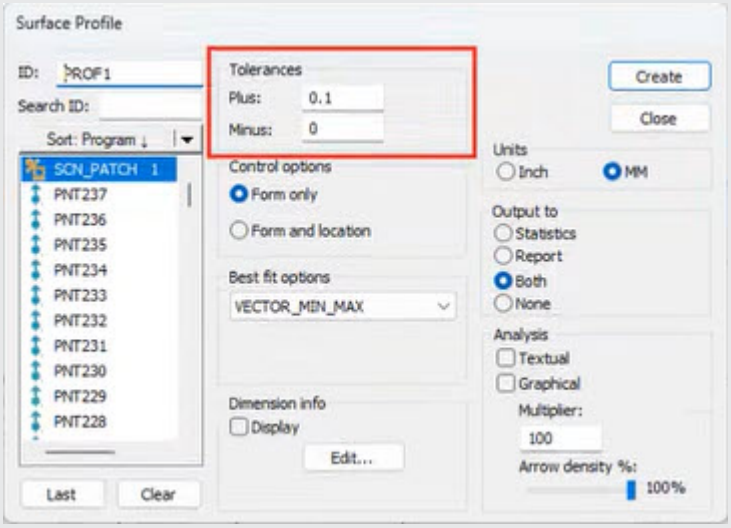
Legacy profile includes a “form only” option which only allows a single, positive tolerance to be entered. To achieve the same behavior using an ASME Y14.5 1994 geometric tolerance profile command, you must divide the total allowable tolerance in half and enter it as equal bilateral values.

The examples below show the differences between the Legacy Surface Profile and the Geometric Tolerance Profile of a Surface commands. Although the examples are for the Surface Profile, the rules are identical for the Line Profile.

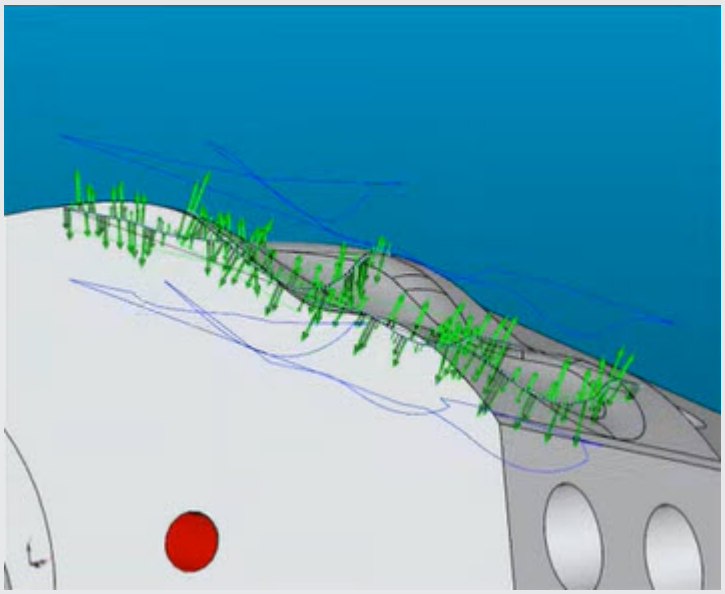
Click on the link below to show examples of the definition, result, and report for a Legacy Surface Profile:

Legacy Surface Profile, Form Only Example

The following example shows a Surface Profile dialog box with a Form only definition:



Example of a legacy Surface Profile dialog box defined for form only with an upper tolerance of 0.1 and a lower tolerance of 0.



Example of the results after the legacy command is executed.

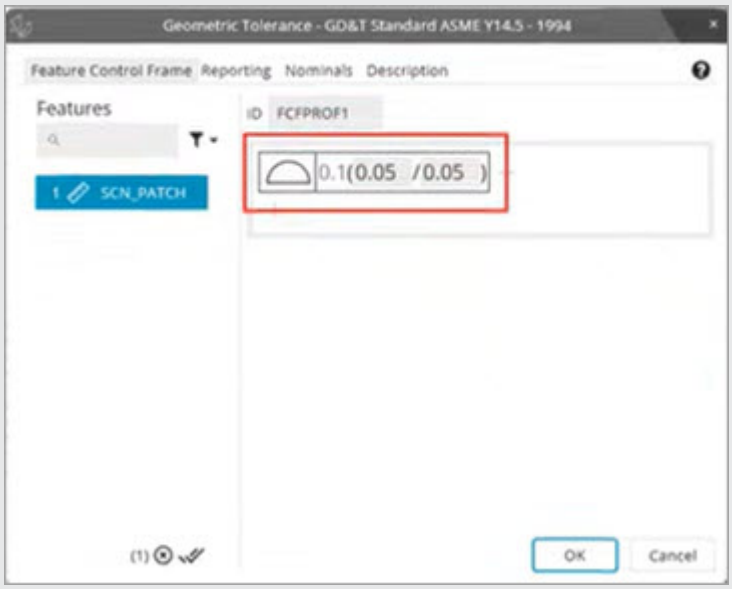
	MM	PROF1 - SCN_PATCH FORMONLY VECTOR_MIN_MAX					
AX	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL
M	0.000000	0.100000	0.000000	0.071202	0.035601	-0.035601	0.000000

Example of what the legacy version of the report might look like.

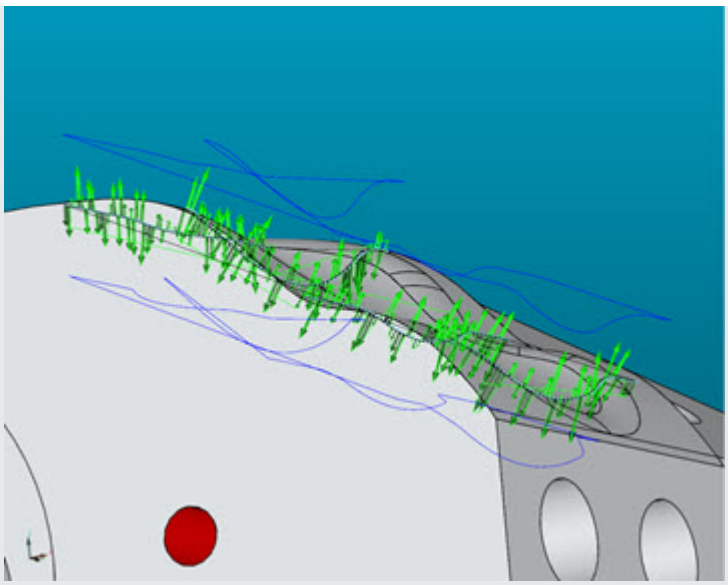
Click on the links below to show correct and incorrect examples of the definitions, results, and reports for a Geometric Tolerance Profile of a Surface:

ASME Y14.5 - 1994 Geometric Tolerance Profile of a Surface Correctly Defined Example

In this example, the Geometric Tolerance Profile of a Surface uses an equivalent upper and lower tolerance value of +/-0.05:



Example of a correctly defined Geometric Tolerance dialog box for a Profile of a Surface using the ASME Y14.5 - 1994 standard.



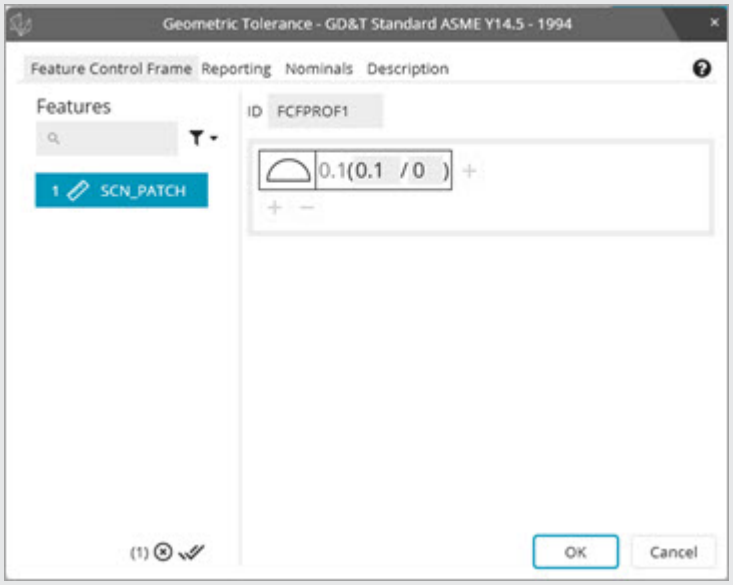
Example of the results after the Geometric Tolerance correctly defined command is executed.

FCFPROF1	MM	0.1	DEFAULT	ASME Y14.5 - 1994				
Feature	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL	
SCN_PATCH	0.000000	0.050000	0.050000	0.071202	0.035601	-0.035601	0.000000	

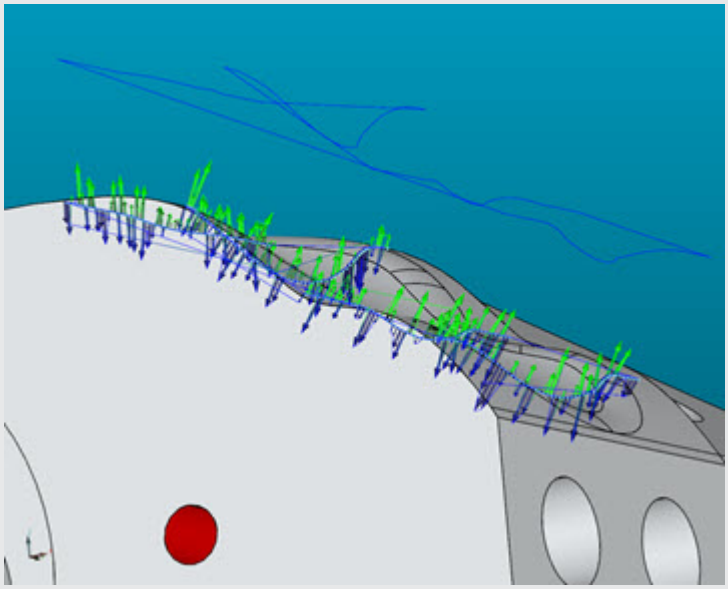
Example of what the Geometric Tolerance version of the report might look like for a correctly defined command.

ASME Y14.5 - 1994 Geometric Tolerance Profile of a Surface Incorrectly Defined Example

In this example, the Geometric Tolerance Profile of a Surface incorrectly uses a single upper tolerance value of 0.1. This results in an unequal tolerance zone causing all negative deviations to be out of tolerance.



Example of an incorrectly defined Geometric Tolerance dialog box for a Profile of a Surface using the ASME Y14.5 - 1994 standard.

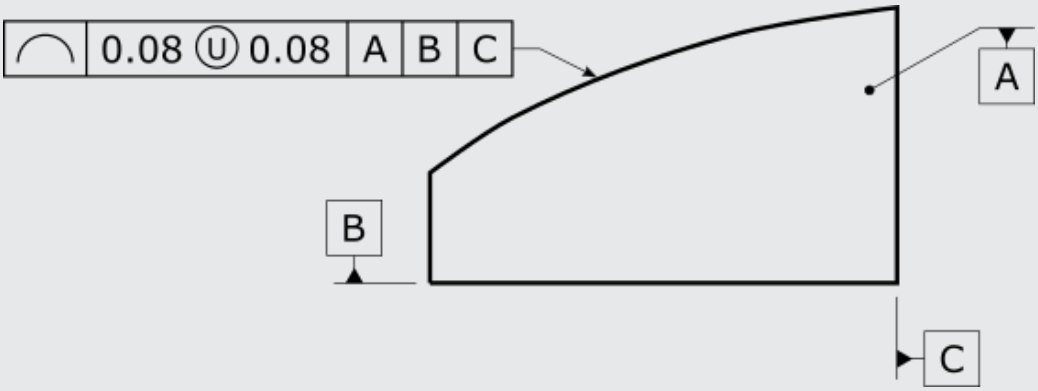


Example of the results after the Geometric Tolerance incorrectly defined command is executed.

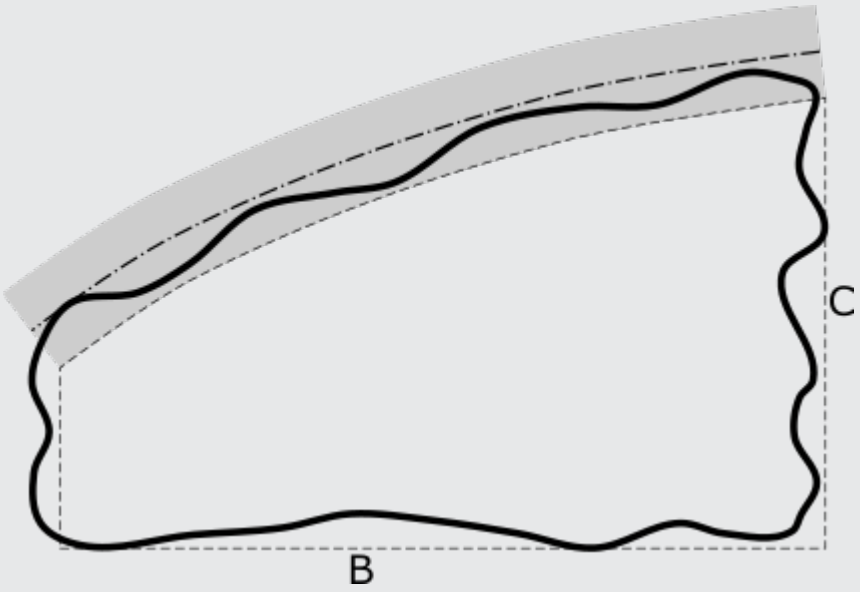
FCFPROF1	MM	0.1			DEFAULT		ASME Y14.5 - 1994
Feature	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL
SCN_PATCH	0.000000	0.100000	0.000000	0.086446	0.036446	-0.036446	0.000000

Example of what the Geometric Tolerance version of the report might look like for an incorrectly defined command.

Suppose you have the Ⓤ modifier specification shown below. The equivalent ISO specification would be 0.08 UZ+0.04.

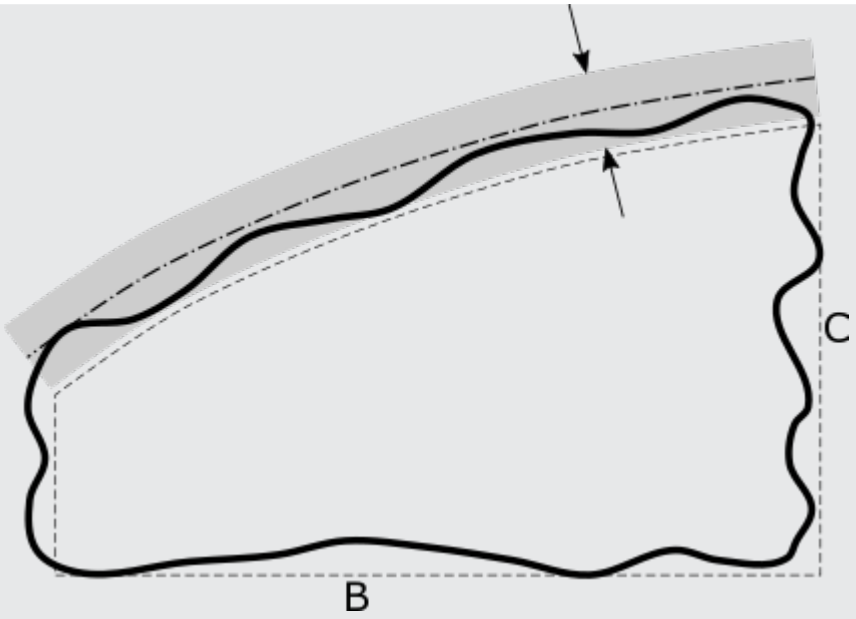


With the above specification, the specified tolerance zone looks like this:



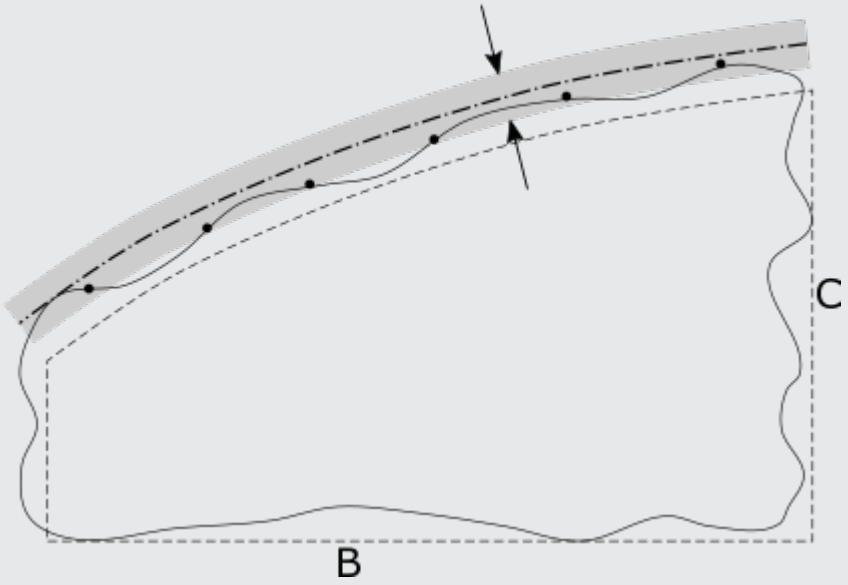
Because this is the specified tolerance zone, the tolerance zone is not minimized, and so it does not represent the actual value. The center of the tolerance zone is offset from the nominal surface and is shown in the dashed-dotted line.

The actual value looks like this:



The center of the tolerance zone stays the same (offset 0.04 from nominal in this case), but the zone is minimized until it just contains the actual surface.

The measured value (with **DEFAULT** datum math) looks like this:



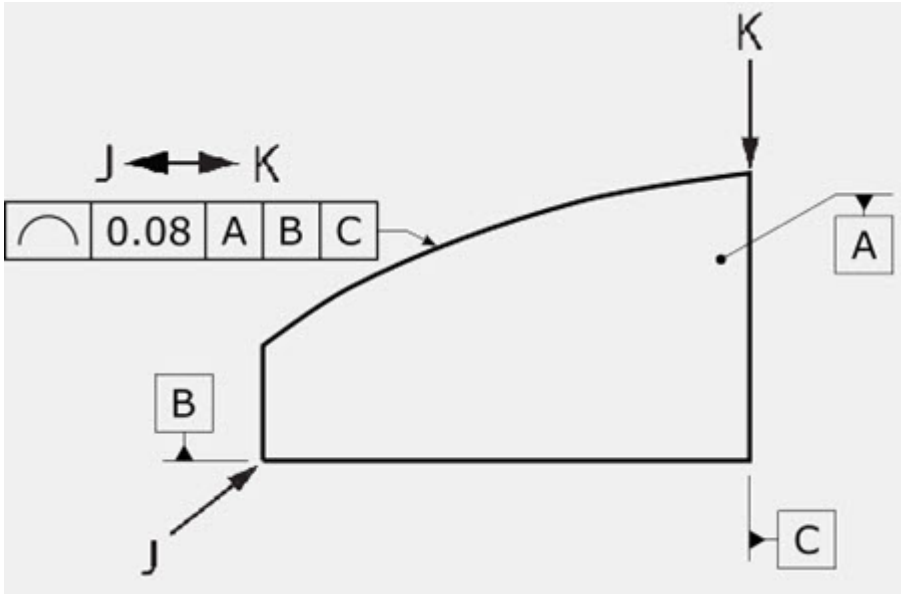
The center of the tolerance zone stays the same (offset 0.04 from nominal in this case) but the zone is minimized around that center until it just contains the measured surface points. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

You may find it helpful to remember that the measured value is equivalent to twice the absolute value of the worst deviation, measured from the center of the tolerance zone.

Profile In Between and Profile All Around

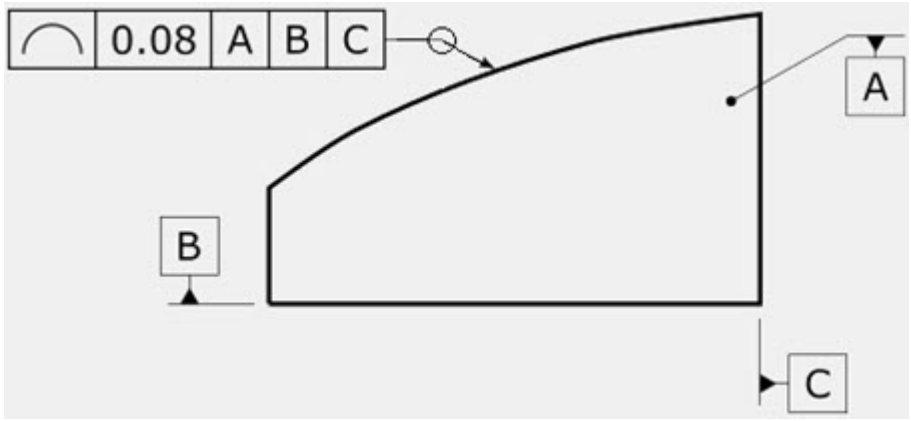
Profile In Between

The **In Between** symbol ⟷ is used between two uppercase letters to identify the extent of the toleranced surface. The surface consist of all points on the surface from the surface designated by the first letter (start) and ending with the surface designated by the second letter (end). This includes all the segments and areas between these letter-designated areas as shown below.



Profile All Around

Profile All Around is applied to the outlines of the cross-sections of a part or to all features represented by a closed outline. It is indicated by the **All Around** circle symbol ○ placed on the intersection of the leader line and the reference line of the tolerance indicator as shown below.



Summary

There is no way to select an **In Between** symbol ⟷ or an **All Around** symbol ○ in the feature control frame builder of the Geometric Tolerance command because they are not part of the feature control frame. Instead, to use the Geometric

Tolerance command to evaluate Profile In Between or Profile All Around, you must employ the correct measurement strategy yourself. To do this, you need to measure multiple cross sections which may need to span multiple surfaces. The easiest way to do this is with the Linear Open Scan command for Profile In Between or the Linear Closed Scan command for Profile All Around.

Another approach is to create multiple scans or a series of auto vector points and combine them into a constructed feature set.

- For Profile In Between, each linear open scan or constructed feature set would represent a single cross section that runs from the point indicated by the start letter to the point indicated by the end letter.
- For Profile All Around, each linear closed scan or constructed feature set would represent a single cross section that runs all around the part or closed outline.

You should evaluate each cross section using a separate profile of a line command and base your conformance decision on the worst value among all cross sections.

Actual Value and Measured Value

Profile tolerance zones have a defined center. They also have a mechanism to grow and shrink the zone around that center until it just envelops the actual surface.

Actual Value:
Each considered feature has its own actual value. This is the size of the smallest tolerance zone that contains the actual surface. The zone is nominally oriented and located to each actual datum, with some exceptions detailed in "How PC-DMIS Solves Datums".

If you have more than one considered feature, and the datum reference frame is not fully constrained, the optimization procedure must simultaneously fit all the features' surfaces in their respective tolerance zones, if possible.

Measured Value:
Each considered feature has its own measured value. This is the size of the smallest tolerance zone that contains the measured surface points. The zone is nominally oriented and located to each measured datum, with some exceptions detailed in "How PC-DMIS Solves Datums".

If you have more than one considered feature, and the datum reference frame is not fully constrained, the PC-DMIS optimization procedure simultaneously fits all features' surface points into their respective tolerance zones. It does this in a proportional way. This guarantees that all toleranced features will fit in their respective tolerance zones if possible.

ASME Y14.5-2009 and ASME Y14.5-2018 use the ASME Y14.5.1-2019 math standard which defines how to calculate an "actual value" of a profile tolerance. Section 8.3.1 (Uniform Tolerance Zones) of the Y14.5.1-2019 standard states, “This concept applies in the same way to equally disposed, unequally disposed and unilateral profile specifications" (see Figures 8-5 thru 8-7). "The actual value of a profile tolerance for a single feature is the value of t_0+2g for the enveloping zone to which the actual surface will conform." This results in the single measured value, equivalent to twice the farthest deviation from the centre of the tolerance zone that PC-DMIS reports.

ASME Y14.5 1994 uses the ASME Y14.5.1M-1994 math standard, which states, "For both unilateral and bilateral profile tolerances two actual values are necessarily calculated: one for surface variations in the positive direction and one for the negative direction. For each direction, the actual value of profile is the smallest intermediate tolerance to which the surface conforms. Note that no single actual value may be calculated for comparison to the tolerance value in the feature control frame, except in the case of unilateral profile tolerances." This means that when you select ASME Y14.5 1994 as your GD&T standard, you no longer get a single measured value, you get the minimum and maximum values instead. The only real difference is the way in which the information is presented, tolerance boundaries and conformance are unaffected. For more information, download the "ProfileReporting_Handout_V2" document from the PC-DMIS Knowledgebase repository.

Validity Rules

All input features (considered and datum) must have the correct specified nominal values and shapes. This ensures that PC-DMIS calculates the measured values correctly, and it ensures that the tolerance command correctly identifies the optimizable degrees of freedom.

Exposed Options

Several types of features expose an ITERATEANDREPIERCE option. These features are points, scans, ellipses, notches, slots, and sets (except for 2D profile vision auto-features and edge point features) when a CAD model is available. When available, PC-DMIS sets the ITERATEANDREPIERCE option to YES by default. It does this to ensure that the center of the tolerance zone is the CAD model surface. When the option is unavailable, or when NO is selected, those feature types create a separate planar tolerance zone for each measured point. The zone is defined by the theoretical point and vector associated with that measured point. This is called the "piecewise planar" approximation, which is excellent in many circumstances. It is poor in these cases:

- If the alignment used to find nominals is significantly different than the optimized datum reference frame

- If the measured data include sharp corners or radii

Due to the sometimes poor behavior of the piecewise-planar approximation, in most cases we recommend that you use a CAD model and keep the ITERATEANDREPIERCE option set to YES. In some circumstances, it makes sense to set this to NO if the computation time is too long. When you set it to NO, it usually improves computation speed, but you are responsible to ensure the piecewise planar approximation is a good approximation.

Lines, circles, and widths do not expose the ITERATEANDREPIERCE option because the geometric tolerance command internally represents the tolerance zones exactly. It is not possible to use the piecewise-planar approximation for those feature types. By contrast, 2D profile vision auto-features, edge point features, and scans made out of edge point features, and "adjust filter" constructed set features do not expose the ITERATEANDREPIERCE options because they always use the piecewise-planar approximation.

When no datums are referenced, the workplane option serves as a datum that defines the plane of the cross section, and defines the optimizable degrees of freedom. It can be set to ZPLUS, ZMINUS, XPLUS, XMINUS, YPLUS, or YMINUS.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data.

For information, see "How PC-DMIS Solves and Uses Datums".

When there are no datum features, the tolerance zone math type controls how the measured surface points are optimized into their respective tolerance zones:

DEFAULT - This does a minimum-zone best fit (also called min-max). This best-fit finds the smallest tolerance zone that contains the surface points. Thus, the **DEFAULT** option produces the smallest measured value for evaluating profile of a line. It is also mathematically very similar to the specification, because if you measure points densely and with high accuracy, the measured value closely approximates the actual value.

LSQ - This does a least-squares best fit. It minimizes the sum of the squares of the deviations to the center of the zone. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly.

Lower Segments of Composite Profile of a Line

A profile of a line tolerance with multiple segments is called a “composite profile of a line”. The first (or upper) segment of a composite profile of a line tolerance is the same as a single segment profile of a line as described above at the beginning of this topic. All lower segments of a composite profile of a line are subtly different. This is because the tolerance zones have unlocked translation compared to the datum

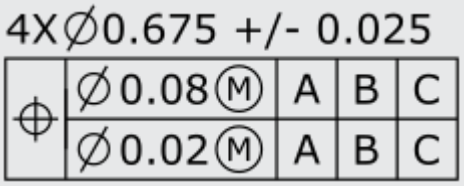
reference frame. However, the tolerance zones remain nominally located and oriented to each other.

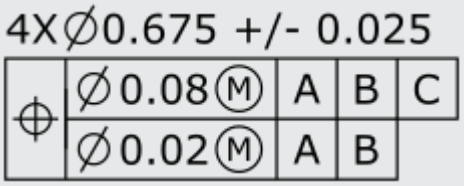
The datum reference frames for the lower segments of a composite profile of a line follow these rules:

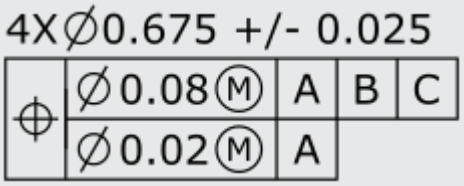
- Each datum reference frame must only use the same datums as the reference frame above it.
- The datums must be in the same order.
- The datums must have the same modifiers.
- A lower segment can have fewer datums than the segment above.

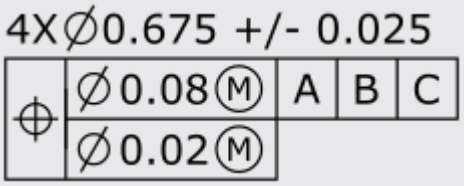
Suppose the upper segment has datums ABC. The lower segment then could reference no datums, datum A, datums AB, or datums ABC. But it could not reference datums BA nor AC nor ABD.

Here are some examples of allowed composite position tolerances:

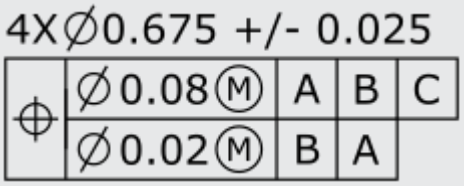


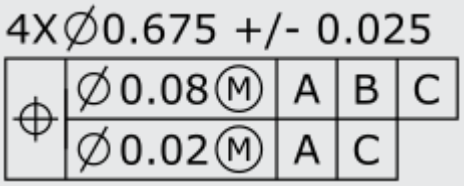


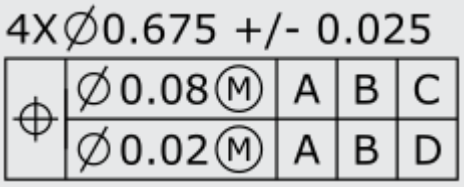




Here are some examples of not-allowed composite position tolerances:

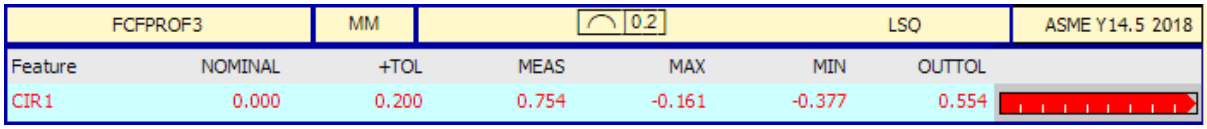






Report

Here is an example report for a profile of a line tolerance of a circle.



FCFPROF3	MM	⌒ 0.2			LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	MEAS	MAX	MIN	OUTTOL
CIR1	0.000	0.200	0.754	-0.161	-0.377	0.554

Profile of a Line Per Unit Tolerances

When the profile of a line tolerance has no datums, a **per unit** check box becomes visible. If you select this check box, profile of a line has two segments. The first (upper) segment is the overall profile of a line as described above. The lower segment is the per-unit profile of a line, which defines a unit length. The per-unit tolerances control the form of every possible unit of the toleranced feature.

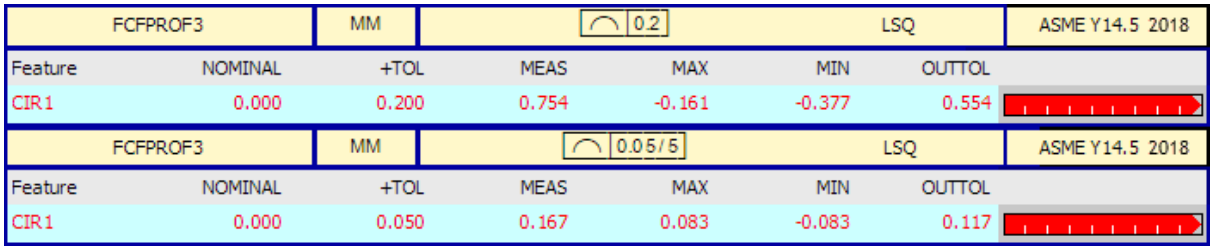
Conceptually, the entire surface cross section is divided into an infinite number of overlapping unit lengths.

Actual Value:
Each of the infinite units has its own actual value. For the entire feature, this is the actual value of the worst unit.

Measured Values:
A very large number of overlapping units contain subsets of the measured points. For any given unit, the measured value is the maximum deviation minus the minimum deviation, where the deviations were calculated with the overall profile of a line. The measured value of the entire feature is the measured value of the worst unit.

Report

Here is an example report for a profile of a line per unit tolerance. The upper label is for the overall profile of a line, and the lower label is for the per unit profile of a line.



FCFPROF3	MM	⌒ 0.2			LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	MEAS	MAX	MIN	OUTTOL
CIR1	0.000	0.200	0.754	-0.161	-0.377	0.554
FCFPROF3	MM	⌒ 0.05/5			LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	MEAS	MAX	MIN	OUTTOL
CIR1	0.000	0.050	0.167	0.083	-0.083	0.117

Profile of a Surface

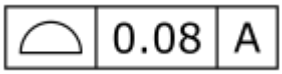
In this Topic

- Profile of a Surface
  - Introduction
  - Allowed Feature Types
  - Tolerance Zones and Allowed Modifiers
  - Profile In Between, Profile All Around, and Profile All Over

Actual Value and Measured Value
Validity Rules
Exposed Options
Lower Segments of Composite Profile of a Surface
Report

Introduction

A profile of a surface specification controls how much the features' surfaces can deviate from being the nominal shapes, which are located and oriented to zero or more datums.



For this geometric tolerance, these three aspects work together:

- Each considered feature's surface data
- Each considered feature's nominal shape and each resulting tolerance zone
- The datum features (if any are referenced)

To evaluate this tolerance, PC-DMIS optimizes each feature's surface data into its respective tolerance zone. The optimization process respects whatever constraints each datum imposes. With multiple considered features, the optimization process simultaneously considers those features. In this way it fits all the toleranced features into their tolerance zones at once.

Allowed Feature Types

You can use these feature types if they have surface data:

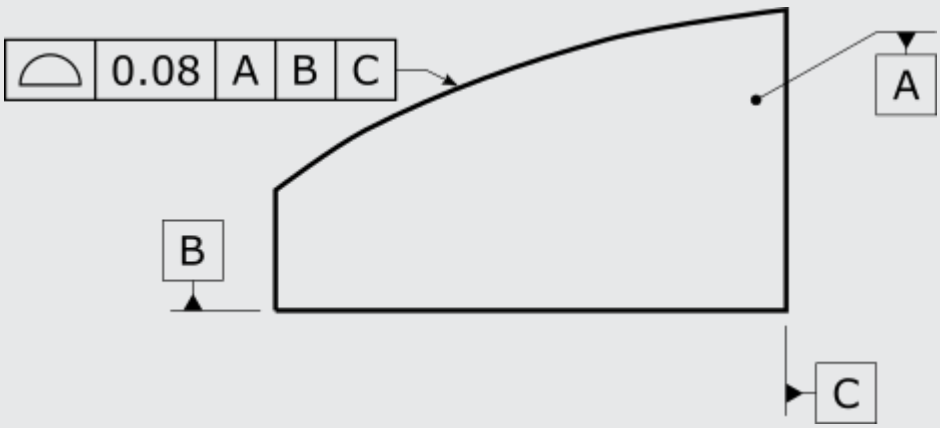
cylinders, spheres, 3D and 1D widths, scans, planes, cones, sets, and tori.

When no datums are referenced, there is no workplane option. Sometimes, the measured data were only measured in a single cross section. This is typically a case where profile of a surface to no datums was specified but the surface is too shallow to measure more than one cross section. In this case, the geometric tolerance command automatically detects the workplane of the cross section. It also uses that workplane as an invisible primary datum to restrict the degrees of freedom to that workplane.

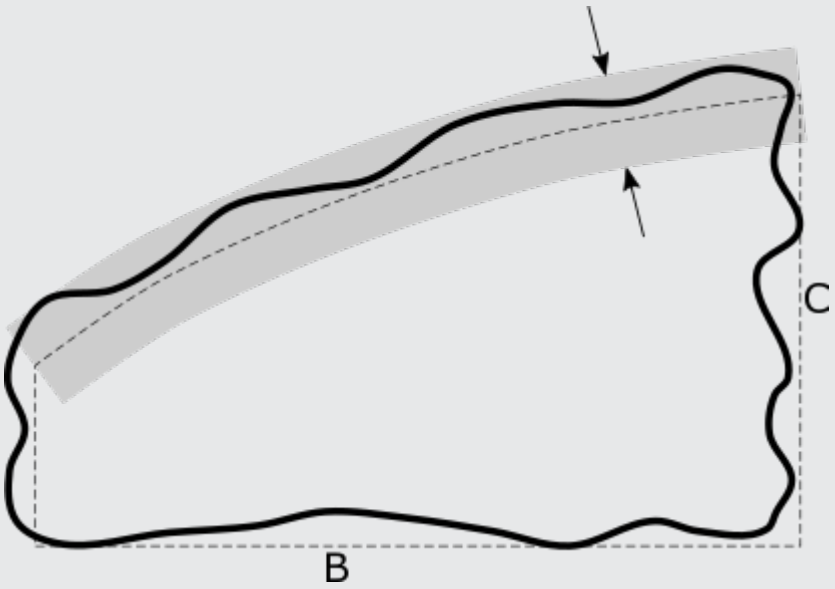
Tolerance Zones and Allowed Modifiers

The tolerance zone is based on the nominal surface of the feature. By default (without any modifiers), the tolerance zone is equal bilateral. This means that half of the tolerance value is on each side of the nominal surface:

Suppose you have this profile of a surface specification:

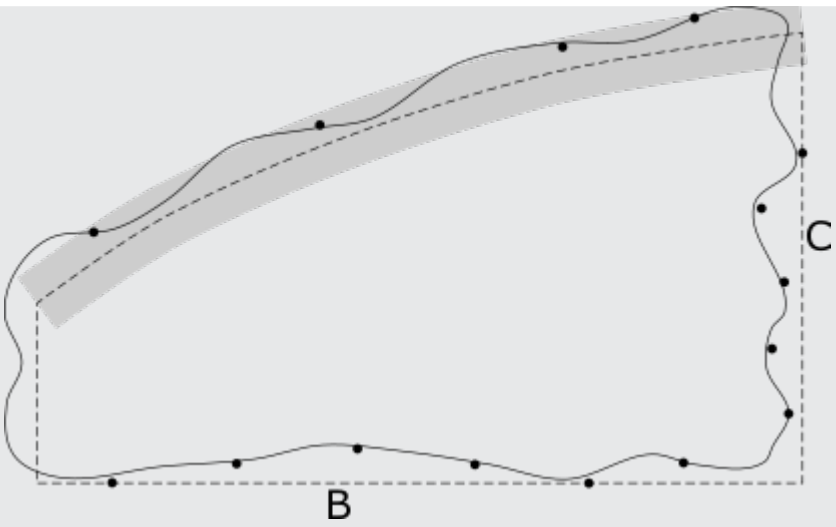


With the above specification, the actual value looks like this:



Since there are no modifiers, the tolerance zone is centered on the nominal surface, which is nominally oriented and located to each actual datum. The solid line indicates the actual surface, the dashed lines represent the nominal surfaces (including the actual datums) and the gray shaded area represents the minimum-size tolerance zone centered on the nominal surface that contains the actual surface.

The measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone center remains the nominal surface, which is nominally oriented and located to each measured datum. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Modifiers may change the nature of the tolerance zone. Starting in ASME Y14.5 2009, PC-DMIS supports the Ⓤ modifier (unequally disposed profile) and, starting in ASME Y14.5 2018, PC-DMIS supports the △ modifier (dynamic profile). Under ISO 1101, PC-DMIS supports the UZ modifier (specified tolerance zone offset) and the OZ modifier (unspecified linear tolerance zone offset). While they are not equivalent, the Ⓤ and UZ modifiers have similar functionality. They offset the center of the tolerance zone from the nominal surface. Likewise, the △ and OZ modifiers have similar functionality. They allow the center of the tolerance zone to progress in the plus-material or minus-material direction. These modifiers did not exist in ASME Y14.5 1994 but it did support the concept of an unequally disposed tolerance zone. This was usually specified graphically on the print by dotted lines showing the extent of tolerance into and out of the material. Therefore, when selecting ASME Y14.5 1994 as the GDT standard for your PC-DMIS measurement routine, you must enter both the upper and lower tolerances when creating the feature control frame. PC-DMIS will then report the maximum and minimum deviations which are compared against the corresponding upper and lower tolerance values to assess conformance.

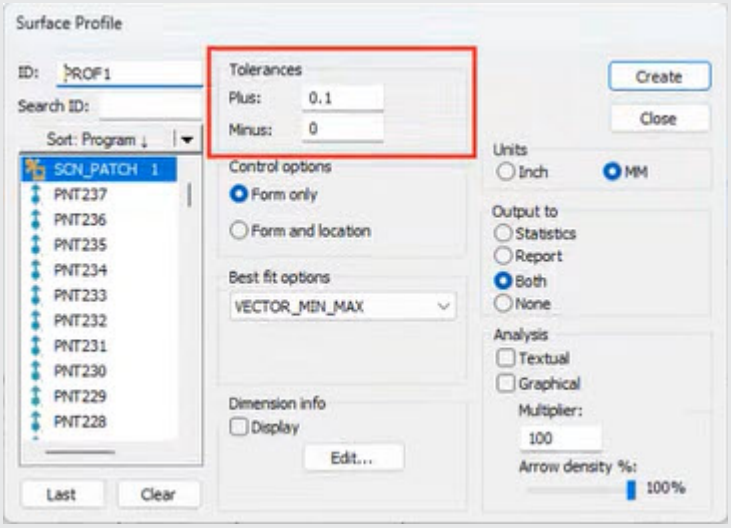
Legacy profile includes a “form only” option which only allows a single, positive tolerance to be entered. To achieve the same behavior using an ASME Y14.5 1994 geometric tolerance profile command, you must divide the total allowable tolerance in half and enter it as equal bilateral values.

The examples below show the differences between the Legacy Surface Profile and the Geometric Tolerance Profile of a Surface commands. Although the examples are for the Surface Profile, the rules are identical for the Line Profile.

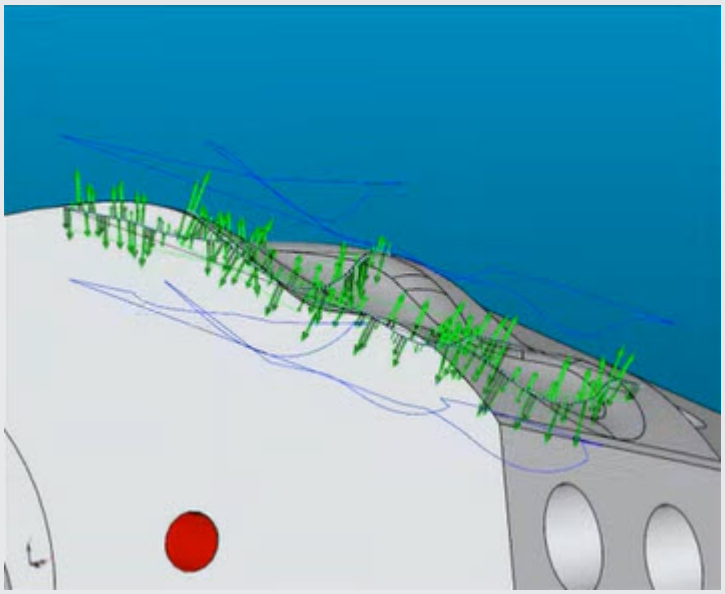
Click on the link below to show examples of the definition, result, and report for a Legacy Surface Profile:

Legacy Surface Profile, Form Only Example

The following example shows a Surface Profile dialog box with a Form only definition:



Example of a legacy Surface Profile dialog box defined for form only with an upper tolerance of 0.1 and a lower tolerance of 0.



Example of the results after the legacy command is executed.

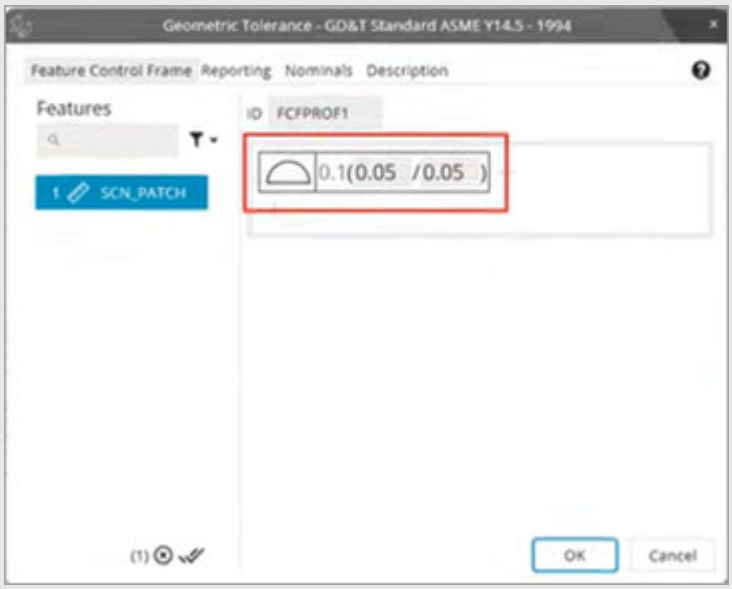
	MM	PROF1 - SCN_PATCH FORMONLY VECTOR_MIN_MAX					
AX	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL
M	0.000000	0.100000	0.000000	0.071202	0.035601	-0.035601	0.000000

Example of what the legacy version of the report might look like.

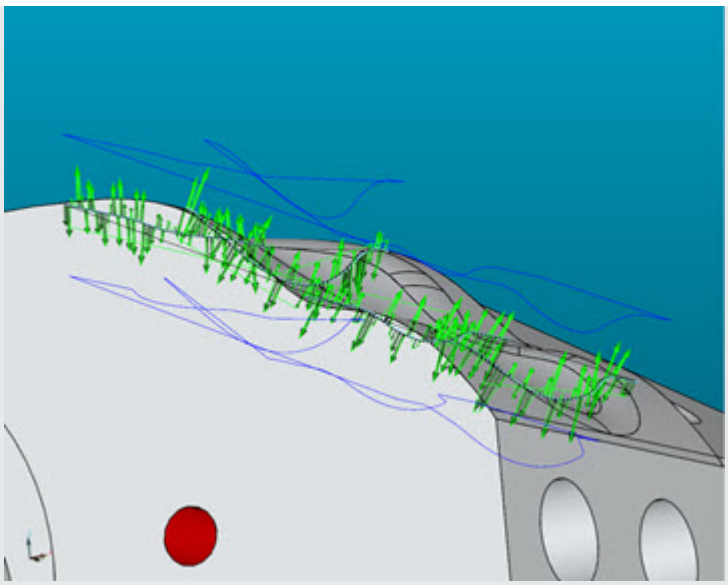
Click on the links below to show correct and incorrect examples of the definitions, results, and reports for a Geometric Tolerance Profile of a Surface:

ASME Y14.5 - 1994 Geometric Tolerance Profile of a Surface Correctly Defined Example

In this example, the Geometric Tolerance Profile of a Surface uses an equivalent upper and lower tolerance value of +/-0.05:



Example of a correctly defined Geometric Tolerance dialog box for a Profile of a Surface using the ASME Y14.5 - 1994 standard.



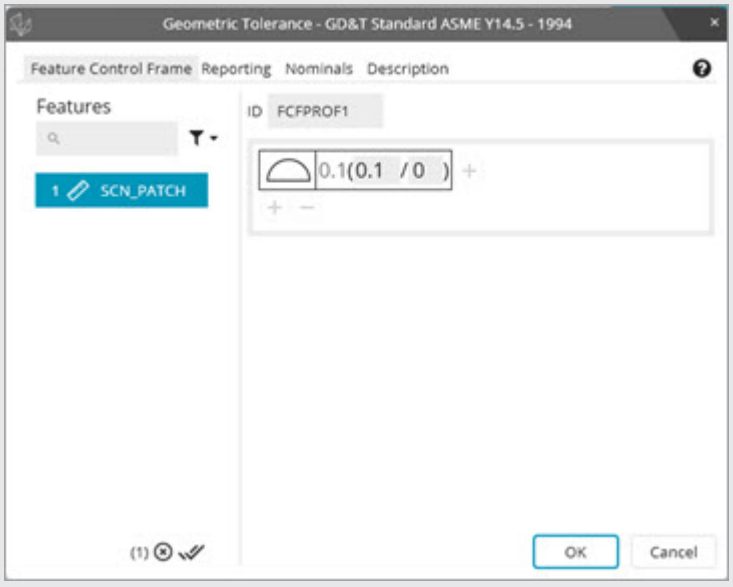
Example of the results after the Geometric Tolerance correctly defined command is executed.

FCFPROF1	MM	0.1	DEFAULT	ASME Y14.5 - 1994				
Feature	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL	
SCN_PATCH	0.000000	0.050000	0.050000	0.071202	0.035601	-0.035601	0.000000	

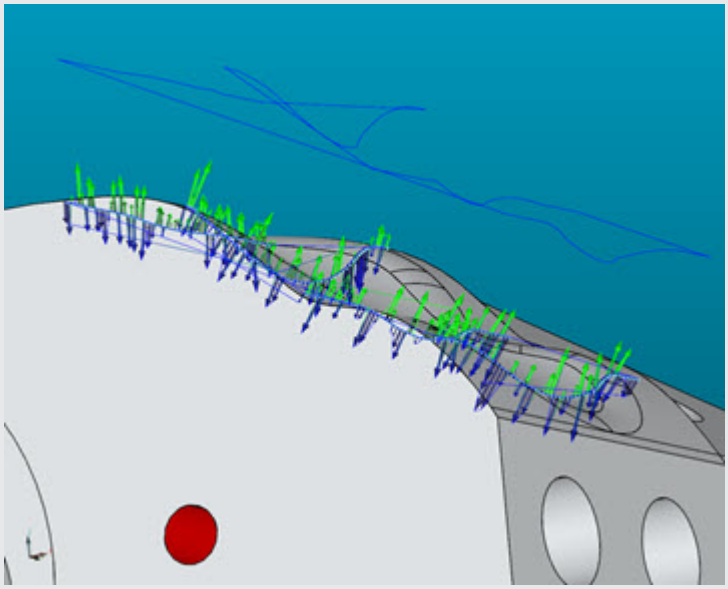
Example of what the Geometric Tolerance version of the report might look like for a correctly defined command.

ASME Y14.5 - 1994 Geometric Tolerance Profile of a Surface Incorrectly Defined Example

In this example, the Geometric Tolerance Profile of a Surface incorrectly uses a single upper tolerance value of 0.1. This results in an unequal tolerance zone causing all negative deviations to be out of tolerance.



Example of an incorrectly defined Geometric Tolerance dialog box for a Profile of a Surface using the ASME Y14.5 - 1994 standard.



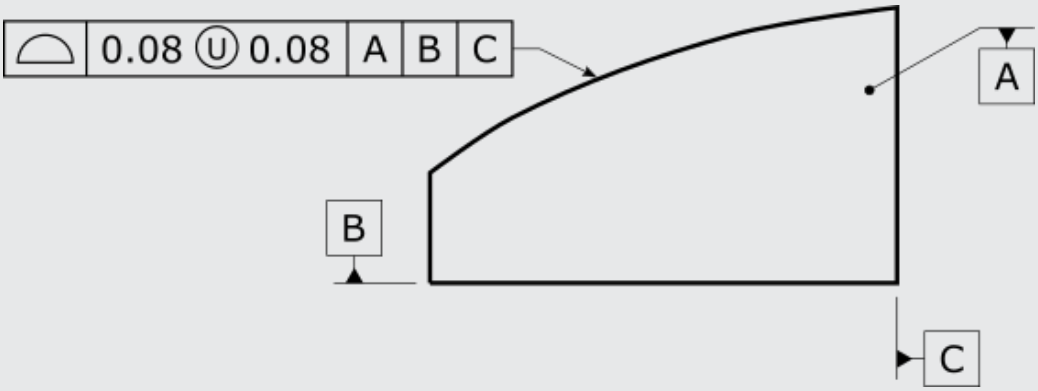
Example of the results after the Geometric Tolerance incorrectly defined command is executed.

FCFPROF1	MM	0.1	DEFAULT	ASME Y14.5 - 1994

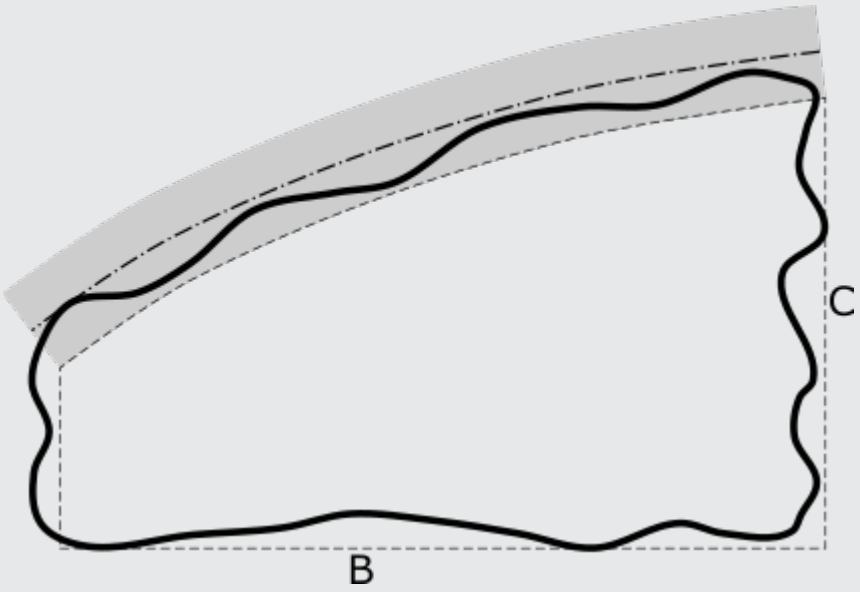
Feature	NOMINAL	+TOL	-TOL	MEAS	MAX	MIN	OUTTOL
SCN_PATCH	0.000000	0.100000	0.000000	0.086446	0.036446	-0.036446	0.000000

Example of what the Geometric Tolerance version of the report might look like for an incorrectly defined command.

Suppose you have the Ⓤ modifier specification shown below. The equivalent ISO specification would be 0.08 UZ+0.04.

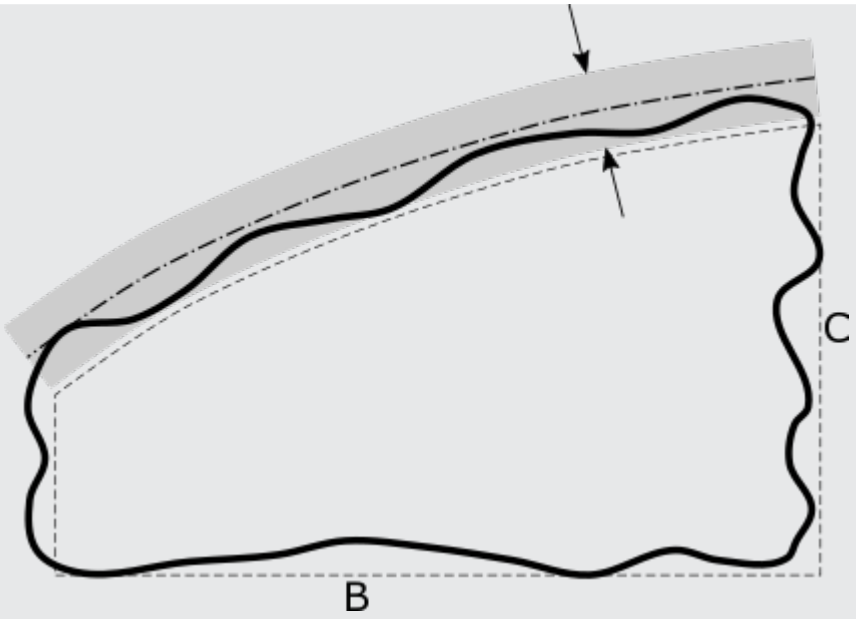


With the above specification, the specified tolerance zone looks like this:



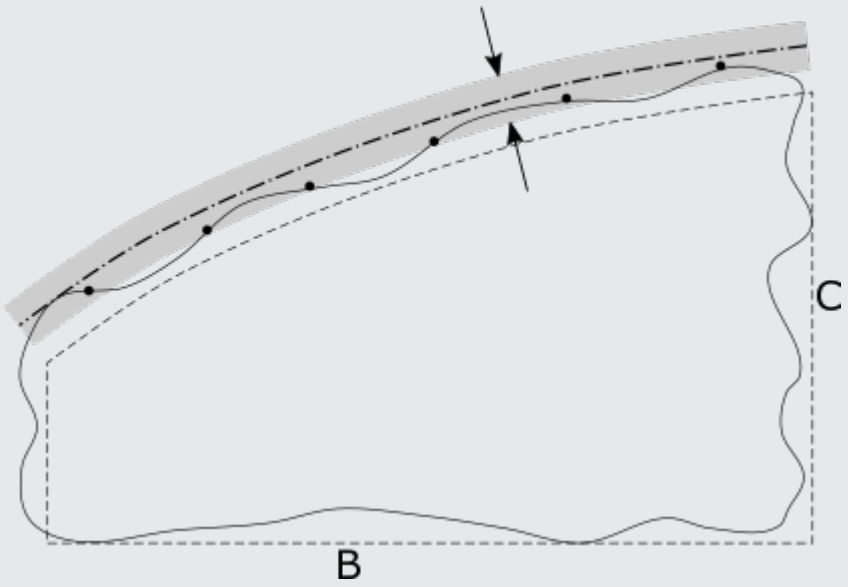
Because this is the specified tolerance zone, the tolerance zone is not minimized, and so it does not represent the actual value. The center of the tolerance zone is offset from the nominal surface and is shown in the dashed-dotted line.

The actual value looks like this:



The center of the tolerance zone stays the same (offset 0.04 from nominal in this case), but the zone is minimized until it just contains the actual surface.

The measured value (with **DEFAULT** datum math) looks like this:



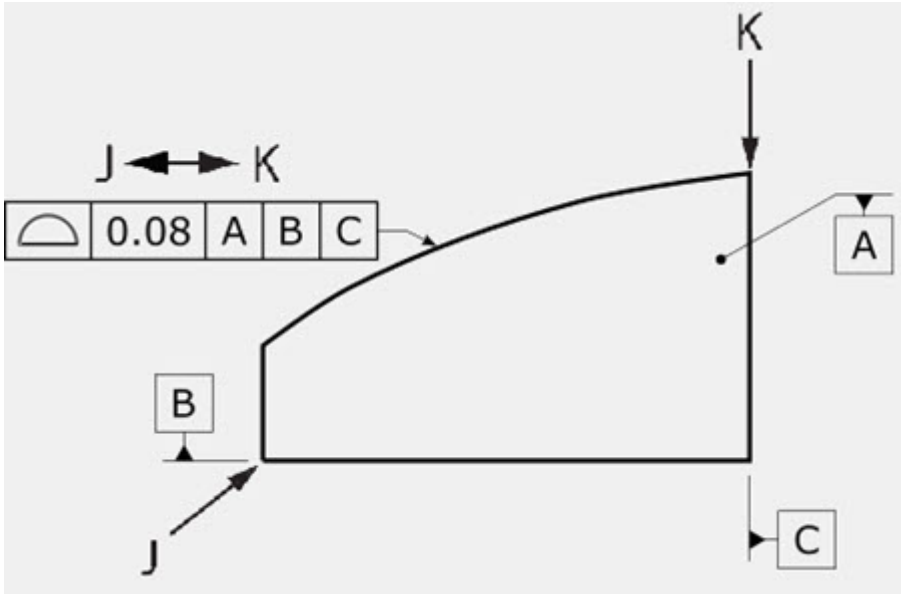
The center of the tolerance zone stays the same (offset 0.04 from nominal in this case) but the zone is minimized around that center until it just contains the measured surface points. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

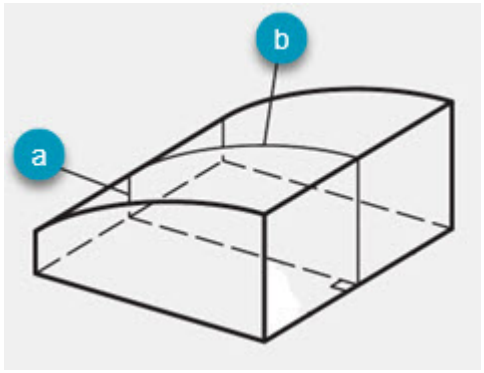
You may find it helpful to remember that the measured value is equivalent to twice the absolute value of the worst deviation, measured from the center of the tolerance zone.

Profile In Between, Profile All Around, and Profile All Over

Profile In Between

The **In Between** symbol ⟷ is used between two uppercase letters to identify the extent of the toleranced surface. The surface consist of all points on the surface from the surface designated by the first letter (start) and ending with the surface designated by the second letter (end). This includes all the segments and areas between these letter-designated areas as shown below.

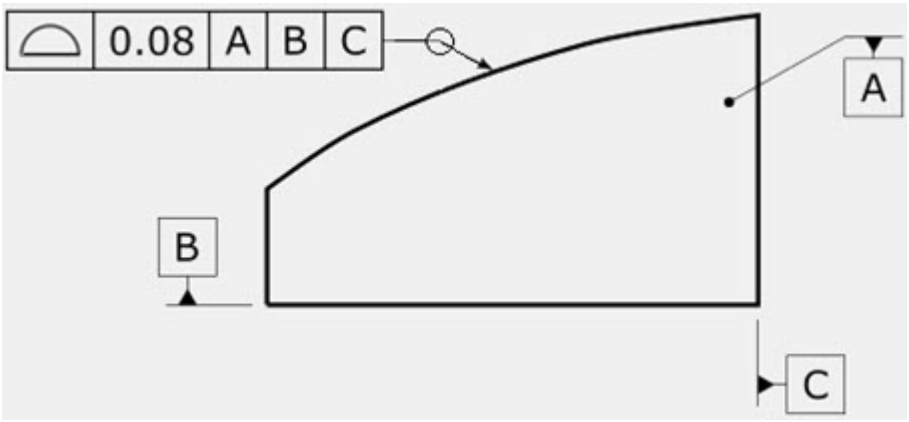


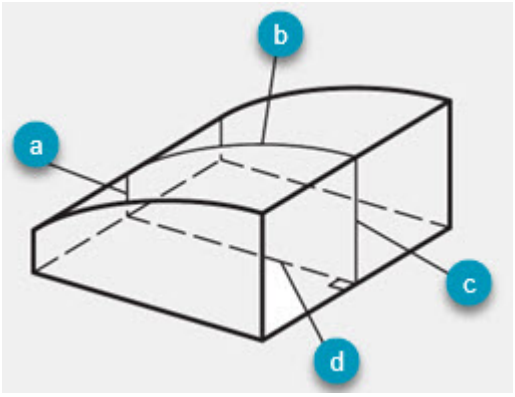


Profile In Between applies only to the surfaces indicated by the collection plane or view orientation, between the designated Start and End points.

Profile All Around

Profile All Around is applied to the outlines of the cross-sections of a part or to all features represented by a closed outline. It is indicated by the **All Around** circle symbol ○ placed on the intersection of the leader line and the reference line of the tolerance indicator as shown below.

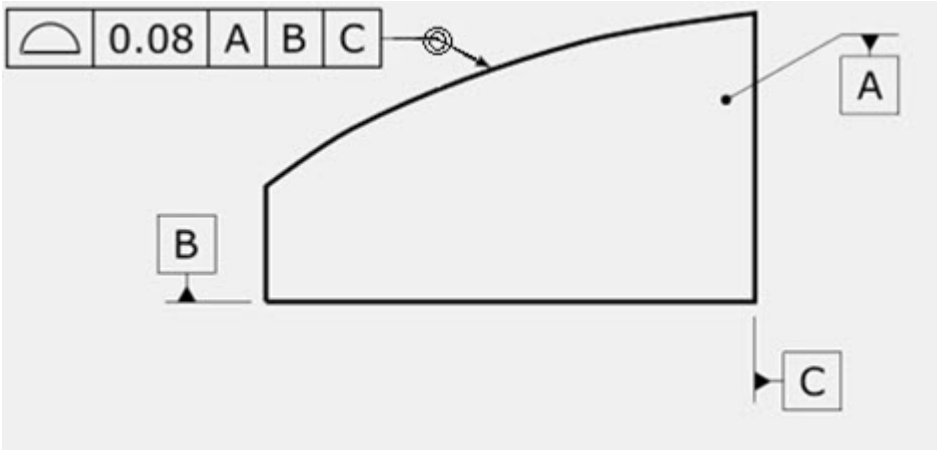


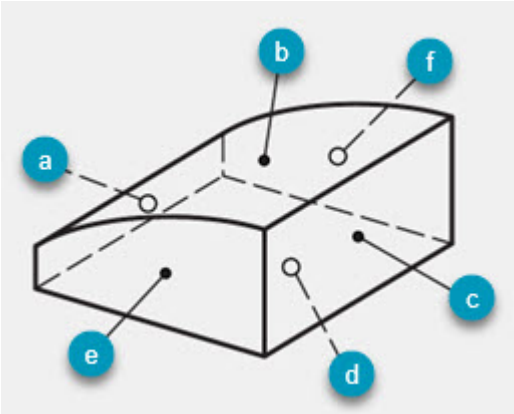


Profile All Around only applies to the surfaces identified by the collection plane or view orientation, not to the entire workpiece.

Profile All Over

Profile All Over is applied to all integral features of a workpiece and is indicated by a two concentric circle symbol ◎ placed on the intersection of the leader line and the reference line of the tolerance indicator as shown below.





Profile All Over applies to all surfaces of the entire workpiece and is three-dimensional.

Summary

There is no way to select a Profile In Between, Profile All Around, or Profile All Over symbol in the feature control frame builder of the Geometric Tolerance command because they are not part of the feature control frame. Instead, to use the Geometric Tolerance command to evaluate Profile In Between, Profile All Around, or Profile All Over, you must employ the correct measurement strategy yourself. You need to measure all the applicable surfaces such that they are treated as a single feature. The easiest way to do this is with a series of Linear Open Scan commands for Profile In Between or a series of Linear Closed Scan commands for Profile All Around or Profile All Over.

Another approach is to create a series of auto features (one for each individual surface). All the scans or auto features that pertain to the surfaces under evaluation must then be combined into a single constructed feature set.

- For Profile In Between, the constructed feature set would represent the entire extent of all surfaces that run from the point indicated by the start letter to the point indicated by the end letter.
- For Profile All Around, the constructed feature set would represent the entire extent of all surfaces that run all around the part or closed outline.
- For Profile All Over, the constructed feature set would represent the entire extent of all outer surfaces of the entire part.

You should use a single profile of a surface command that references the constructed feature set as the considered feature to evaluate any of these requirements.

Actual Value and Measured Value

Profile tolerance zones have a defined center. They also have a mechanism to grow and shrink the zone around that center until it just envelops the actual surface.

Actual Value:
Each considered feature has its own actual value. This is the size of the smallest tolerance zone that contains the actual surface. The zone is nominally oriented and located to each actual datum, with some exceptions detailed in "How PC-DMIS Solves Datums".

If you have more than one considered feature, and the datum reference frame is not fully constrained, the optimization procedure must simultaneously fit all the features' surfaces in their respective tolerance zones, if possible.

Measured Value:
Each considered feature has its own measured value. This is the size of the smallest tolerance zone that contains the measured surface points. The zone is nominally oriented and located to each measured datum, with some exceptions detailed in "How PC-DMIS Solves Datums".

If you have more than one considered feature, and the datum reference frame is not fully constrained, the PC-DMIS optimization procedure simultaneously fits all features' surface points into their respective tolerance zones. It does this in a proportional way. This guarantees that all toleranced features will fit in their respective tolerance zones if possible.

ASME Y14.5-2009 and ASME Y14.5-2018 use the ASME Y14.5.1-2019 math standard which defines how to calculate an "actual value" of a profile tolerance. Section 8.3.1 (Uniform Tolerance Zones) of the Y14.5.1-2019 standard states, “This concept applies in the same way to equally disposed, unequally disposed and unilateral profile specifications" (see Figures 8-5 thru 8-7). "The actual value of a profile tolerance for a single feature is the value of t_0+2g for the enveloping zone to which the actual surface will conform." This results in the single measured value, equivalent to twice the farthest deviation from the centre of the tolerance zone that PC-DMIS reports.

ASME Y14.5 1994 uses the ASME Y14.5.1M-1994 math standard, which states, "For both unilateral and bilateral profile tolerances two actual values are necessarily calculated: one for surface variations in the positive direction and one for the negative direction. For each direction, the actual value of profile is the smallest intermediate tolerance to which the surface conforms. Note that no single actual value may be calculated for comparison to the tolerance value in the feature control frame, except in the case of unilateral profile tolerances." This means that when you select ASME Y14.5 1994 as your GD&T standard, you no longer get a single measured value, you get the minimum and maximum values instead. The only real difference is the way in which the information is presented, tolerance boundaries and conformance are unaffected. For more information, download the "ProfileReporting_Handout_V2" document from the PC-DMIS Knowledgebase repository.

Validity Rules

All input features (considered and datum) must have the correct specified nominal values and shapes. This ensures that PC-DMIS calculates the measured values correctly, and it ensures that the tolerance command correctly identifies the optimizable degrees of freedom.

Exposed Options

Several types of features expose an ITERATEANDREPIERCE option. These are points, scans, sets, and torii (except for 2D profile vision auto-features and edge point features) when a CAD model is available. When available, the option is set to YES by default, because that ensures the center of the tolerance zone is the CAD model surface. When the option is unavailable, or when NO is selected, those feature types create a separate planar tolerance zone for each measured point, defined by theoretical point and vector associated with that measured point. This is called the "piecewise planar" approximation, which is excellent in many circumstances. It is poor in these cases:

- If the alignment used to find nominals is significantly different than the optimized datum reference frame
- If the measured data include sharp corners or radii

Due to the sometimes-poor behavior of the piecewise-planar approximation, in most cases we recommend that you use a CAD model and keep the ITERATEANDREPIERCE option set to YES. In some circumstances, it makes sense to set it to NO if the computation time is too long. When you set it to NO in this way, it usually improves computation speed, but you are responsible to ensure that the piecewise planar approximation is a good approximation.

Cylinders, spheres, widths, planes, and cones do not expose the ITERATEANDREPIERCE option because the geometric tolerance command internally represents the tolerance zones exactly. It is not possible to use the piecewise-planar approximation for those feature types. By contrast, 2D profile vision auto-features, edge point features, scans made out of edge point features, and "adjust filter" constructed set features do not expose the ITERATEANDREPIERCE options because they always use the piecewise-planar approximation.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data.

For information, see "How PC-DMIS Solves and Uses Datums".

When there are no datum features, the tolerance zone math type controls how the measured surface points are optimized into their respective tolerance zones:

DEFAULT - This does a minimum-zone best fit (also called min-max). This best-fit finds the smallest tolerance zone that contains the surface points. Thus, the **DEFAULT** option produces the smallest measured value for evaluating profile of a

surface. It is also mathematically very similar to the specification, because if you measure points densely and with high accuracy, the measured value closely approximates the actual value.

LSQ - This does a least-squares best fit. It minimizes the sum of the squares of the deviations to the center of the zone. This option produces a larger measured value (it is more conservative than the **DEFAULT** option). But in general, this option computes more quickly.

Lower Segments of Composite Profile of a Surface

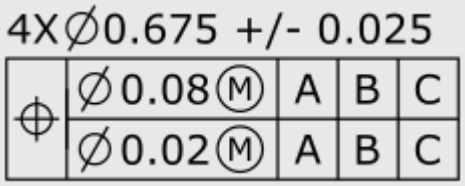
A profile of a surface tolerance with multiple segments is called a “composite profile of a surface”. The first (or upper) segment of a composite profile of a surface tolerance is the same as a single segment profile of a surface as described above at the beginning of this topic. All lower segments of a composite profile of a surface are subtly different. This is because the tolerance zones have unlocked translation compared to the datum reference frame. However, the tolerance zones remain nominally located and oriented to each other.

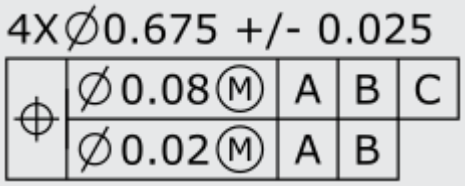
The datum reference frames for the lower segments of a composite profile of a surface follow these rules:

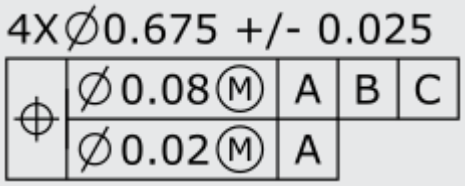
- Each datum reference frame must only use the same datums as the reference frame above it.
- The datums must be in the same order.
- The datums must have the same modifiers.
- A lower segment can have fewer datums than the segment above.

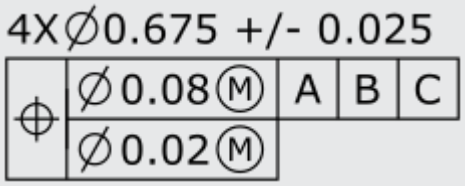
Suppose the upper segment has datums ABC. The lower segment then could reference no datums, datum A, datums AB, or datums ABC. But it could not reference datums BA nor AC nor ABD.

Here are some examples of allowed composite position tolerances:

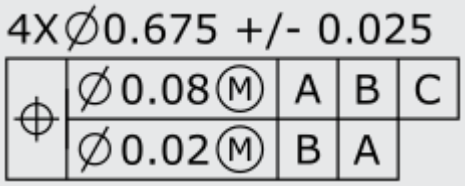


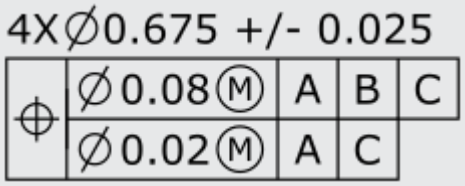


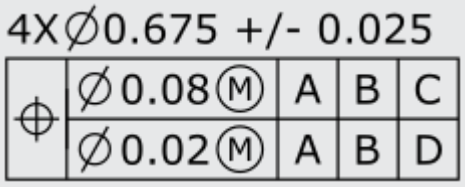




Here are some examples of not-allowed composite position tolerances:







Report

Here is an example report for a profile of a surface tolerance of a plane.

FCFPROF2	MM	⌓ 1 A B C	LSQ	ASME Y14.5 2018

Feature	NOMINAL	+TOL	MEAS	MAX	MIN	OUTTOL	
PLN17	0.000000	1.000000	1.122074	0.561037	-0.147759	0.122074	

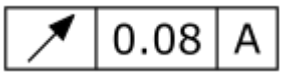
Circular Runout

In this Topic

- Circular Runout
  - Introduction
  - Allowed Feature Types
  - Circular Features
  - Cylindrical Features
  - Conical Features
  - Planar Features
  - Validity Rules
  - Allowed Modifiers
  - Exposed Options
  - Report

Introduction

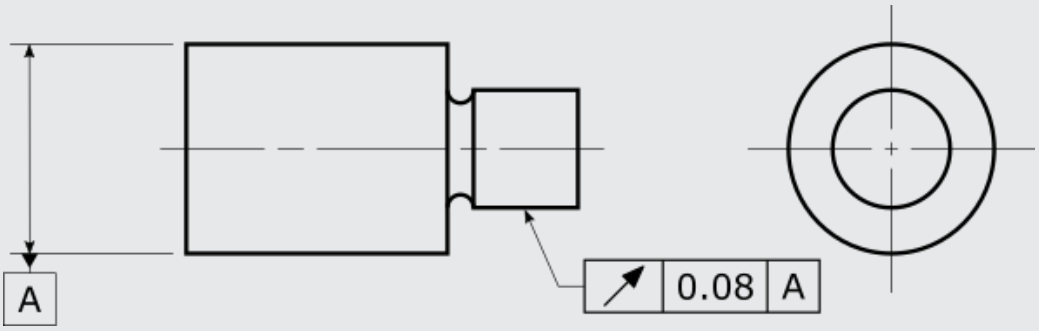
A circular runout specification controls how much the feature's surface cross sections can deviate from being perfect circles centered on some datum axis.



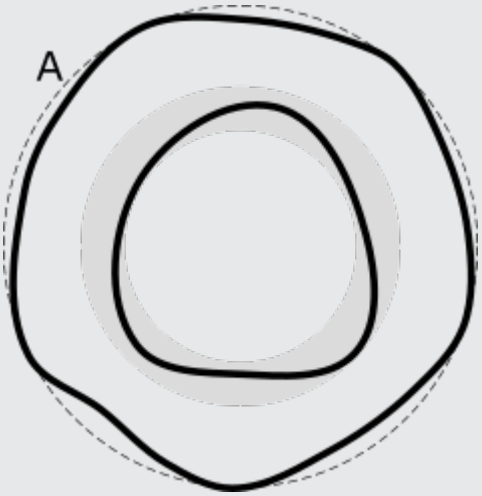
Actual Value:
For a cross section, this is the minimum distance between two circles. These circles are centered on and oriented perpendicular to the datum axis. They contain the entire cross section between them.

For a whole feature, this is the worst actual value of all the cross sections.

Suppose you had this circular runout specification:

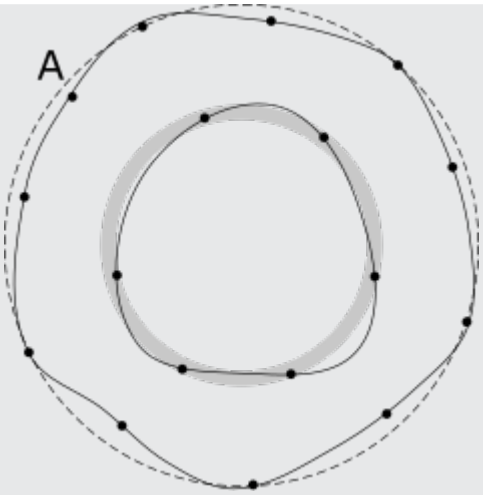


With the above specification, the actual value for one of the cross sections looks like this:



The actual part surface uses the solid line, the actual datum uses the dashed line, and the smallest tolerance zone containing the cross section's surface is shown in the shaded area. The tolerance zone is exactly coaxial to the actual datum axis. The actual value for the whole feature is the worst actual value of all the cross sections.

Finally, the measured value (with DEFAULT datum math) for one of the cross sections looks like this:



The measured tolerance zone is exactly coaxial to the measured datum axis. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value. The measured value for the whole feature would be the worst measured value of all the cross sections.

Allowed Feature Types

You can use circular, cylindrical, conical, or planar features that have surface data. For details on the circles, cylinders, cones, and planes that have surface data, refer to "Feature Types With and Without Surface Data". These features must be nominally concentric with the datum axis.

Circular Features

This geometric tolerance interprets circular features as a single cross section.

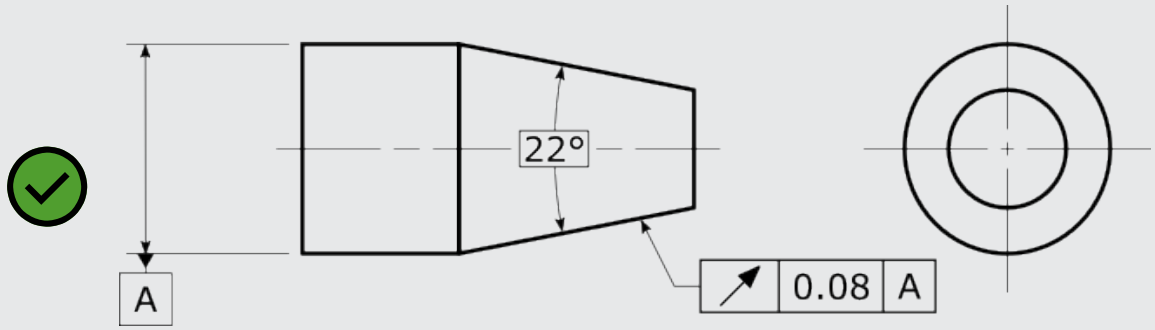
Measured Value:
This is the distance between two circles. The circles contain all the measured points between them. The circles are centered on and perpendicular to the datum axis.

Usually, the two circles are coplanar. This means the surface has zero conical angle. However, the "cone half angle" option in the command lets you specify a conical surface. In that case, PC-DMIS angles the two circles so that the tolerance zone is perpendicular to the nominal surface. The cone half angle represents the angle of the nominal surface, not the tolerance zone.

- For outer circles, positive angles mean the circle vector points towards the cone vertex, and negative angles mean the circle vector points away from the cone vertex.
- For inner circles, positive angles mean the circle vector points away from the cone vertex, and negative angles mean the circle vector points towards the cone vertex.

These inner and outer conventions were chosen to represent typical defaults. Most of the time, outer circle vectors point toward the cone vertex—the positive angle case. Also, most of the time, inner circle vectors point away from the cone vertex—the positive angle case.

Suppose you have the following specification, and you decide to measure the cone as a series of circles (even though we recommend using a cone feature):



In that case, the circles are outer circles, which means if the circle's vector points to the right (towards the cone vertex), the cone half angle should be set to +11°. If the circle's vector points to the left (away from the cone vertex), the cone half angle should be set to -11°.

Cylindrical Features

This geometric tolerance divides up the data from a cylindrical feature into multiple cross sections. The tolerance evaluates the runout on each cross section. To maximize your chances of finding the worst actual cross section, we recommend that you measure the cylinder with many cross sections.

Measured Value:
Across the entire feature, this is the measured value of the worst cross section. If you didn't measure the measured data in cross sections, PC-DMIS gives you an error.

Conical Features

This geometric tolerance divides up the data from a conical feature into multiple cross sections. The tolerance evaluates the runout on each cross section. It orients each tolerance zone perpendicular to the nominal surface. To maximize your chances of finding the worst actual cross section, we recommend that you measure the cone with many cross sections.

Measured Value:
Across the entire feature, this is the measured value of the worst cross section. If you didn't measure the measured data in cross sections, PC-DMIS gives you an error.

Planar Features

This geometric tolerance divides up the data from a planar feature into one or more circular sections around the datum axis. The tolerance orients each tolerance zone perpendicular to the nominal surface. To maximize your chances of finding the worst actual circular section, we recommend that you measure the plane with many circular sections.

Measured Value:
Across the entire feature, this is the measured value of the worst circular section. If you didn't measure the measured data in circular sections that surround the datum axis, PC-DMIS gives you an error.

Validity Rules

The datum reference frame must establish a clear datum axis.

Allowed Modifiers

None. This geometric tolerance does not allow modifiers.

Exposed Options

For circular features, the "cone half angle" option allows the circle to represent a cross section of a conical surface, instead of a cylindrical surface. This adjusts the orientation of the tolerance zone. Both positive and negative cone half angles make sense, which allows you to control whether the nominal cone opening direction is parallel or antiparallel to the nominal circle's vector.

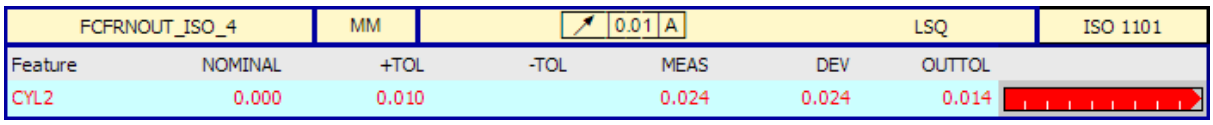
For inner circles with positive cone half angles, the circle vector points from smaller cone diameters to larger cone diameters. Negative cone half angles are the reverse. This convention was chosen because it makes positive angles be the most common case for most users.

For outer circles with positive cone half angles, the circle vector points from larger cone diameters to smaller cone diameters. Negative cone half angles are the reverse. This convention was chosen because it makes positive angles be the most common case for most users.

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for a circular runout tolerance of a cylinder.



FCFRNOUT_ISO_4	MM	0.01 A	LSQ	ISO 1101

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	0.000	0.010		0.024	0.024	0.014

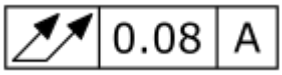
Total Runout

In this Topic

- Total Runout
  - Introduction
  - Allowed Feature Types
  - Cylindrical Features
  - Conical Features
  - Planar Features
  - Validity Rules
  - Allowed Modifiers
  - Exposed Options
  - Report

Introduction

A total runout specification controls how much the feature can deviate from having perfect form centered on some datum axis.



Allowed Feature Types

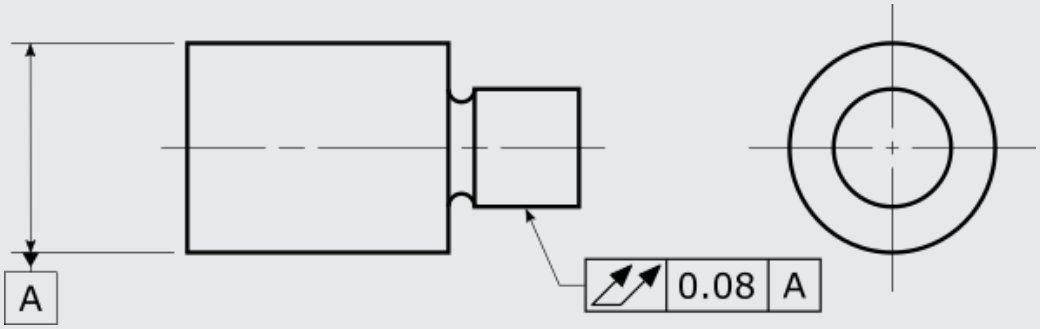
You can use cylindrical, conical, or planar features that have surface data. For details on the cylinders, cones, and planes that have surface data, refer to "Feature Types With and Without Surface Data". They must be nominally concentric with the datum axis.

Cylindrical Features

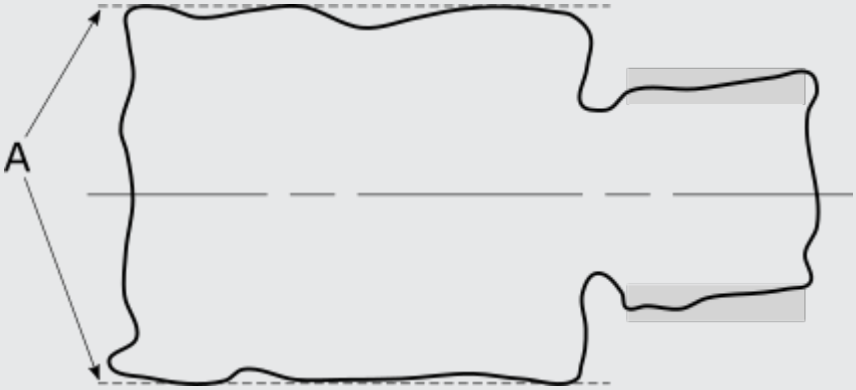
Actual Value:
This is the minimum distance between two concentric cylinders that contain the actual surface between them. The cylinders are concentric with the actual datum axis.

Measured Value:
This is the minimum distance between two concentric cylinders that contain all the measured points between them. The cylinders are concentric with the measured datum axis.

Suppose you had this total runout specification:

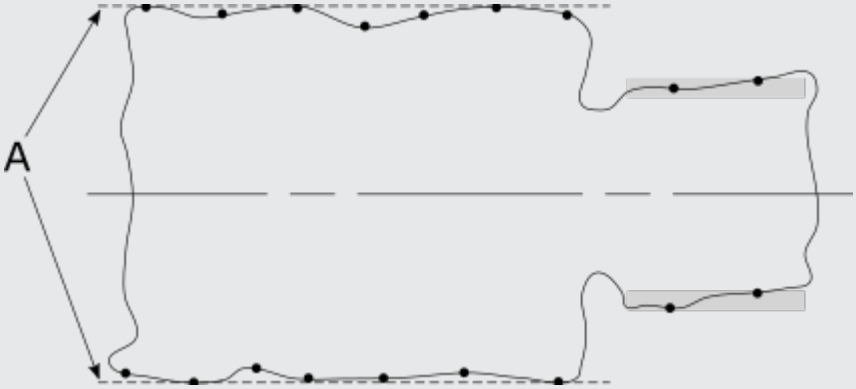


With the above specification, the actual value look like this:



The actual part surface uses the solid line, the actual datum uses the dashed line, and the smallest tolerance zone containing the actual feature surface is shown in the shaded area. The tolerance zone is exactly coaxial to the actual datum axis.

Finally, the measured value (with **DEFAULT** datum math) looks like this:



The measured tolerance zone is exactly coaxial to the measured datum axis. In this case, the measured points were not measured densely enough, and so the measured value is smaller than the actual value.

Conical Features

Actual Value:
This is the minimum distance between two concentric cones that contain the entire actual surface between them. The cones are concentric with the actual datum axis. They also have the same cone angle as the nominal cone surface.

Measured Value:
This is the minimum distance between two concentric cones that contain all the measured points between them. The cones are concentric with the measured datum axis. They also have the same cone angle as the nominal cone surface.

Planar Features

Actual Value:
This is the minimum distance between two parallel planes that contain the entire actual surface between them. The planes are perpendicular to the actual datum axis.

Measured Value:
This is the minimum distance between two parallel planes that contain all the measured points between them. The planes are perpendicular to the measured datum axis.

Validity Rules

The datum reference frame must establish a clear datum axis.

Allowed Modifiers

None. This geometric tolerance does not allow modifiers.

Exposed Options

When at least one datum feature has surface data, the datum math type controls how to compute the measured datums from the datum features' surface data. For more information, see "How PC-DMIS Solves and Uses Datums".

Report

Here is an example report for a total runout tolerance of a cone.

FCFRNOUT_ISO_115	MM	0.01 A-B	LSQ	ISO 1101

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CON1	0.000	0.010		0.012	0.012	0.002

How PC-DMIS Solves and Uses Datums

In this Topic
How PC-DMIS Solves and Uses Datums

Introduction
Degrees of Freedom Constrained by a Datum Reference Frame
Datum Math Types under ASME Y14.5
Datum Math Types under ISO 1101
Modifiers for Datums
    The [DF] Modifier
Datum Planes with Surface Data under ASME Y14.5
Datum Planes with Surface Data under ISO 1101
Illustrations of Datum Planes: Filtration, Best Fitting, and Orientation Constraints
Datum Planes without Surface Data
Datum Plane Cross Sections
Datum Plane Samples
Datum Cylinders with Surface Data under ASME Y14.5
Datum Cylinders with Surface Data under ISO 1101
Illustrations of Datum Cylinders: Location Constraints and No Location Constraints
Datum Cylinders without Surface Data, and Surfaceless Axes
Datum Cylinder Cross Sections
Datum Widths under ASME Y14.5
Datum Widths under ISO 1101
Datum Slots and Notches
Datum Cones with Surface Data under ASME Y14.5
Datum Cones with Surface Data under ISO 1101
Datum Cones without Surface Data
Datum Spheres with Surface Data under ASME Y14.5
Datum Spheres with Surface Data under ISO 1101
Datum Spheres without Surface Data, and Surfaceless 3D Points
Datum Patterns
    Illustrations of Datum Patterns
Common Datums: Coaxial Cylinders
    Illustrations of Coaxial Datums
Common Datums: Offset Parallel Planes
Datums with a Material Modifier
    Illustrations of Datums with a Material Modifier
Determining the Size of the Material Boundary
    Rules to Determine Geometric Tolerance
        Primary Datums Under ISO 1101
        Secondary Datums Under ISO 1101
        Tertiary Datums Under ISO 1101
        Primary Datums Under ASME Y14.5
        Secondary Datums Under ASME Y14.5
        Tertiary Datums Under ASME Y14.5
        Datum Reference Messages
        Using Modifiers on Datums and Their Reported Measurement Results
Datums with Unconstrained Location Compared to Higher-Precedence Datums
Optimized Shift
    The Gage-Centered Pose
    Enabling Optimized Shift
    Single-Segment Examples
        Direction of the Reported Shift Values
    Composite Callout Examples

Introduction

Many geometric tolerances reference one or more datums. Datums serve to align the actual surfaces with the nominal geometry so that position, perpendicularity, runout, and other tolerances, can be evaluated with respect to those datums.

First, PC-DMIS best-fits all of the actual primary datum surfaces to their respective datum simulators.

Second, PC-DMIS best-fits the actual secondary datum surfaces to their datum simulators while preserving the relationship (interaction) between the actual primary datum surfaces and their datum simulators.

Third, PC-DMIS best-fits the actual tertiary datum surfaces to their datum simulators, while preserving the relationship (interaction) between the actual primary and secondary datum surfaces and their datum simulators.

Finally, all toleranced features are optimized into their tolerance zones, preserving the relationship (interaction) between the datum surfaces and their simulators.

This process is much like using a functional gage:

First, the part is brought to the gage and the actual primary datum surface is aligned to the gage's primary datum simulator.

Second, the actual secondary datum surface is aligned to the gage's secondary datum simulator, keeping the actual primary datum surface aligned to the gage's primary datum simulator.

Third, the actual tertiary datum surface is aligned to the gage's tertiary datum simulator, keeping the actual primary and secondary datum surfaces aligned.

Finally, the considered features are evaluated, using gage pins or something similar.

Terminology Note: There are several phrases used in the GDT standards to describe the concept of a datum simulator:

- ASME Y14.5 - 1994 and 2009 uses "datum simulator".
- ASME Y14.5 - 2018 uses "true geometric counterpart".
- ISO 5459 uses "associated datum feature".

These phrases are approximately equivalent. This document uses the phrase "datum simulator" for brevity.

There are two ways to think about the datum reference frame process, which are equivalent to each other.

The gage point of view.

The actual part can be aligned to the nominal geometry as described above. The nominal geometry (or gage) is fixed and the actual part is aligned to it.

The part point of view

The actual part is fixed and the nominal geometry (or gage) is aligned to it. Under the part point of view, PC-DMIS best-fits the datum simulators to the actual part. You can use either point of view to describe how geometric tolerances and datums are solved in PC-DMIS.

The remainder of this topic uses the part point of view to describe how PC-DMIS solves datums.

The ASME Y14.5.1 2019 math standard introduced the first mathematical definition for datums. Prior to this, ASME Y14.5.1M 1994 used the "candidate datum set" concept. The geometric tolerance command's DEFAULT Datum Math always yields a valid candidate datum. Therefore, the PC-DMIS behavior described in this section is the same for all versions of ASME Y14.5 (1994, 2009, and 2018).

Comparison to Past Practice

Prior to PC-DMIS 2020 R2, with the XactMeasure command, datum reference frames were handled rather like a PC-DMIS alignment, with a level feature, a rotate feature, and one or more origin features. All of these features you selected from the datum features. Starting with version 2020 R2, the geometric tolerance command does not use any alignment concepts for datum reference frames. Instead, it uses gage concepts for the datum reference frames. This enables support for more unusual datum reference frames that cannot be represented by the level-rotate-origin framework.

Degrees of Freedom Constrained by a Datum Reference Frame

A geometric tolerance without any datum references has no degrees of freedom constrained: all three degrees of translation and all three degrees of orientation are free. Each successive datum constrains more degrees of freedom. Specifically, after a datum has been solved, any translations and rotations are not allowed to alter the datum simulator. For example, after a primary datum plane is solved, the secondary and tertiary datums are not allowed to move the primary datum plane. That means translations within the primary datum plane are allowed, and rotations within the primary datum plane are allowed, but rotation or translation out of plane are not allowed.

These constraints are mathematically defined in terms of invariance classes. Each invariance class is defined below, with some examples of datums exhibiting the invariance class. The examples are not exhaustive: many more examples exist that are not listed.

- **Spherical invariance:** the spherical invariance class constrains three translation degrees of freedom and leaves unconstrained all 3 rotations about a center-point. This includes both spherical features and surfaceless 3D points.
- **Planar invariance:** the planar invariance class constrains rotation about two directions orthogonal to the surface normal, and translation along the surface normal. It leaves unconstrained the rotation about the surface normal and translation along the two directions orthogonal to the surface normal. This includes planes, plane cross sections (lines on a surface) referenced as a secondary or tertiary datum, plane samples (points on a surface), and all width types.
- **Cylindrical invariance:** the cylindrical invariance class constrains rotation about the two directions orthogonal to the axis vector, and translation along the two directions orthogonal to the axis vector. It leaves unconstrained the rotation about the axis, and translation along the axis. This includes cylinders, circles referenced as a secondary or tertiary datum, surfaceless axes, and conical surfaces which are treated as axis-only (see the subtopics on cones below).
- **Revolute invariance:** the revolute invariance class constrains rotation about the two directions orthogonal to the axis vector, and all three translation degrees of freedom. It leaves unconstrained the rotation about the axis. This includes circles referenced

as a primary datum (see "Datum Cylinder Cross Sections" below) and patterns of two non-concentric spheres.

- **Prismatic invariance:** the prismatic invariance class constrains all three orientation degrees of freedom, and translation along two directions orthogonal to the translation vector. It leaves unconstrained translation along the translation vector. This includes plane cross sections (lines on a surface) referenced as a primary datum (see "Datum Plane Cross-Sections" below) and patterns of parallel non-coaxial cylinders.
- **Complex invariance:** the complex invariance class constrains all degrees of freedom. This includes patterns of non-parallel cylinders and patterns of three non-coaxial spheres. Datum reference frames with complex invariance are often called fully-constrained datum reference frames.

When more than one datum is referenced, the invariance class of each datum must be combined so that none of the datum simulators are allowed to move.

Example 1: A primary datum plane combined with a secondary datum circle in that plane yields a revolute invariance class. The primary datum plane constrains three degrees of freedom, and the secondary datum circle constrains two more degrees of freedom (translation in the plane).

Example 2: A primary datum plane, a secondary datum line on a surface, and a tertiary datum point on a surface yields a complex invariance class. The primary datum constrains three degrees of freedom yielding a planar invariance class. Adding the secondary datum constrains two more degrees of freedom, yielding a prismatic invariance class. Adding the tertiary datum constrains the remaining translation degree of freedom, yielding a complex invariance class.

Example 3: A primary datum sphere combined with a secondary datum sphere (not concentric with the primary) has a revolute invariance class. The primary datum sphere constrains three degrees of translation, and the secondary datum sphere constrains two degrees of orientation. The line between the two spheres is the axis of the revolute invariance class.

Comparison to Past Practice

Starting in PC-DMIS 2020 R2, the geometric tolerance command analyzes the datum reference frame in terms of invariance classes. This enables correct handling of unusual datum reference frames where the vectors are not square to each other. For example, a primary datum plane with a secondary datum plane at a 30-degree angle to the primary has a prismatic invariance class. Prior to this version, PC-DMIS did not fully support these unusual datum reference frames.

Datum Math Types under ASME Y14.5

As discussed in "Specification Versus Verification" in "Introduction to Geometric Tolerances and Feature Control Frames", we offer multiple math types for datum computation. PC-DMIS provides two ways to compute an ASME measured datum simulator using measured surface data. These are the two datum math types available:

DEFAULT - This does a void filtration (planar surfaces only) and a constrained least squares best fit. This algorithm is extremely similar to the actual simulator definition and so it is a good choice when the measurement uncertainty of each point is much less than the form error of the surface.

LSQ - This does a plain least squares best fit to the surface data. No void filtering is done. This algorithm is mathematically rather different than the actual simulator definition, but it is a better choice than **DEFAULT** when the measurement uncertainty of each point is much larger than the form error of the surface and much larger than the orientation error of the surface to higher precedence datums.

In between the two extremes, where the measurement uncertainty of each point is at least on the same order of magnitude as the form error, and at most on the order of magnitude as the orientation error, it can be difficult to predict which math type will yield a closer measured approximation to the actual datum simulator. The only way to be sure which type is the better choice is to do a careful study:

First, use a collection of manufactured parts that represent the range of manufacturing errors to control.

Second, measure the parts densely, using lots of cross sections. Use high-end equipment with much lower measurement uncertainty of each point than the form error. Compute the datums with the **DEFAULT** datum math type described above.

Finally, measure the parts with the actual equipment and measurement strategies used in production.

You can experiment with the two math types to see which one is closer to the high-accuracy result.

For any datum math option, we recommend that you measure the datum surface densely to maximize how well the measured datum simulator approximates the actual datum simulator.

Datum Math Types under ISO 1101

As discussed in the "Specification Versus Verification" subtopic in "Introduction to Geometric Tolerances and Feature Control Frames", we offer multiple math types for datum computation. PC-DMIS provides three ways to compute an ISO measured datum simulator using measured data. These are the three datum math options available:

DEFAULT - This does a void filtration (all surface types) and a constrained min max, maximum inscribed, or minimum circumscribed best fit (depending on feature type, whether the surface is inner or outer, and the material modifier). This algorithm is extremely similar to the actual simulator definition, and so it is a good choice when the measurement uncertainty of each point is much less than the form error of the surface.

LSQ - This does a plain least squares best fit to the surface data. No void filtering is done. This algorithm is mathematically rather different than the actual simulator definition, but it is a better choice than **DEFAULT** when the measurement uncertainty of each point is much larger than the form error of the surface and much larger than the orientation error of the surface to higher precedence datums.

In between the two extremes, where the measurement uncertainty of each point is at least on the same order of magnitude as the form error, and at most on the order of magnitude as the orientation error, it can be difficult to predict which math type will yield a closer measured approximation to the actual datum simulator. The only way to be sure which type is the better choice is to do a careful study:

First, use a collection of manufactured parts that represent the range of manufacturing errors to control.

Second, measure the parts densely, using lots of cross sections. Use high-end equipment with much lower measurement uncertainty of each point than the form error. Compute the datums with the **DEFAULT** datum math type described above.

Finally, measure the parts with the actual equipment and measurement strategies used in production.

You can experiment with the two math types to see which one is closer to the high-accuracy result.

CL2 - This does a void filtration (all surface types) and a constrained least squares best fit. Like the **DEFAULT** datum math option, this option requires the measurement uncertainty of each surface point to be much less than the form error of the surface. This option deviates from ISO 5459 - 2011; reasons for doing so might include:

- Stability: The constrained least squares algorithm gives a more stable result than constrained min max, maximum inscribed, or minimum circumscribed best fits.
- Comparison to physical inspection methods: The constrained least squares algorithm is a closer approximation to a surface plate than constrained min max.
- Comparison to assembly: The constrained least squares algorithm is a closer approximation to the assembled state than constrained min max.

- Support future standards: The past several unpublished drafts of ISO 5459 have specified constrained least squares as the default, and so it seems likely the next edition of ISO 5459 will do the same.

For any datum math option, we recommend that you measure the datum surface densely to maximize how well the measured datum simulator approximates the actual datum simulator.

Modifiers for Datums

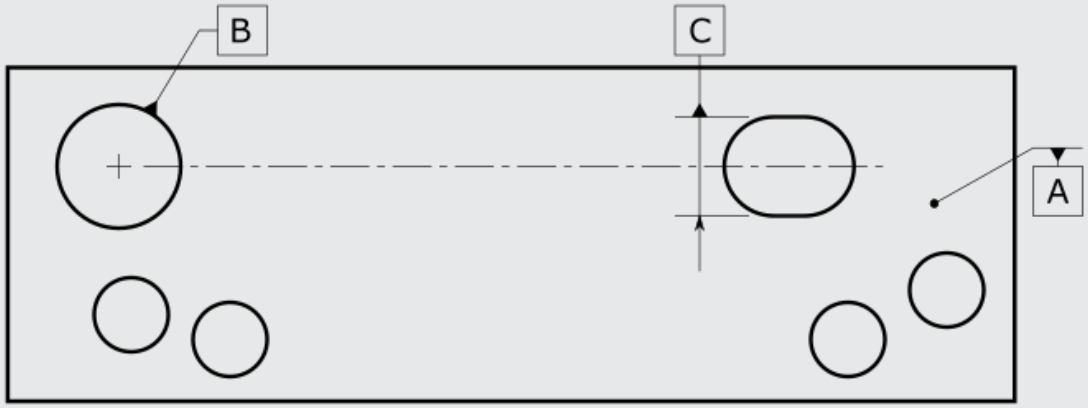
Datums can have several types of modifiers. The most common types are the material modifiers Ⓜ and Ⓛ. These modifiers are described in "Datums with a Material Modifier". When you mark the **Advanced Modifiers** check box, additional modifiers become accessible, including the translation modifier for ASME and the [DF] modifier for ISO.

The [DF] Modifier

The [DF] modifier is not standardized. This means it is not in any published ISO standard. The modifier has been introduced in some unpublished drafts of ISO 5459. In those drafts, [DF] stands for "Distance Fixed". This means the datum location is constrained to any higher-precedence datum or datums, overriding the default variable distance. Even though this modifier is not part of the standard, we found that this modifier is necessary to support several ISO functional requirements, especially plane-circle-width datum reference frames (and width variations like slots and mid-lines).

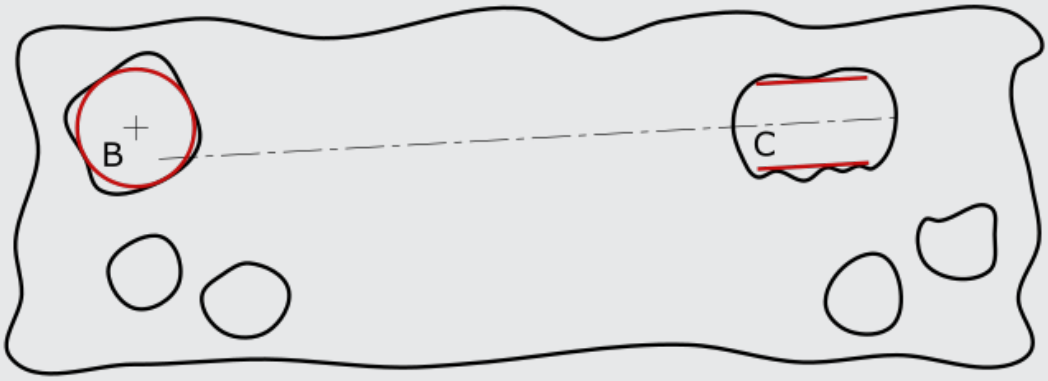
- Without the modifier, the orientation of the slot fully controls the in-plane rotation angle. This gives unstable, non-functional results (especially when the slot is short).
- With the modifier, the actual angle from the circle to the slot controls the in-plane rotation angle. This is the functional intent.

Suppose you have the following ISO specification, where the primary datum is A, the secondary datum is B, and the tertiary datum is C:



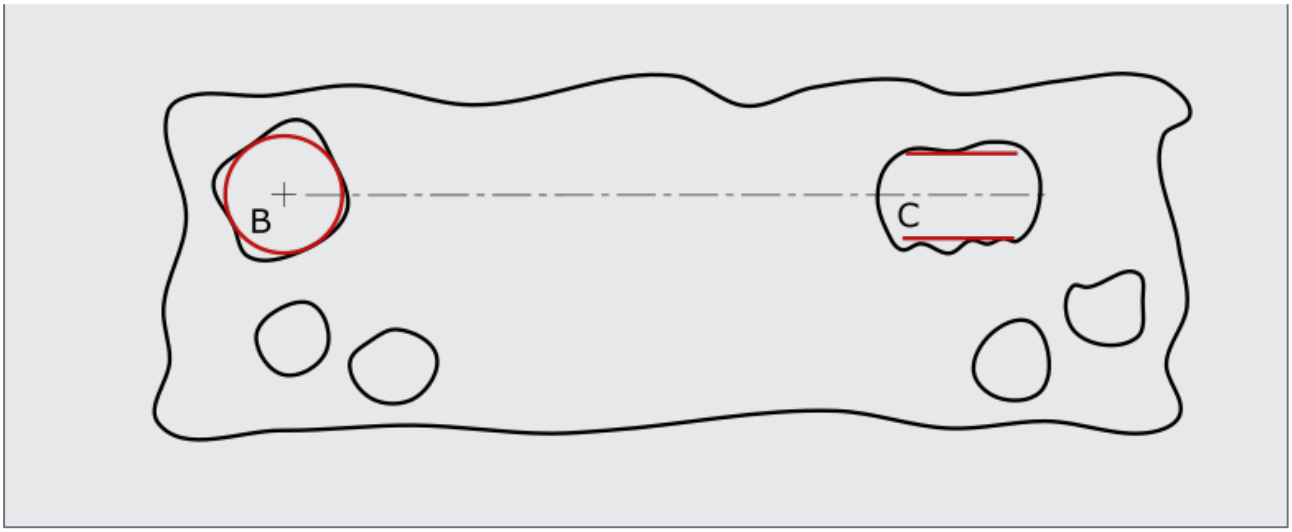
With No Modifiers

Of course, the real part has location, orientation, and form errors on the real surface. With no modifiers, the datum reference frame looks like the image below. The in-plane rotation angle comes completely from the orientation of datum C. Often, this makes the in-plane rotation angle unstable. This is because small variations in C lead to large changes in the in-plane rotation angle:



With the [DF] Modifier

For this datum reference frame, usually datum C needs a [DF] modifier. The modifier constrains the datums' relative location. This constraint makes datum C coplanar with datum B. This matches the typical functional intent and makes the datum reference frame more stable:



Datum Planes with Surface Data under ASME Y14.5

For planes, the actual datum plane simulator is defined by ASME Y14.5.1 - 2019. The actual surface is filtered to remove dents and other voids, and then a perfect plane is fitted to the filtered surface using constrained least squares. This fit makes the simulator external to the material while it maximizes contact and stability. In cases where the actual surface would rock, the constrained least squares definition produces a stabilized solution.

Secondary and tertiary actual datum plane simulators are nominally constrained in orientation to higher-precedence datum simulators. Secondary and tertiary datum planes are not restricted in location compared to higher-precedence datum simulators. This means that a translation modifier on a datum plane is the same thing as no translation modifier.

When datum planes have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on planes that have surface data, refer to "Feature Types With and Without Surface Data".

Datum Planes with Surface Data under ISO 1101

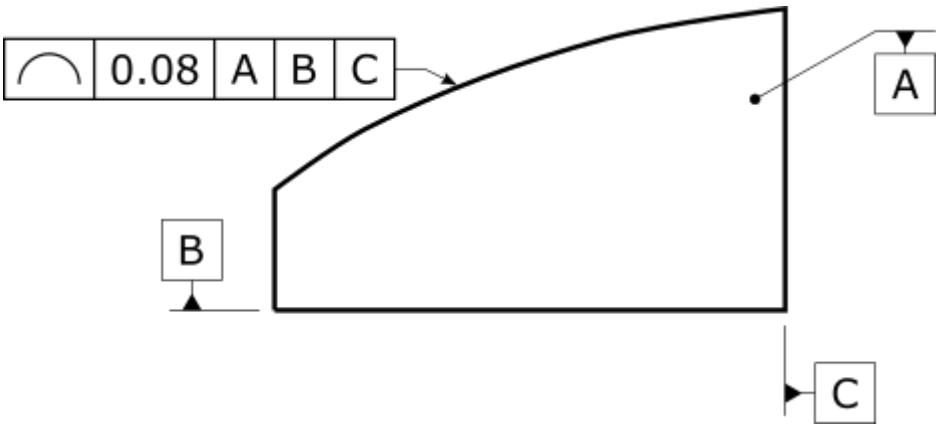
For planes, the actual datum plane simulator is defined by ISO 5459 - 2011, using the phrase "associated datum feature." The actual surface is filtered to remove dents and other voids, and then a perfect plane is fitted to the filtered surface using constrained min max. This fit makes the simulator external to the material, with the low points of the filtered surface as close as possible to the actual datum simulator.

Secondary and tertiary actual datum plane simulators are nominally constrained in orientation to higher-precedence datum simulators. They are not restricted in location compared to higher-precedence datum simulators unless the datum has a [DF] modifier. When the datum has a [DF] modifier, the simulators are nominally constrained in orientation and location to the higher-precedence datum simulators.

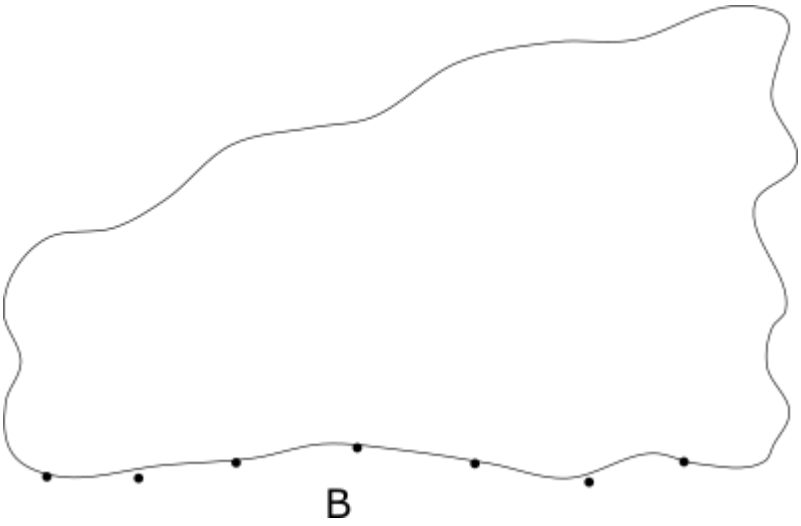
When datum planes have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on planes that have surface data, refer to "Feature Types With and Without Surface Data".

Illustrations of Datum Planes: Filtration, Best Fitting, and Orientation Constraints

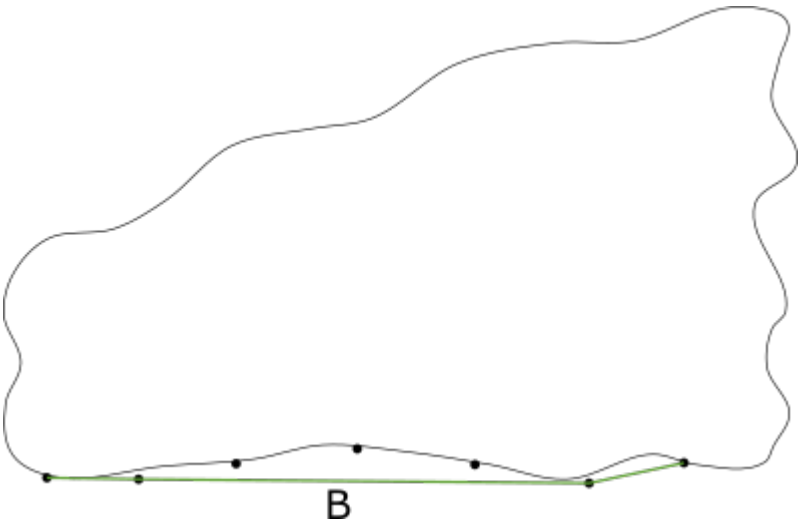
As an example of the best-fitting process, suppose you had the following specification:



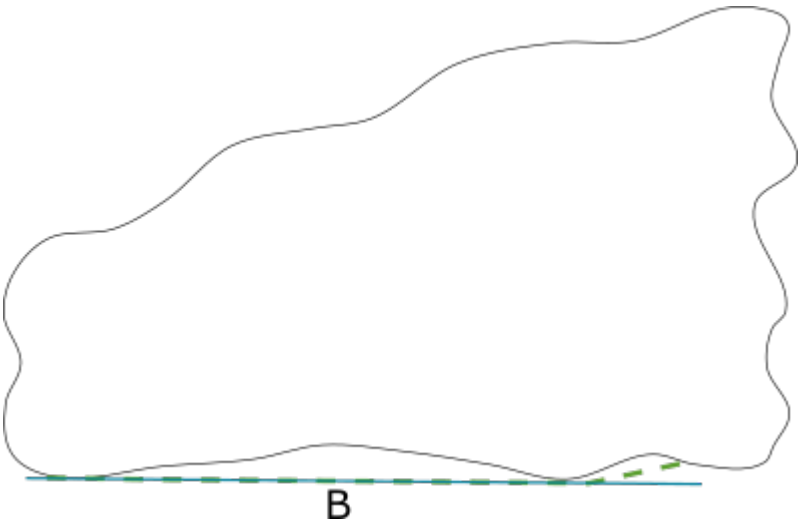
It's easier to illustrate void filtering and best fitting in 2D than 3D, so let's suppose datum B was measured as a line instead of as a plane (even though we recommend that you measure it as a plane). The measured points might look like this:



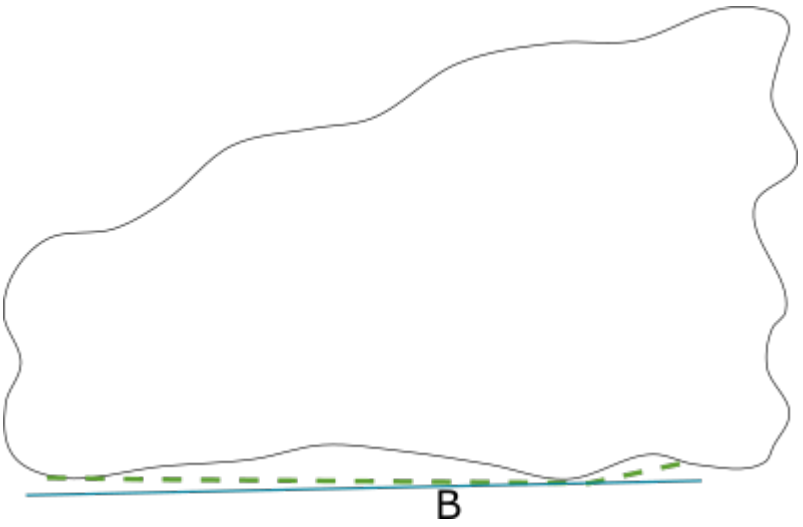
Then the void-filtered surface would look like shown below in green:



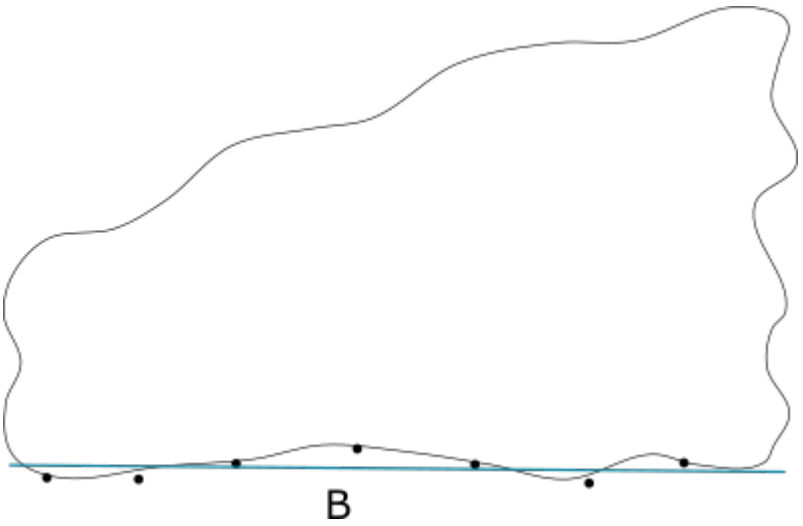
Then the best-fitted constrained least squares line (the result of the **DEFAULT** datum math option for ASME, or the **CL2** datum math option for ISO) is best fitted to the filtered surface, and looks like this shown in blue:



By contrast, the best-fitted constrained min max line (the result of the **DEFAULT** datum math option for ISO) is best-fitted to the filtered surface, and looks like this shown in blue:

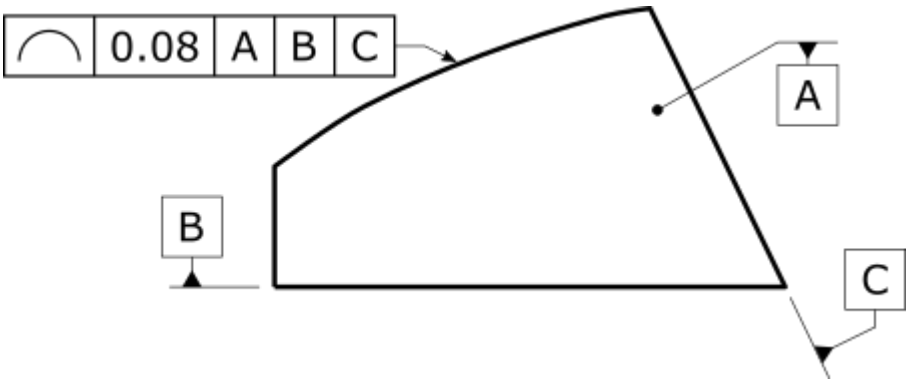


Meanwhile, the (unconstrained) least squares line (the result of the **LSQ** datum math option) is best-fitted to the original measured points, and looks like this shown in blue:

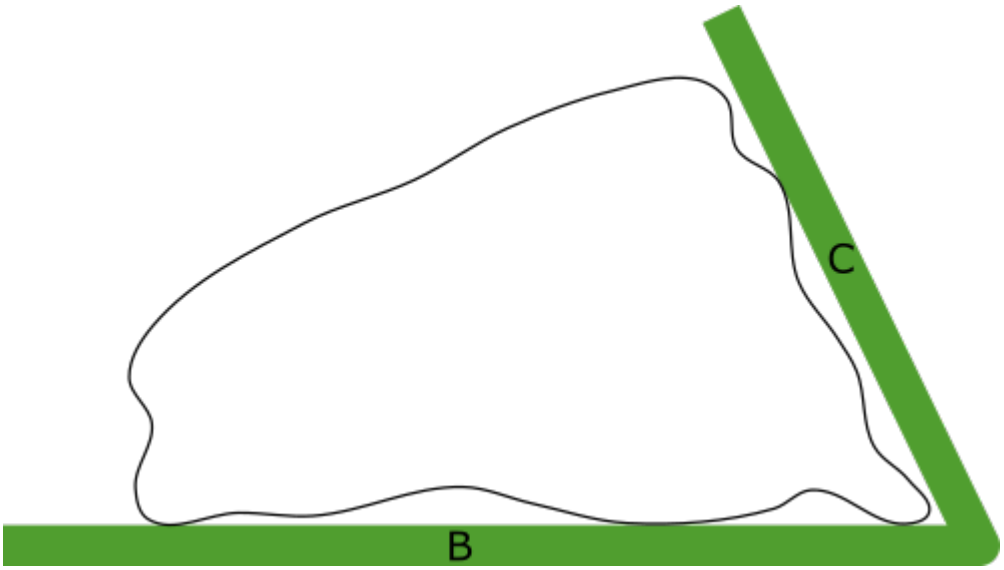


In this particular case, the **DEFAULT** datum math option is a better approximation to the actual datum than the **LSQ** datum math option. However, as seen in "Introduction to Geometric Tolerances and Feature Control Frames", the **LSQ** datum math option is a better approximation to the actual datum when the measurement uncertainty of each measured point is larger. Meanwhile, the **CL2** math type under ISO gives a better approximation to the assembled behavior of the datum, but a poorer approximation to the actual (specified) datum.

As an example of orientation constraints, suppose you had the following specification:



Then the actual (specified) datums for a realistic part might look like this:



Note that actual datum simulator C has the nominal angle to actual datum simulator B. Also note that the datum simulator C contacts the actual surface in exactly one point.

Datum Planes without Surface Data

In rare applications, a datum plane is defined that does not have a surface. For example, a plane that contacts three spheres. PC-DMIS supports applications like this by allowing for datum feature types that do not have surface data.

When a primary datum plane does not have surface data, the geometric tolerance command uses the MEAS values of the plane as the datum simulator.

When a secondary or tertiary datum plane does not have surface data, the geometric tolerance command constructs a measured plane that is as close as possible to the MEAS plane while being nominally oriented to the higher-precedence datum simulators. As seen above, ASME and ISO secondary and tertiary datum planes are not nominally located to higher-precedence datums (unless there is a [DF] modifier).

As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum planes without surface data, because you

are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

For details on planes that do not have surface data, refer to "Feature Types With and Without Surface Data".

Datum Plane Cross Sections

Datum line features can be either a cross section of a planar surface (see "Feature Types With and Without Surface Data") or a surfaceless axis. This subtopic here discusses the cross section case. The surfaceless axis is discussed in "Datum Cylinders without Surface Data, and Surfaceless Axes".

Because these types of line features represent a cross section of a planar surface, the above sections on datum planes apply here. For example, the actual datum simulator is an entire planar surface. By contrast, the measured datum simulator is a cross section of a planar surface because only a cross section of the surface was measured. This means we recommend that you do not use line features that represent a plane cross section as a primary datum. You should only use these line features as a datum if higher precedence datums have already defined the workplane of the line. However, PC-DMIS still allows these lines to be used as a primary datum. In that case PC-DMIS levels to the view of the line before it considers the datum line itself:

The view of a measured line or best-fit line is its workplane.

The view of an auto-line is its nominal surface vector.

Because lines are not recommended as a primary datum, PC-DMIS shows a warning message, as discussed in "Troubleshooting error messages and warnings". It may seem counter intuitive, but a primary datum line on a surface constrains five degrees of freedom (prismatic invariance class), while a primary datum plane constrains only three degrees of freedom (planar invariance class). This means if you measure less data, you are actually constraining more degrees of freedom. This is because you are requiring the line's view to become an implicit datum that is higher precedence than any datums in your feature control frame.

If you use a line on a surface as a primary datum, the warning cannot be turned off. This is because we strongly recommend that you do not use surface lines as primary datums. However, PC-DMIS supports this case for legacy applications. Instead, we recommend that you measure the datum plane first, and use that as the primary datum. Then measure a plane or line as the secondary datum.

Most line features referenced as a datum are on a surface—they have surface data. While it is possible in PC-DMIS to measure a line on a non-planar surface, the geometric tolerance command always handles lines on surfaces as coming from a planar surface. Because the measured datum simulator is only a cross section of the planar surface, the

measured algorithms (void filtering and fitting) are all two-dimensional instead of three-dimensional.

We do not recommend that you use lines representing cross sections of planar surfaces for datum features unless you already know that the orientation error is very small between higher precedence datums and the planar surface represented by the line. Instead, we recommend that you use plane features to represent the secondary or tertiary planar surface whenever possible.

PC-DMIS handles datum lines that represent a planar cross section that do not have surface data somewhat like datum planes that do not have surface data. The geometric tolerance command constructs a measured plane cross section that is as close as possible to the MEAS plane cross section while being nominally oriented to the higher-precedence datum simulators. As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum features without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

The measured surface normal of a line is parallel to the cross product of its line vector and its workplane vector.

For details on lines that represent a plane cross section and that have surface data, refer to "Feature Types With and Without Surface Data"

Datum Plane Samples

The geometric tolerance command treats most point types as a single sample of a planar surface. While it is possible in PC-DMIS to measure a point on a non-planar surface, the geometric tolerance command always handles datum points on a surface as if they come from a planar surface. Therefore, the above sections on datum planes apply here. For example, the actual datum simulator is an entire planar surface. By contrast, the measured datum simulator is a single sample of a planar surface. This is because only a sample of the surface was measured. That means void filtering does not take place. The orientation of the planar surface must be completely constrained by higher-precedence datums (no fitting takes place). You can only use a sample of a planar surface as a tertiary datum.

We do not recommend that you use points on a surface for datum features unless you already know that the orientation error is very small between higher precedence datums and the planar surface represented by the point. Instead, we recommend that you use plane features to represent the tertiary planar surface whenever possible.

For details on the points that represent a sample of a planar surface, refer to "Feature Types With and Without Surface Data"

Datum Cylinders with Surface Data under ASME Y14.5

For cylinders, the actual datum simulator is defined by ASME Y14.5.1 - 2019. A perfect cylinder is fitted to the actual surface using constrained least squares.

There is no void filtration. This fit makes the simulator external to the material while it maximizes contact and stability. In cases where the actual surface would rock, the constrained least squares definition produces a stabilized solution.

Secondary and tertiary actual datum cylinder simulators are nominally constrained in orientation and location to higher-precedence datum simulators. When a translation modifier is present, secondary or tertiary datum cylinders are not restricted in location compared to higher-precedence datum simulators, but they remain constrained in orientation.

When datum cylinders have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on the cylinders that have surface data, refer to "Feature Types With and Without Surface Data".

Datum Cylinders with Surface Data under ISO 1101

For cylinders, the actual datum simulator is defined by ISO 5459 - 2011, using the phrase "associated datum feature." The actual surface is filtered to remove dents and other voids, and then a perfect cylinder is fitted to the filtered surface using a maximum-inscribed (inner cylinders) or minimum-circumscribed (outer cylinders) formula. This fit makes the simulator external to the material and can be considered a mating envelope.

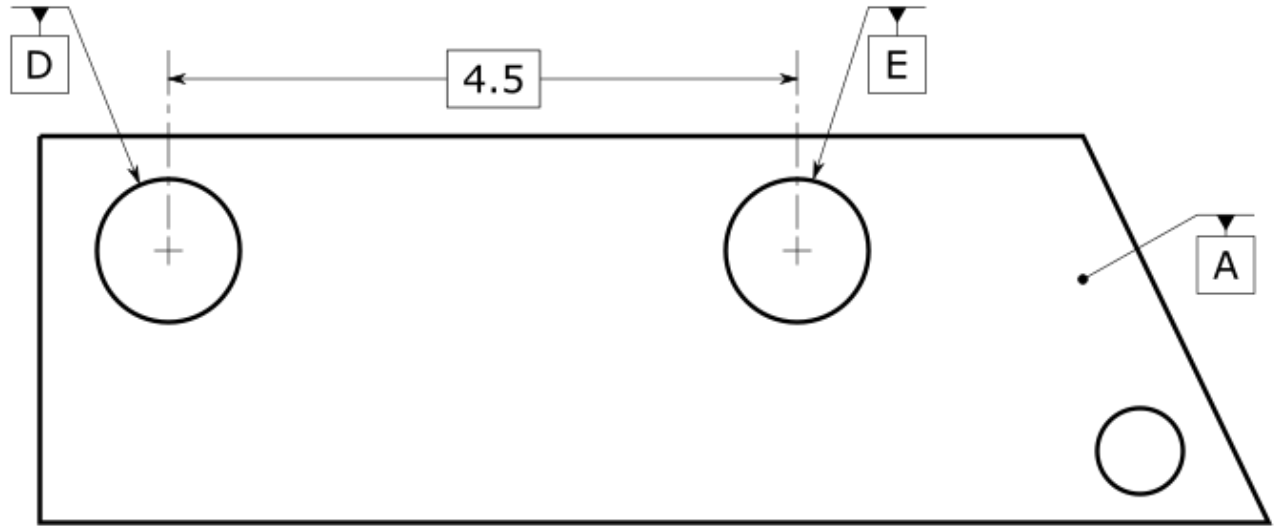
Unfortunately, inscribed and circumscribed algorithms are notoriously unstable, and so PC-DMIS, with the **DEFAULT** datum math type, uses a constrained least squares algorithm to inscribe or circumscribe this type of datum feature. The constrained least squares algorithm produces almost the same diameter as a pure inscribing or circumscribing, but the algorithm is much more stable. Therefore, **DEFAULT** and **CL2** datum math types are the same for this type of datum feature.

Secondary and tertiary actual datum cylinder simulators are nominally constrained in orientation to higher-precedence datum simulators. They are not restricted in location compared to higher-precedence datum simulators unless the datum has a [DF] modifier. When the datum has a [DF] modifier, the simulators are nominally constrained in orientation and location to the higher-precedence datum simulators.

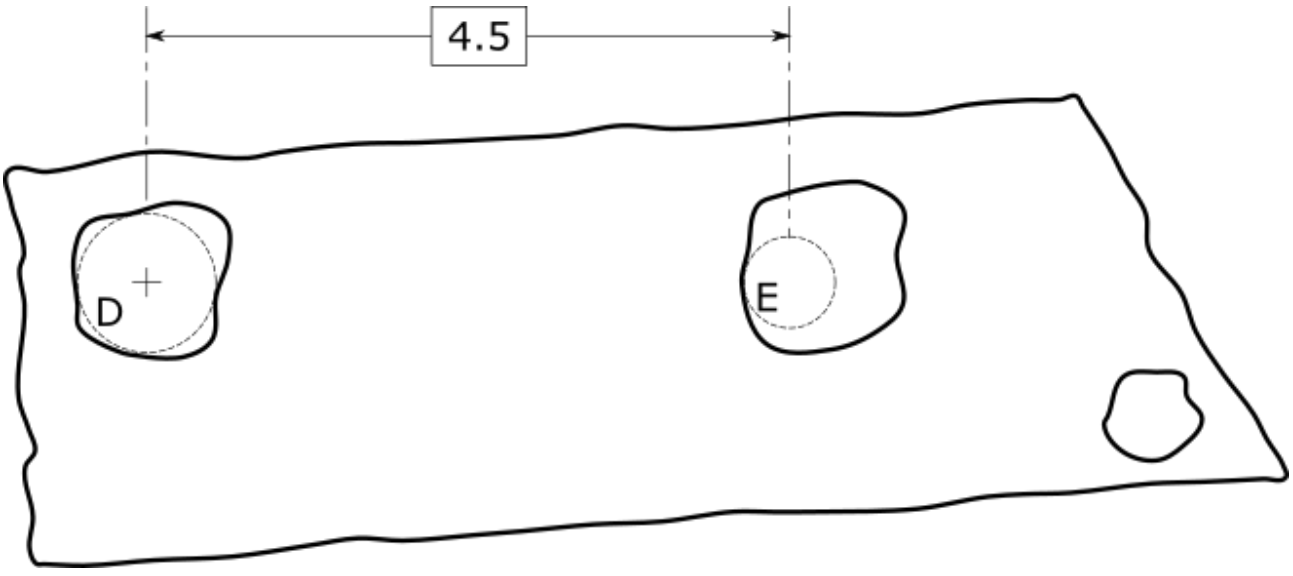
When datum cylinders have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on the cylinders that have surface data, refer to "Feature Types With and Without Surface Data".

Illustrations of Datum Cylinders: Location Constraints and No Location Constraints

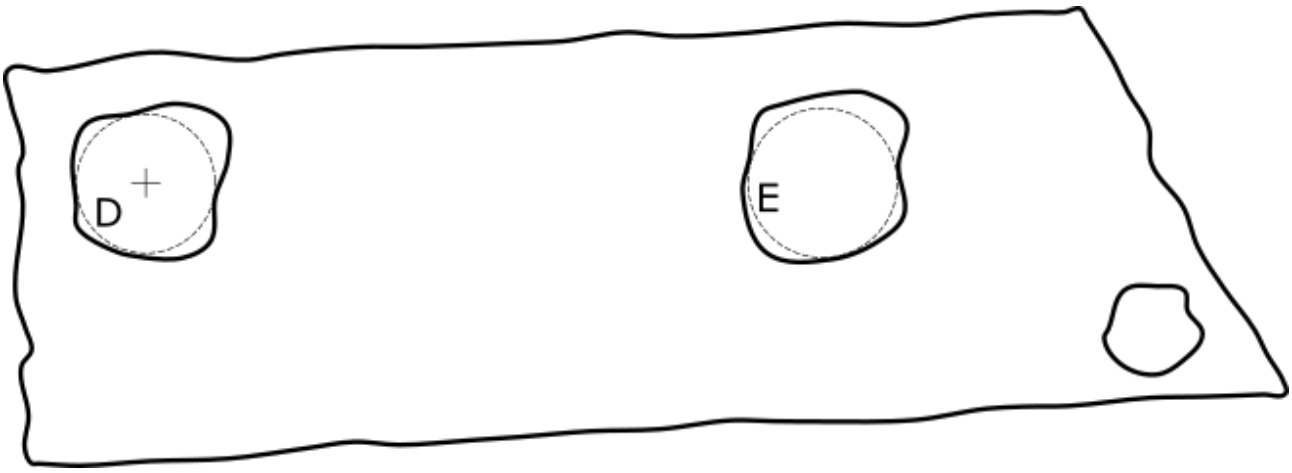
A frequently seen datum reference frame is the primary datum plane, followed by a secondary datum cylinder, followed by a tertiary datum cylinder. Here's an illustration of a specification, where the primary datum plane is A, the secondary datum cylinder is D, and the tertiary datum cylinder is E:



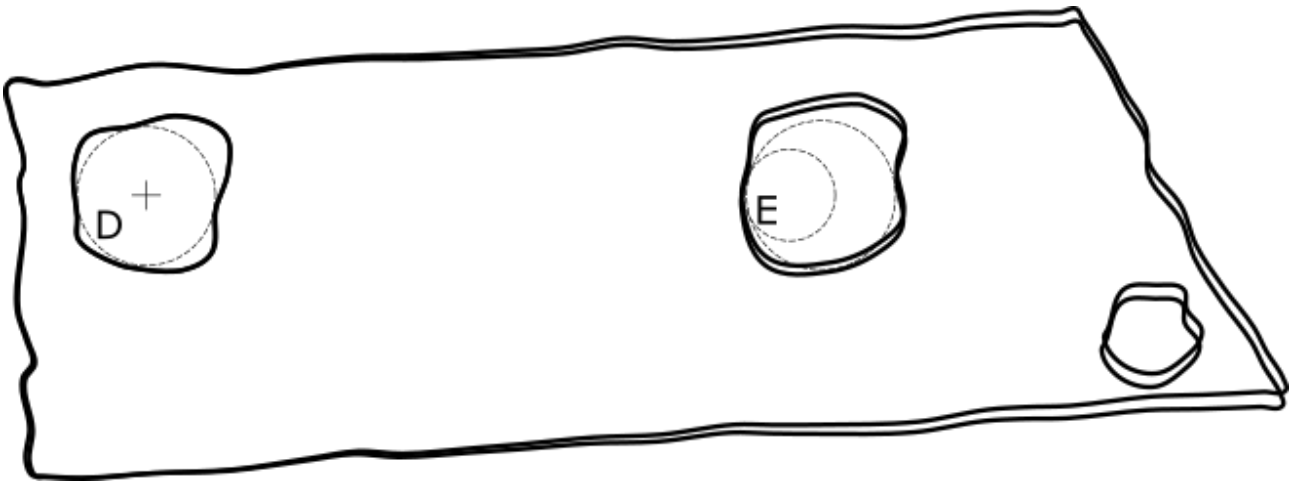
Under ASME (or under ISO with a [DF] modifier on the tertiary datum), the secondary and tertiary datums are nominally located with respect to each other. This results in the following datum reference frame:



Under ISO (or under ASME with a translation modifier on the tertiary datum), there are no location constraints between the secondary datum D and the tertiary datum E. This results in the following datum reference frame, which has a different rotation:



The difference between the two datum reference frames is easier to see when they are superimposed on each other:



Datum Cylinders without Surface Data, and Surfaceless Axes

In rare applications, a datum cylinder or axis is defined that does not have a surface. For example, a datum cylinder that circumscribes three pins. PC-DMIS supports applications like this by allowing for datum feature types that do not have surface data.

When a primary datum cylinder or axis does not have surface data, the geometric tolerance command uses the MEAS values of the cylinder or axis as the datum simulator.

When a secondary or tertiary datum cylinder or axis does not have surface data, the geometric tolerance command constructs a measured axis that is as close as possible to the MEAS axis while being nominally constrained to the higher-precedence datum simulators.

- For ASME datum cylinders or axes without surface data that do not have a translation modifier, and for ISO datum cylinders or axes without surface data

that have a [DF] modifier, the datum simulator is nominally located and oriented to the higher-precedence datum simulators.

- For ASME datum cylinders or axes without surface data that have a translation modifier, and for ISO datum cylinders or axes without surface data, the datum simulator is nominally oriented to the higher-precedence datum simulators (unless a [DF] modifier is present).

As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum cylinders or axes without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

For details on the cylinders that have and do not have surface data, and the feature types that represent surfaceless axes, refer to "Feature Types With and Without Surface Data".

Datum Cylinder Cross Sections

The geometric tolerance command treats datum circles as a cross section of a cylindrical surface. While PC-DMIS can measure a circle on a non-cylindrical surface, the geometric tolerance command always handles datum circles as coming from a cylindrical surface. Therefore, the above sections on datum cylinders apply here. For example, the actual datum simulator is an entire cylindrical surface. By contrast, the measured datum simulator is a cross section of a cylindrical surface. This is because only a cross section of the surface was measured. That means the measured algorithms (void filtering and fitting) are all two-dimensional instead of three-dimensional (surface data are filtered for ISO datum circles, not ASME ones). This also means it is not recommended to use a circle as a primary datum. More precisely, you should not use a circle as a datum until higher precedence datums have defined the vector of the cylindrical axis.

However, PC-DMIS allows circles to be used as a primary datum, in the which case PC-DMIS levels to the view of the circle before considering the datum circle itself. The view of a circle is its nominal vector. Because circles are not recommended as a primary datum, you will get a warning, as discussed in "Troubleshooting error messages and warnings". It may seem counter intuitive, but a primary datum circle constrains five degrees of freedom (revolute invariance class) while a primary datum cylinder constrains only four degrees of freedom (cylindrical invariance class). This means if you measure less data, you are actually constraining more degrees of freedom. This is because you are requiring the circle's view to become an implicit datum that is higher precedence than any datums in your feature control frame.

If you use a circle as a primary datum, the warning cannot be turned off. This is because we strongly recommend that you do not use circles as primary datums. However, PC-DMIS supports this case for legacy applications. Instead, we recommend that you measure the datum plane first, and use that as the primary datum. Then measure a cylinder or circle as the secondary datum.

For the reasons discussed in "Datum Cylinders with Surface Data under ISO 1101", datum circles in the geometric tolerance command that reference ISO 1101 are the same for **DEFAULT** and **CL2** datum math types.

We do not recommend that you use circles for datum features unless you already know that the orientation error is very small between higher precedence datums and the cylindrical surface represented by a circle. Instead, we recommend that you use cylinder features to represent the cylindrical surface whenever possible.

PC-DMIS handles datum circles without surface data somewhat similarly to datum cylinders without surface data. The geometric tolerance command constructs a measured cylinder cross section that is as close as possible to the MEAS cylinder cross section while being nominally constrained to the higher-precedence datum simulators.

- For ASME datum circles without surface data that do not have a translation modifier, and for ISO datum circles without surface data that have a [DF] modifier, the datum simulator is nominally located and oriented to the higher-precedence datum simulators.

- For ASME datum circles without surface data that have a translation modifier, and for ISO datum circles without surface data that do not have a [DF] modifier, the datum simulator is nominally oriented to the higher-precedence datum simulators.

As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum circles without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

For details on the circles that have and do not have surface data, refer to "Feature Types With and Without Surface Data".

Datum Widths under ASME Y14.5

For widths, the actual datum simulator is defined by ASME Y14.5.1 - 2019. A perfect width is fitted to the actual surface using constrained least squares.

There is no void filtration. This fit makes the simulator external to the material while it maximizes contact and stability. In cases where the actual surface would rock, the constrained least squares definition produces a stabilized solution.

Secondary and tertiary actual datum width simulators are nominally constrained in orientation and location to higher-precedence datum simulators. When a translation modifier is present, secondary or tertiary datum widths are not restricted in location compared to higher-precedence datum simulators, but they remain constrained in orientation.

All widths in PC-DMIS have surface data. PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option.

When possible, use the 3D width type. This is because it represents the entire width surface. When the width is too shallow to measure as a 3D width, you can use a 2D width as a secondary or tertiary datum. You should not use a 2D width as a datum until higher precedence datums have defined the workplane of the 2D width. 2D widths as datums have the same cautions as described in "Datum Plane Cross-Sections".

When the width is too small to use even as a 2D width, you can use a 1D width as a tertiary datum. Higher precedence datums must have completely defined the orientation of the 1D width surface. 1D widths have the same cautions as described in "Datum Plane Samples".

Datum Widths under ISO 1101

For widths, the actual datum simulator is defined by ISO 5459 - 2011, using the phrase "associated datum feature." The actual surface is filtered to remove dents and other voids, and then a perfect width is fitted to the filtered surface using a maximum-inscribed (inner widths) or minimum-circumscribed (outer widths) formula. This fit makes the simulator external to the material and can be considered a mating envelope.

Unfortunately, inscribed and circumscribed algorithms are notoriously unstable, and so PC-DMIS, with the **DEFAULT** datum math type, uses a constrained least squares algorithm to inscribe or circumscribe this type of datum feature. The constrained least squares algorithm produces almost the same diameter as a pure inscribing or circumscribing, but the algorithm is much more stable. Therefore, **DEFAULT** and **CL2** datum math types are the same for this type of datum feature.

Secondary and tertiary actual datum widths simulators are nominally constrained in orientation to higher-precedence datum simulators. They are not restricted in location compared to higher-precedence datum simulators unless the datum has a [DF] modifier. When the datum has a [DF] modifier, the simulators are nominally constrained in orientation and location to the higher-precedence datum simulators.

All widths in PC-DMIS have surface data. PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option.

When possible, use the 3D width type. This is because it represents the entire width surface. When the width is too shallow to measure as a 3D width, you can use a 2D width as a secondary or tertiary datum. You should not use a 2D width as a datum until higher precedence datums have defined the workplane of the 2D width. 2D widths as datums have the same cautions as described in "Datum Plane Cross-Sections".

When the width is too small to use even as a 2D width, you can use a 1D width as a tertiary datum. Higher precedence datums must have completely defined the orientation of the 1D width surface. 1D widths have the same cautions as described in "Datum Plane Samples".

Datum Slots and Notches

In some cases, it makes sense to use a slot or notch command as a secondary or tertiary datum. The geometric tolerance command treats datum slots and notches like 2D widths without any surface data. While slot and notch commands typically do have surface data, they do not collect enough surface data in the right places to use that surface data in a geometric tolerance context. Therefore, the geometric tolerance command treats slots and notches as 2D widths without surface data.

When you use slots and notches as a datum without any modifier, PC-DMIS treats them like a mid-line: a datum plane cross section that does not have surface data. When you use them as a datum with a material modifier Ⓜ or Ⓛ, PC-DMIS treats slots and notches much like a 2D width without surface data would be treated. For more information, see "Datums with a Material Modifier".

Be careful with datum slots and notches.

You should only use them if you already know that the form of the features is very good. If you suspect that the manufactured form error might be significant, do not use a slot or notch command. Instead, measure a scan around the perimeter of the feature and then tolerance the form, orientation, and location of the feature with a profile of a line tolerance.

If you need to reference the feature as a datum, instead of a slot or notch, use a constructed 2D or 3D width (with surface data).

Datum Cones with Surface Data under ASME Y14.5

For cones, the actual datum simulator is incompletely defined by ASME Y14.5.1 - 2019. A perfect cone is fitted to the actual surface using constrained least squares.

There is no void filtration. This fit makes the simulator external to the material while it maximizes contact and stability. In cases where the actual surface would rock, the constrained least squares definition produces a stabilized solution.

ASME Y14.5 specifies that primary datum cones constrain five degrees of freedom: three degrees of translation and two of rotation. Only a single degree of rotation is left (rotation about the cone axis). Unfortunately, ASME Y14.5 and Y14.5.1 are ambiguous in how translation along the axis is constrained (there are multiple possible interpretations, each yielding a different constrained translation along the axis). Furthermore, in our experience most drawings that reference datum cones are intended to use only the axis of the cone as the datum. Therefore, the PC-DMIS geometric tolerance command treats all datum cones as axis-only. This means they constrain at most four degrees of freedom (not translation along the cone).

Under ASME Y14.5, the geometric tolerance command fits datum cones such that both the central diameter and cone angle are optimized.

Secondary and tertiary actual datum cone simulators are nominally constrained in orientation and location to higher-precedence datum simulators. When a translation modifier is present, secondary or tertiary datum cones are not restricted in location compared to higher-precedence datum simulators, but they remain constrained in orientation.

When datum cones have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on the cones that have surface data, refer to "Feature Types With and Without Surface Data".

Datum Cones with Surface Data under ISO 1101

For cones, the actual datum simulator is defined by ISO 5459 : 2011. A perfect cone is fitted to the actual surface using constrained min max. The actual surface is filtered to remove dents and other voids, and then a perfect cone is fitted to the filtered surface using constrained min max. This fit results in the simulator being external to the material, with the low points of the filtered surface as close as possible to the actual datum simulator. ISO 5459 : 2011 further specifies that the cone angle is fixed at nominal (not optimized).

ISO 5459 specifies that primary datum cones constrain five degrees of freedom: three degrees of translation and two of rotation. Only a single degree of rotation is left (rotation about the cone axis). Unfortunately, this gives an unstable translation along the cone axis. This is because small changes in actual cone diameter frequently yield large changes in the defined translation along the cone axis. Furthermore, in our experience most drawings referencing datum cones are intended to use only the axis of the cone as the datum. Therefore, the PC-DMIS geometric tolerance command treats all datum cones as axis-only: they only constrain at most four degrees of freedom (translation along the cone is free). This is equivalent to PC-DMIS assuming that the [SL] modifier is present on all datum cones (either implicitly or explicitly).

Secondary and tertiary actual datum cone simulators are nominally constrained in orientation to higher-precedence datum simulators. They are not restricted in location compared to higher-precedence datum simulators unless the datum has a [DF] modifier. When the datum has a [DF] modifier, the simulators are nominally constrained in orientation and location to the higher-precedence datum simulators.

When datum cones have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on the cones that have surface data, refer to "Feature Types With and Without Surface Data".

Datum Cones without Surface Data

When a cone without surface data is referenced as a datum, because cones are treated as axis-only, the behavior is the same as described above in "Datum Cylinders without Surface Data, and Surfaceless Axes". As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum cones without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

For details on the cones that do not have surface data, refer to "Feature Types With and Without Surface Data".

Datum Spheres with Surface Data under ASME Y14.5

For spheres, the actual datum simulator is defined by ASME Y14.5.1 - 2019. A perfect sphere is fitted to the actual surface using constrained least squares.

There is no void filtration. This fit makes the simulator external to the material while it maximizes contact and stability. In cases where the actual surface would rock, the constrained least squares definition produces a stabilized solution.

Secondary and tertiary actual datum sphere simulators are nominally constrained in location to higher-precedence datum simulators unless a translation modifier is present. This is because spheres have no orientation and so are not constrained in orientation to higher precedence datums.

When datum spheres have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details of what spheres have surface data, refer to "Feature Types With and Without Surface Data".

Datum Spheres with Surface Data under ISO 1101

For spheres, the actual datum simulator is defined by ISO 5459 - 2011, using the phrase "associated datum feature." The actual surface is filtered to remove dents and other voids, and then a perfect sphere is fitted to the filtered surface using a maximum-inscribed (inner-spheres) or minimum-circumscribed (outer-spheres) formula. This fit makes the simulator external to the material and can be considered a mating envelope.

Unfortunately, inscribed and circumscribed algorithms are notoriously unstable, and so PC-DMIS, with the **DEFAULT** datum math type, uses a constrained least squares algorithm to inscribe or circumscribe this type of datum feature. The constrained least squares algorithm produces almost the same diameter as a pure inscribing or circumscribing, but the algorithm is much more stable. Therefore, **DEFAULT** and **CL2** datum math types are the same for this type of datum feature.

Secondary and tertiary actual datum spheres are not restricted in location or orientation compared to higher-precedence datum simulators, unless the datum has a [DF] modifier. This is because spheres have no orientation and so are not constrained in orientation to higher precedence datums. When the datum has a [DF] modifier, the simulators are nominally constrained in location to the higher-precedence datum simulators.

When datum spheres have surface data, PC-DMIS computes a measured approximation to the actual datum using that surface data and using the chosen datum math option. For details on the spheres that have surface data, refer to "Feature Types With and Without Surface Data".

Datum Spheres without Surface Data, and Surfaceless 3D Points

In rare applications, a datum sphere or 3D point is defined that does not have a surface. For example, a datum sphere that circumscribes three spheres. PC-DMIS supports applications like this by allowing for datum feature types that do not have surface data.

When a primary datum sphere or surfaceless 3D point does not have surface data, the geometric tolerance command uses the MEAS values of the sphere as the datum simulator.

When a secondary or tertiary datum sphere or surfaceless 3D point does not have surface data, the geometric tolerance command uses the MEAS values of the sphere as the datum simulator. For ASME datum spheres or surfaceless 3D points without surface data that do not have a translation modifier, and for ISO datum spheres with a [DF] modifier, a re-location is done in a way that minimally translates the centroid and ensures the datum simulator is nominally located to higher-precedence datum simulators.

As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum spheres or surfaceless 3D points without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

For details on the spheres that do not have surface data, and the feature types that represent surfaceless 3D points, refer to "Feature Types With and Without Surface Data".

Datum Patterns

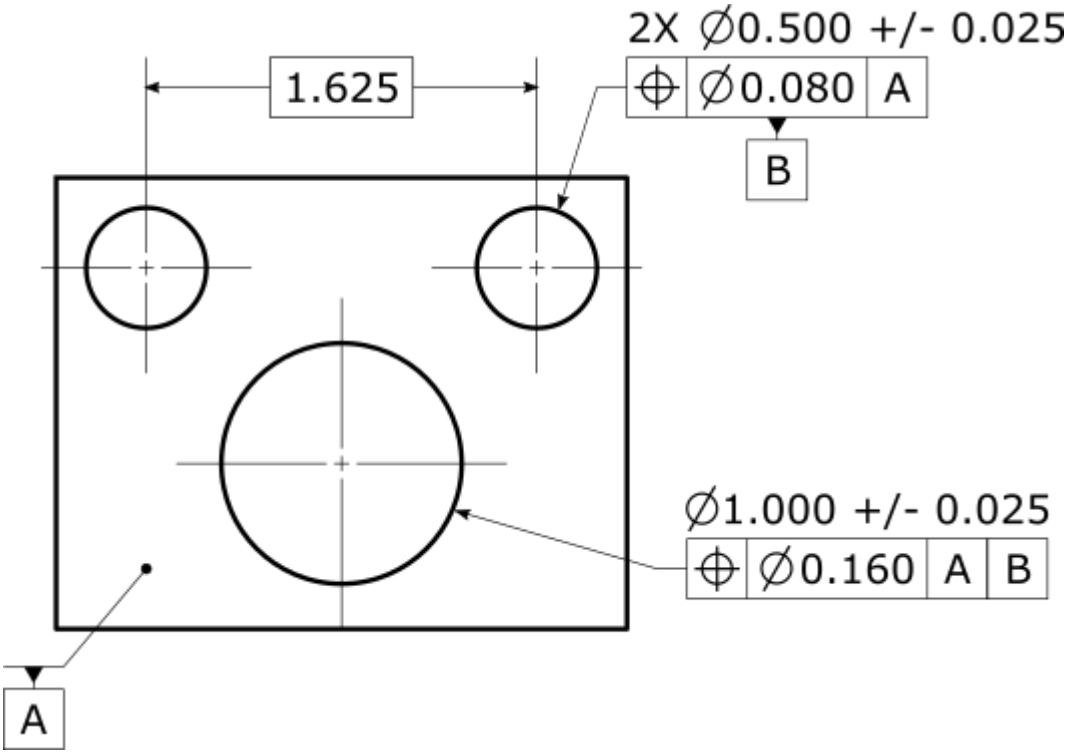
Datum patterns consist of features of size (cylinders, circles, widths, and spheres) that have the same nominal size, size tolerance, and are all inner or all outer. The datum simulators for the pattern are nominally oriented and located with respect to each other.

Under ASME Y14.5, the sizes of the simulators match each other because the nominal sizes and size tolerances and the in and out options are all the same. This is because paragraph 7.12.4 of ASME Y14.5 2018 states that the simulators must grow and shrink simultaneously. The set of actual datum simulators is defined by a constrained least squares fit. The fit is applied to the features' surfaces simultaneously, keeping the nominal location and orientation between each simulator as well as keeping the sizes matched. There is no void filtration.

ISO 5459 does not clearly state whether the simulators must have matched sizes or independent sizes. In our interpretation, the sizes are independent for ISO datum simulators within a pattern. See the example after paragraph 6.2.3 of ISO 5459 : 2011, as well as Figure A.8 which informed our interpretation. The set of actual datum simulators is defined by a constrained min max fit. The fit is applied to the features' void-filtered surfaces simultaneously, keeping the nominal location/orientation between each simulator, but allowing the sizes to vary independently.

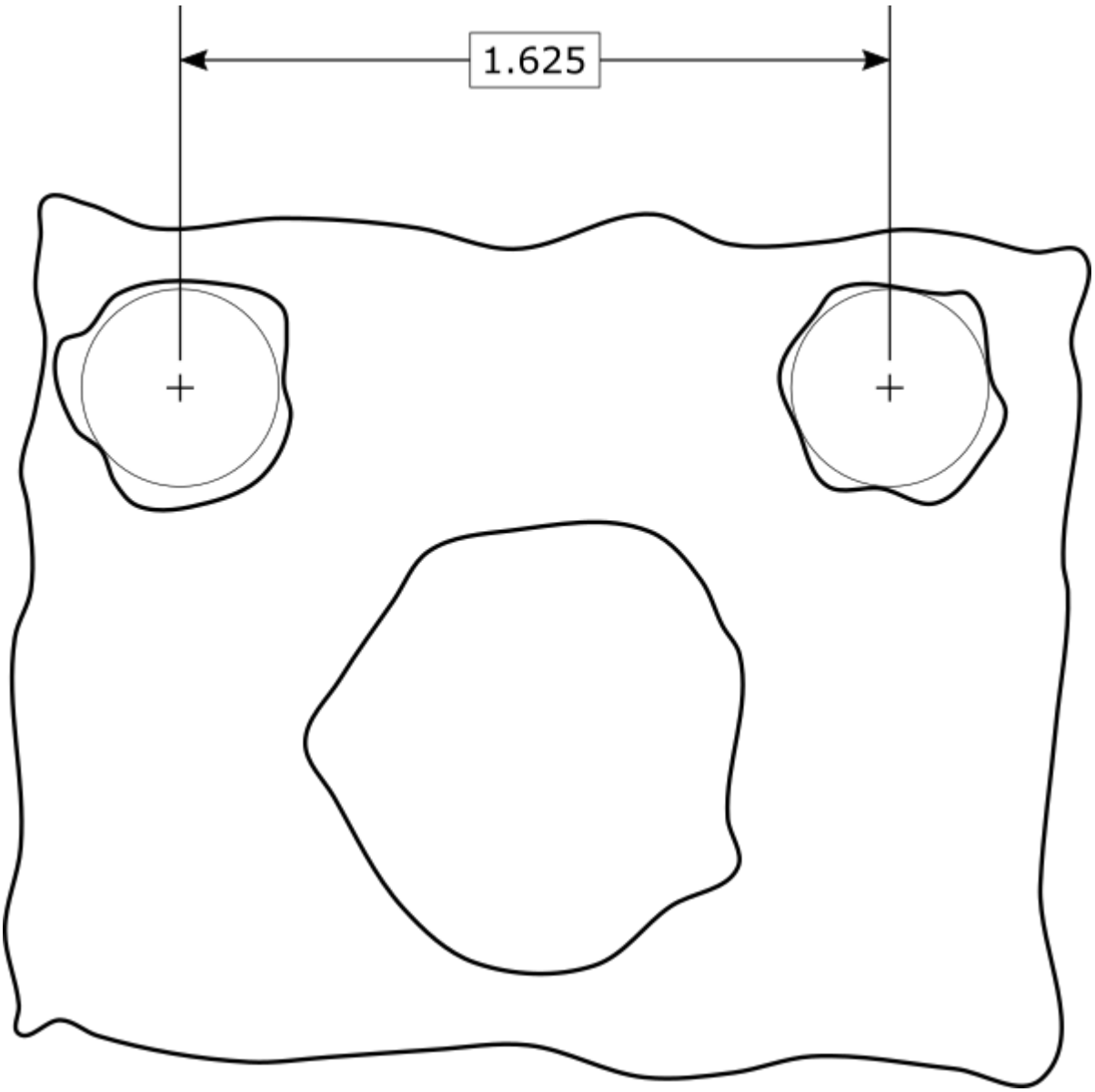
Illustrations of Datum Patterns

Suppose you had the following specification:



ASME

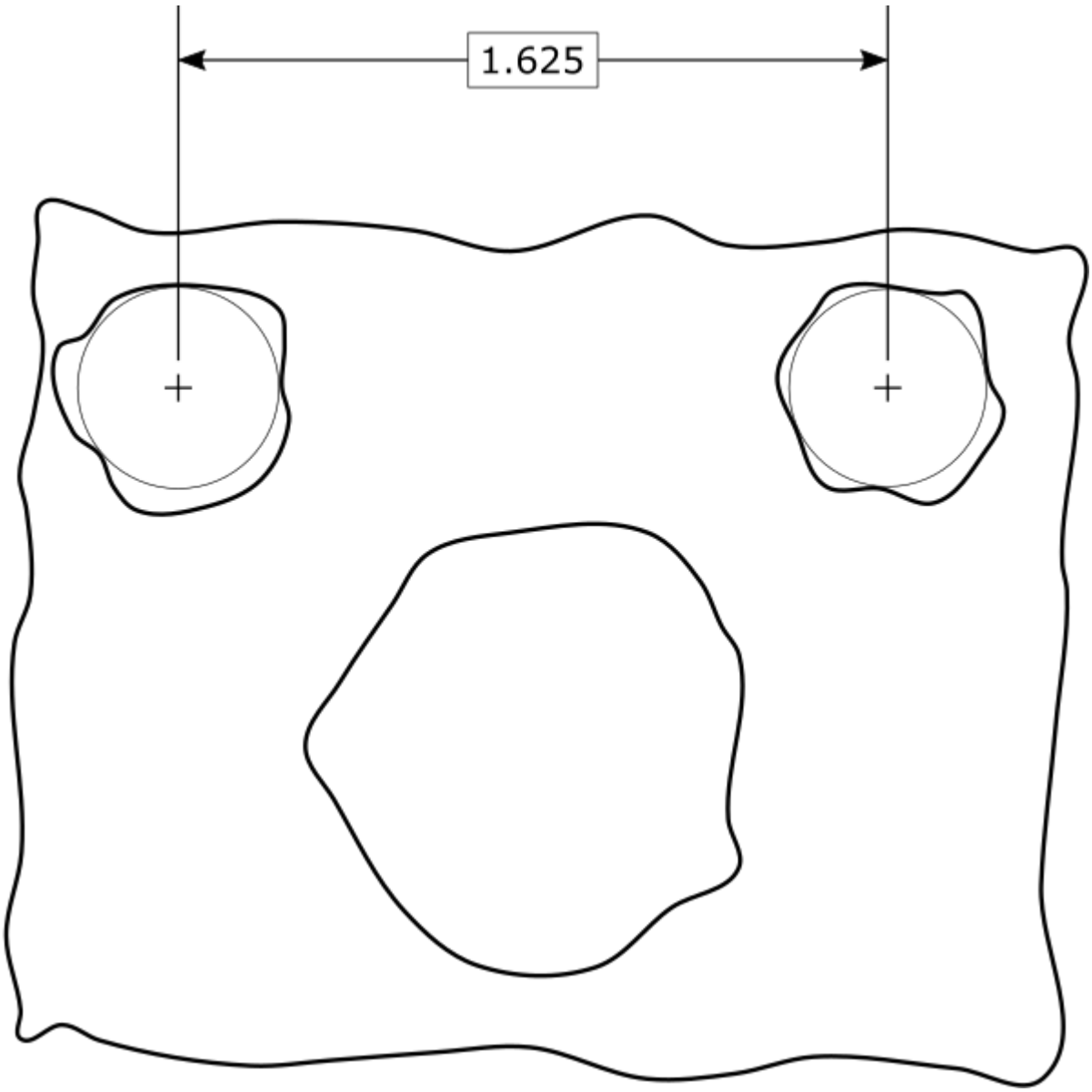
Given the above specification, the actual A | B datum reference frame under ASME is fully constrained, and looks like this:



Notice that the pattern's datum simulators are nominally located with respect to each other and have the same size.

ISO

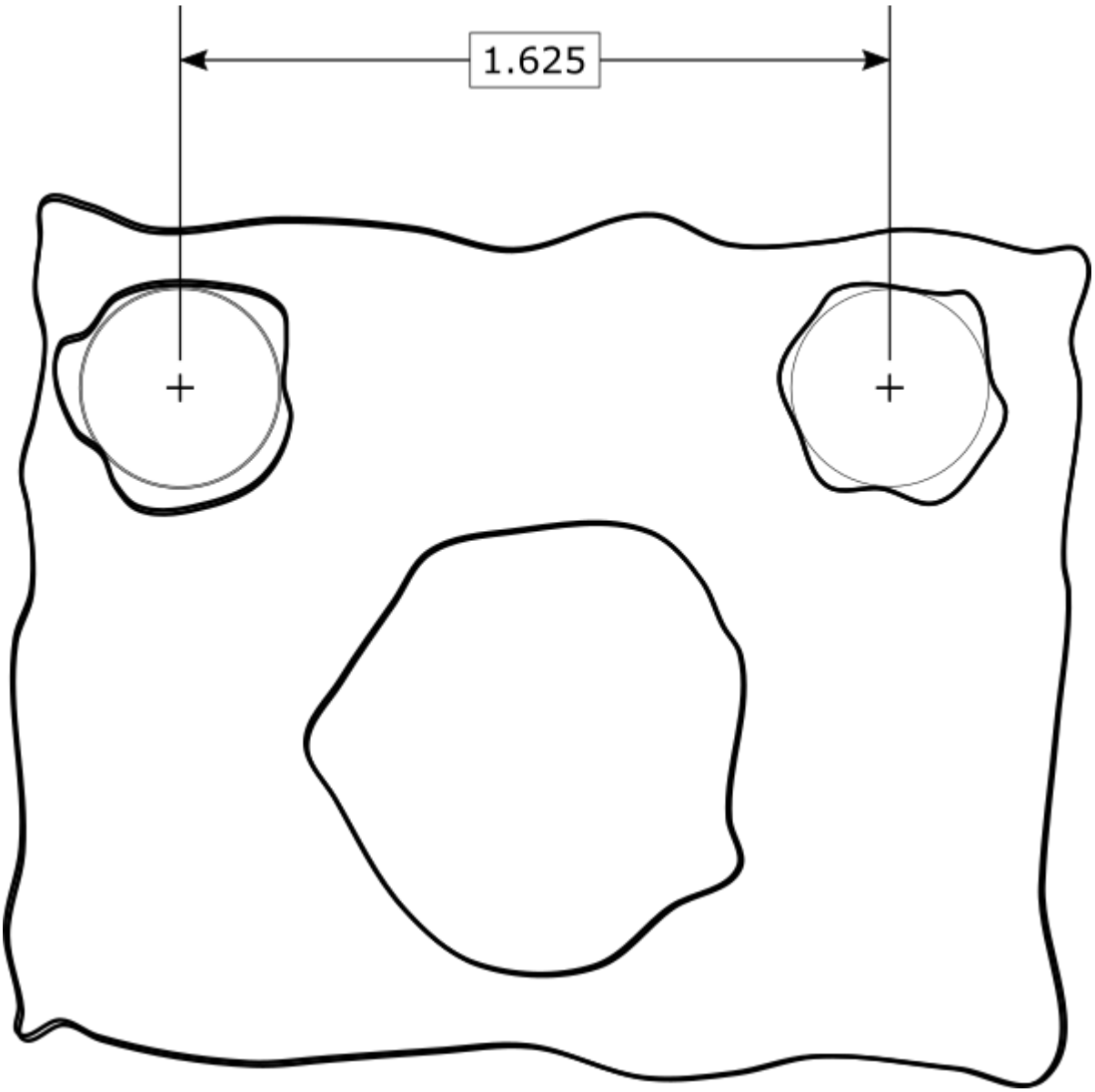
Given the above specification, the actual A | B datum reference frame under ISO is also fully constrained, and looks like this:



Notice that the pattern's datum simulators are nominally located with respect to each other, but do not have the same size. The overall rotation is different between ASME and ISO since the datum sizes are different.

ASME and ISO Superimposed

It's easier to see the difference between ASME and ISO when the two above images are superimposed:



Common Datums: Coaxial Cylinders

A common datum of multiple coaxial cylinders uses a hyphen in the datum reference, for example A-B or A-D-F. Typically, these items will differ between the cylinders: the nominal sizes, the size tolerance, or the in or out state. The datum simulators for the pattern are nominally oriented and located with respect to each other. This means the simulators will be coaxial.

Under ASME Y14.5, the sizes of the simulators are related to each other but typically do not match each other. Paragraph 7.12.4 of ASME Y14.5 2018 states that the simulators must grow and shrink simultaneously from their respective MMBs to their LMBs. The set of actual datum simulators is defined by a constrained least squares fit to the features' surfaces simultaneously. The fit also keeps these: the nominal location and orientation between each simulator and the sizes related by the simultaneous growing or shrinking from the maximum material boundary (MMB) to the least material boundary (LMB). There is no void filtration.

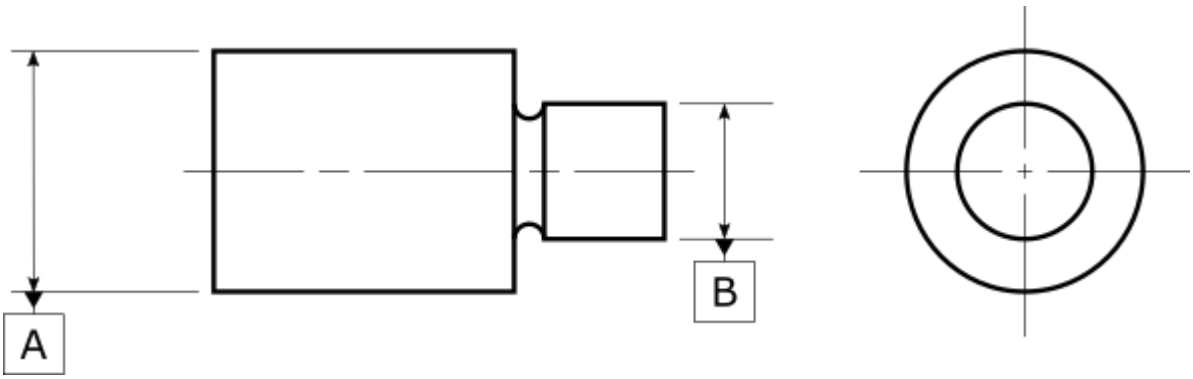
In order for PC-DMIS to correctly grow or shrink the simulator sizes simultaneously, you must create the size tolerances and geometric tolerances on datums *before* you allow any geometric tolerances to reference those datums. That is, the tolerances on the datum must be earlier in the measurement routine than the geometric tolerances referencing the datum.

If you later edit any size tolerance on a datum, you must ensure that all subsequent geometric tolerances that reference that datum have the correct size tolerance information for the datum.

ISO 5459 does not clearly state whether the simulators must have related sizes or independent sizes. In our interpretation, the sizes are independent for ISO datum simulators within a common datum. See the example after paragraph 6.2.3 of ISO 5459 : 2011, as well as Figure A.8 which informed our interpretation. The set of actual datum simulators is defined by a constrained min max fit to the features' void-filtered surfaces simultaneously. This fit also keeps the nominal location and orientation between each simulator, but it allows the sizes to vary independently.

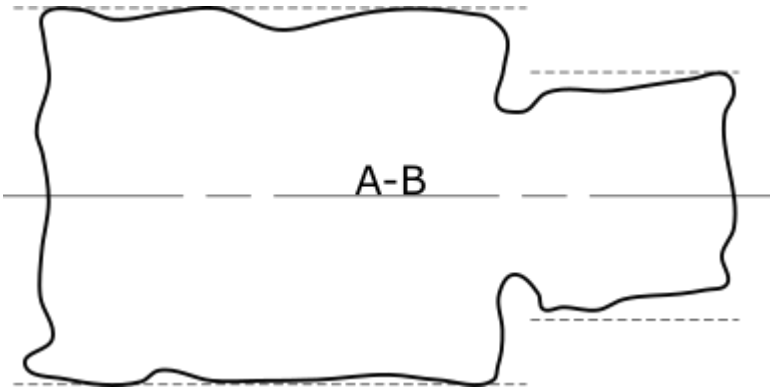
Illustrations of Coaxial Datums

Suppose you had the following specification:



ASME

In that case, the A-B common datum under ASME would look this:



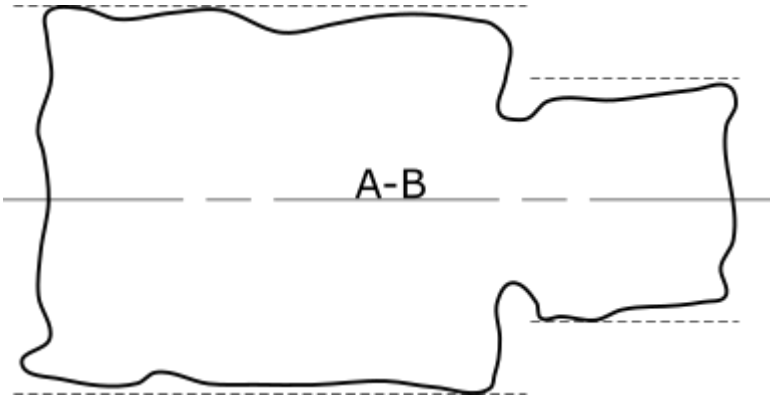
Notice that the A and B simulators are exactly coaxial.

ISO

With the above specification, the A-B common datum under ISO would be different, for two reasons:

1. The less-important reason is that the ISO one has independent sizes.
2. The more-important reason is that the ISO association for common datums is constrained min-max by default. This minimizes the distance between the low points of the filtered datum surfaces and the simulators.

The ISO common datum looks like this:



Common Datums: Offset Parallel Planes

A common datum of offset parallel planes uses a hyphen in the datum reference, for example A-B or A-D-F. The datum simulators for the pattern are nominally oriented and located with respect to each other. This means the simulators will be parallel and offset by their nominal distances.

Under ASME Y14.5, the set of actual datum simulators is defined by a constrained least squares fit to the features' void-filtered surfaces simultaneously. This fit also keeps the nominal location/orientation between each simulator.

Under ISO 1101, the set of actual datum simulators is defined by a constrained min max fit to the features' void-filtered surfaces simultaneously, keeping the nominal location and orientation between each simulator.

Datums with a Material Modifier

Datum cylinders, circles, spheres, widths, slots, and notches may have material modifiers Ⓜ or Ⓛ. A material modifier makes the geometric tolerance command handle the datum differently than datums that don't have them.

Without a material modifier, the datum completely constrains the normal degrees of freedom.

With a material modifier, the datum merely requires that a material boundary fits inside the feature surface or that the feature surface fits inside a material boundary.

The behavior is much like a functional gage. For example, in a typical case, the physical datum simulator on the gage is a pin, and it must fit in the hole on the actual part, but it is allowed to wiggle within the hole. Thus, the normal degrees of freedom are not completely constrained.

The geometric tolerance command approximates the interaction between the boundary and the measured surface. It does this with an axis-in-zone approximation. First, a surface envelope is computed. Then, the axis of the surface envelope is constrained to be within a perfect-form zone. The surface envelope size and the material boundary size determine the size of the zone. The zone is nominally oriented and located to higher-precedence datums. The zone is like the feature itself: spherical for spherical features, diametric for cylindrical and circular features, and planar for width, slot, and notch features. Here, we use the "axis of the surface envelope" concept rather loosely:

For a datum sphere, it is a single point.

For a datum cylinder, it is the axis.

For a datum width, it is the center-plane.

This axis-in-zone approximation is typically conservative unless the orientation error of the mating envelope is extreme. It is also typically an excellent approximation unless the form error of the surface is extreme. The conservative nature of the approximation means that even if the form error of the surface is extreme, the geometric tolerance command will not accept nonconforming parts, so long as the orientation error of the mating envelope is not extreme. PC-DMIS uses this approximation for two main reasons: (1) computation times are much faster, and (2) it does not require a densely measured datum surface (although we always recommend that you measure datum surfaces densely).

With surface data, the datum math type is available. For the **DEFAULT** datum math type (and the **CL2** datum math type under ISO), with a maximum material modifier, PC-DMIS computes the surface envelope externally to the material. It uses constrained least squares (it is a mating envelope). With a minimum material modifier, the surface envelope

is internal to the material, but PC-DMIS still uses constrained least squares (it is a minimum material envelope). ISO datums with the **DEFAULT** or **CL2** datum math option void-filter the surface before the fit, but ASME datums do not. For the **LSQ** datum math type, regardless of the material modifier, the surface envelope uses plain unfiltered least squares.

When there is no surface data, the MEAS feature is used as the surface envelope. See the "Feature Types With and Without Surface Data" topic for lists of the feature types that do not have surface data. As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using datum features without surface data, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards.

The rules to compute the size of the material boundary are complicated; please see "Determining the Size of the Material Boundary". A print rarely specifies a material boundary size. When specified, it overrides the rules that compute the size of the material boundary. The geometric tolerance command supports this: first you click **Advanced Modifiers**, and then you type a material boundary size.

In order for PC-DMIS to correctly determine the size of the material boundary, you must create the size tolerances and geometric tolerances on datums *before* you allow any geometric tolerances to reference those datums. That is, the tolerances on the datum must be earlier in the measurement routine than the geometric tolerances that reference the datum.

If you later edit any size tolerance on a datum, you must ensure that all subsequent geometric tolerances that reference that datum have the correct size tolerance information for the datum.

Once PC-DMIS compute the surface envelope and the size of the material boundary, the size of the zone is the difference between the surface envelope size and the size of the material boundary:

For inner features with a maximum material modifier and for outer features with a minimum material modifier, this is the surface envelope size minus the material boundary size.

For outer features with a maximum material modifier and for inner features with a minimum material modifier, this is the material boundary size minus the surface envelope size.

The datum surface envelope axis must stay in the zone for all lower-precedence datums and for tolerance zone calculations that use this datum. However, the axis is not optimized in the zone. The axis is only required to be in the zone.

A zone size of zero or negative implies that the datum violates its size tolerance. In the functional gage world, in a case like this, the gage's pin would not fit in its actual part hole. In this case, the geometric tolerance command does not fail the position or profile tolerance just because the datum is out of tolerance, but it lets the datum's size tolerance fail the datum. Instead of failing the position or profile tolerance, the datum is re-evaluated without a material modifier.

In some cases, a secondary or tertiary datum with a material modifier has translation available to it. This is always true under ISO and is also the case under ASME when a translation modifier is present. In this case, the tolerance zone is allowed to move relative to higher-precedence datums until it optimally contains the surface envelope axis. After that, its position is fixed relative to higher precedence datums. The orientation of the zone is still nominal relative to higher precedence datums.

The extent of the "axis of the mating envelope" is defined in the following way:

For datum spheres, it is the center-point of the surface envelope.

For datum cylinders, the axis of the surface envelope, is extrapolated to the end-faces in the same way as the toleranced feature is extrapolated. For information, see "Deriving the Toleranced Feature".

For datum circles, it is the center-point of the surface envelope.

For datum widths, the measured surface points are projected to the center-plane of the surface envelope, and the axis of the mating envelope is the minimal convex polygon that contains all the projected points.

For datum slots, the measured mid-line is extrapolated to the length of the slot.

For datum notches, the measured mid-line is extrapolated to the width of the notch.

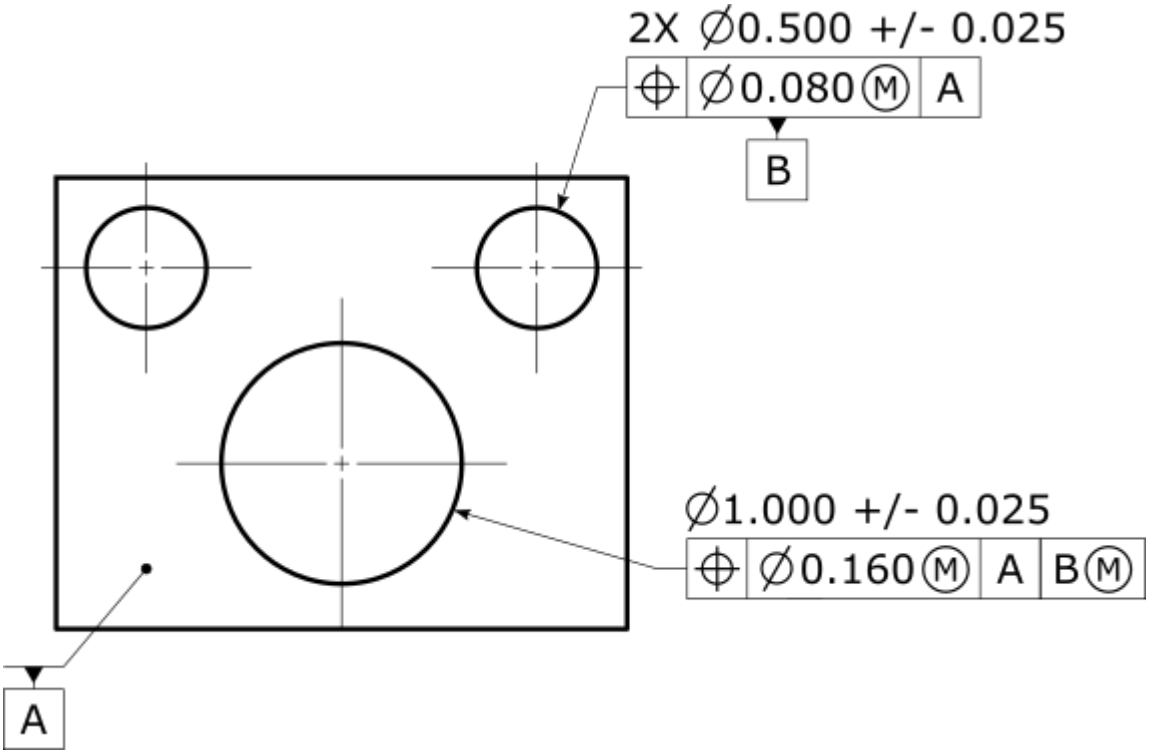
Datum patterns and common datums of coaxial cylinders are also allowed to have a material modifier. In this case, each surface envelope and zone size is calculated independently. This maximizes the accuracy of the axis-in-zone approximation. The zones are nominally oriented and located with respect to each other.

When translation is not available (ASME without a translation modifier), the zones are nominally oriented and located with respect to higher-precedence datums.

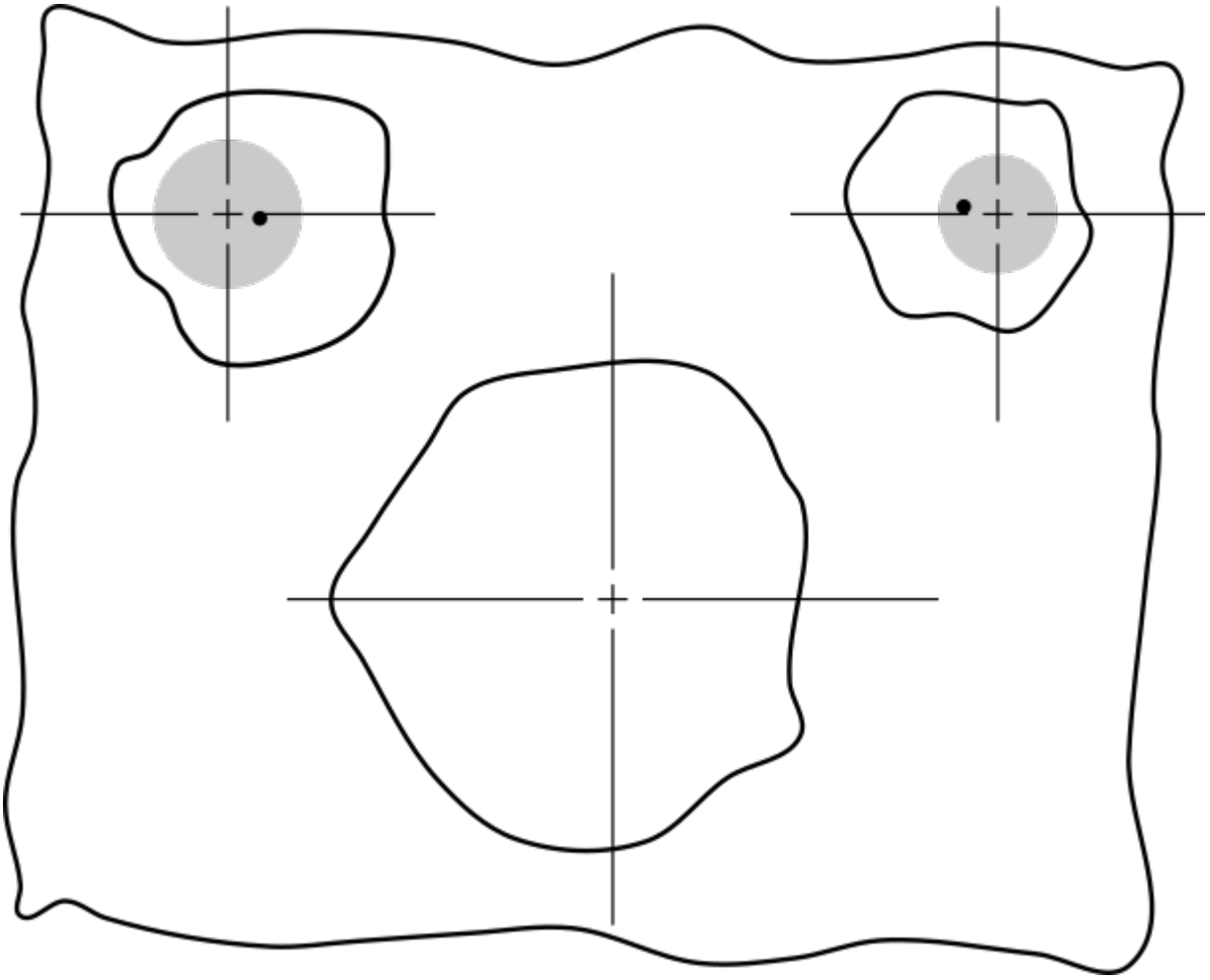
When translation is available (whether ASME with a translation modifier or ISO), the tolerance zones may translate together until they optimally contain the surface envelope axes. However, the zones remain nominally oriented and located with respect to each other, and they remain nominally oriented to higher precedence datums.

Illustrations of Datums with a Material Modifier

Suppose you had the following specification, using a secondary datum pattern referenced at maximum material boundary (MMB):

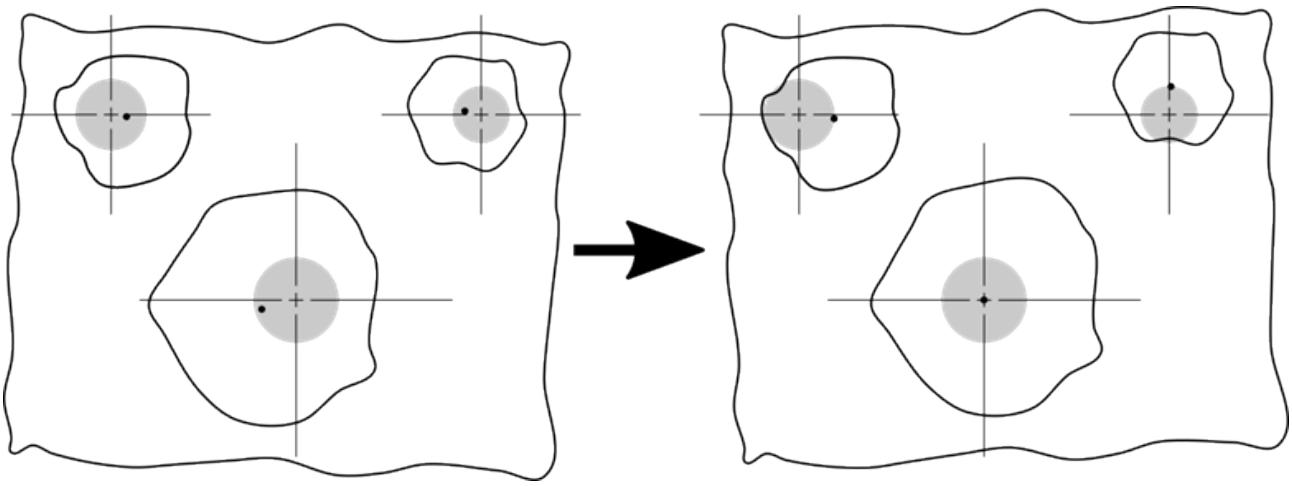


Solving the datum reference frame might result in this illustration:



The solid lines are the actual surface, the gray shaded areas represent the datum zones, and the small dots represent the datum axis that must be in the zone. The left datum has a bigger zone because the left actual datum hole is larger. The sizes of the zones are exaggerated to show how things work.

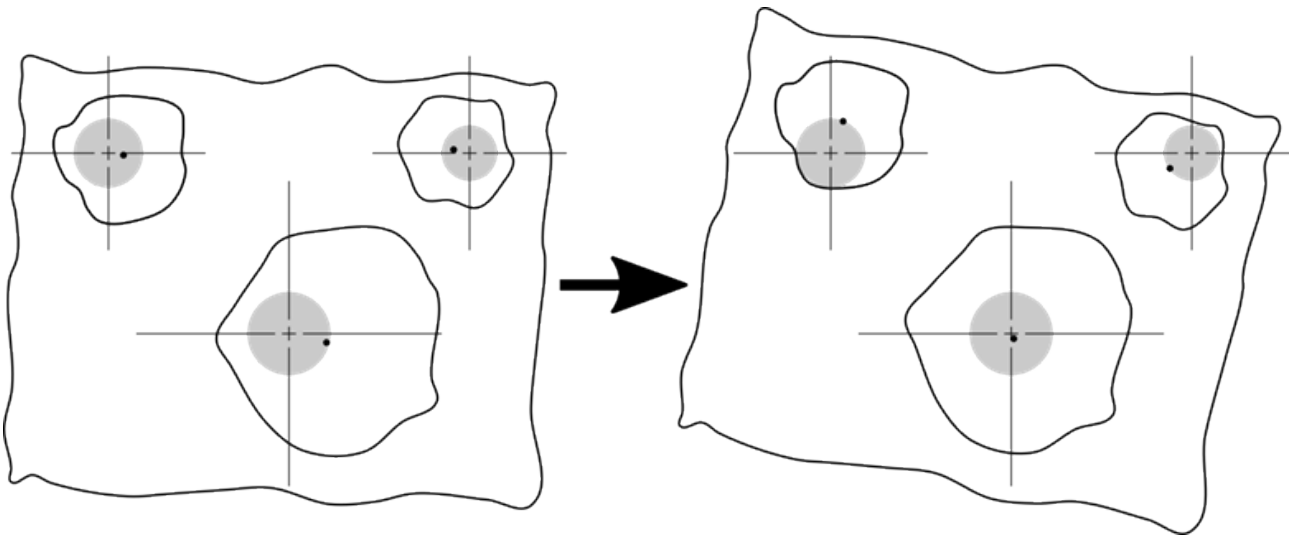
Since the datum axes are allowed to move within the datum zones (but not go outside them), that means the actual value for the position tolerance can be zero. Here is an illustration of how this works:



On the left is before optimization, showing the axis and tolerance zone of the big hole. On the right is after optimization, where the axis of the big hole is optimized to perfect location (yielding zero actual value) while the datum axes are forced to remain inside their zone but are not optimized.

The surfaces are shown to aid understanding, but they do not participate in the optimization. The optimization only uses the zones and the axes.

However, if the big hole has a large enough position error, the measured value will not be zero. Below is an illustration of such a case:



The big hole's actual value is optimized to be as small as possible, but the datums need to stay in their zones and so the actual value isn't zero.

Determining the Size of the Material Boundary

When you reference a datum with a material boundary modifier, the geometric tolerance command needs to compute the size of the material boundary unless the material

boundary size is specified. The material boundary size is based on the size tolerance of the datum feature and the applicable geometric tolerances.

For datums with a maximum material modifier, you start with the maximum material condition size. This is the upper limit of size for outer features, and the lower limit of size for inner features. Then you adjust this size by the applicable geometric tolerance.

For datums with a least material modifier, you start with the least material condition size. This is the lower limit of size for outer features, and the upper limit of size for inner features. Then you adjust this size by the applicable geometric tolerance.

For material boundaries that contain the feature surface, the applicable geometric tolerance increases the size of the material boundary. This is for outer features with a maximum material modifier and for inner features with a least material modifier.

For material boundaries that are contained by the feature surface, the material boundary size is decreased by the applicable geometric tolerance. This is for inner features with a maximum material modifier and for outer features with a least material modifier.

Rules to Determine Geometric Tolerance

The rules that determine which geometric tolerance to apply, if any, are complicated. They are described in ISO 2692:2014 (section 4) and described in ASME Y14.5-2018 (section 7.9). The PC-DMIS rules are shown below. In the ISO case, the rules are adapted to the tolerance types that PC-DMIS supports.

Primary Datums Under ISO 1101

For primary datums with ISO, we adjust the material boundary by the form tolerance at the same material condition, if it exists:

If the primary datum is a cylinder, we use the axis straightness at the same material condition, if it exists.

If the primary datum is a sphere or width, we ignore the form tolerance and do not adjust the material boundary.

If the primary datum is A Ⓜ and datum cylinder A is a shaft and has an axis straightness at Ⓜ, the maximum material boundary (MMB) is the maximum size of the shaft plus the axis straightness tolerance.

Secondary Datums Under ISO 1101

For secondary datums with ISO, we adjust the material boundary by the orientation or location tolerance at the same material condition that references the same primary datum, with the same datum modifiers, and that does not have any secondary datum. All other tolerances are ignored. If more than one such tolerance exists, the material boundary adjustment uses the smallest such tolerance.

Tertiary Datums Under ISO 1101

For tertiary datums with ISO, we adjust the material boundary by the orientation or location tolerance at the same material condition that references the same primary datum, with the same datum modifiers, and that references the same secondary datum, with the same datum modifiers, and that does not have any tertiary datum. All other tolerances are ignored. If more than one such tolerance exists, the material boundary adjustment uses the smallest such tolerance.

Primary Datums Under ASME Y14.5

For primary datums with ASME, we adjust the material boundary by the axis straightness form tolerance, if it exists. We do not adjust the material boundary for spheres and widths.

If the primary datum is A Ⓜ and datum cylinder A is a shaft and has an axis straightness, the maximum material boundary is the maximum size of the shaft plus the axis straightness tolerance.

In rare cases, a pattern is used as a primary datum with a material boundary modifier. In such cases, we first look for a position tolerance on the pattern without datums. If the position tolerance exists, PC-DMIS adjusts the material boundary by the position tolerance value. If no such position exists, then we look for an axis straightness tolerance.

Secondary Datums Under ASME Y14.5

For secondary datums with ASME, we adjust the material boundary by the orientation or position tolerance on the secondary datum.

- for a single datum, we use the orientation tolerance of the datum to the primary datum
- if no such orientation tolerance exists, we use the position tolerance of the datum pattern to the primary datum with the same datum modifiers
- for a datum pattern, we use the position tolerance of the datum pattern to the primary datum with the same datum modifiers

- for a common datum, we use the simultaneous position tolerances of the common datum features to the primary datum with the same datum modifiers
- the only geometric tolerances that are considered are specified at the same material condition as the secondary datum in question, and have no secondary datum

Tertiary Datums Under ASME Y14.5

For tertiary datums with ASME, we adjust the material boundary by a position tolerance on the tertiary datum (not orientation). The position tolerance must reference the same primary and secondary datums, with the same datum modifiers, without any tertiary datum. The position tolerance must be at the same material condition as the tertiary datum in question. If no such position tolerance is found, the material boundary is not adjusted by any geometric tolerance. For a single datum, the position tolerance must reference that feature. For a datum pattern, the position tolerance must reference the pattern. For a common datum, there must be simultaneous position of the common datum features.

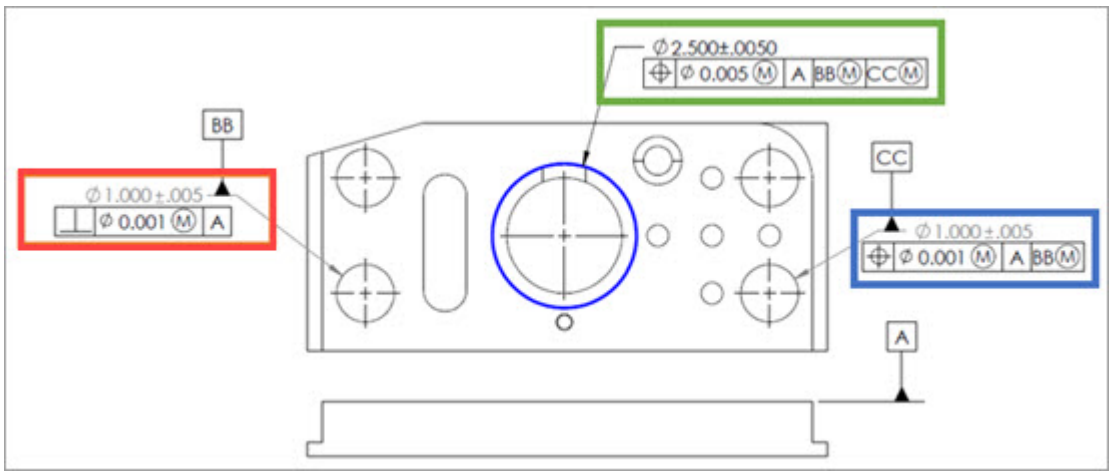
Datum Reference Messages

When you reference a datum with a modifier, PC-DMIS shows this message in the **Geometric Tolerance** dialog box:

> *Datums referenced at MMB/LMB must have been previously toleranced at MMC/LMC to their higher precedence datums. Where no such tolerance is found, zero at MMC/LMC will be applied.*

This is to remind you that the applicable Geometric Tolerance command(s) should have already been created for any datum feature(s) with a material condition modifier. If you have not created the applicable Geometric Tolerances for the referenced datum with a material condition modifier, PC-DMIS uses a Geometric Tolerance of 0.0 MMC to determine the MMB/LMB (Maximum/Least Material Boundary) value.

This reminder message cannot be turned off or disabled.



In this example, the position callout to evaluate is:

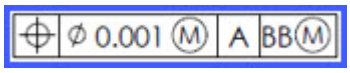
Example of the geometric tolerance for a position feature.

In addition to the feature's modifier, the secondary and tertiary datums also have modifiers.

Per the secondary datum rules, datum BB has an Applicable Geometric Tolerance to datum A:

Example of the geometric tolerance for datum BB.

Per the tertiary datum rules, datum CC has an applicable geometric tolerance to datums A and BB:

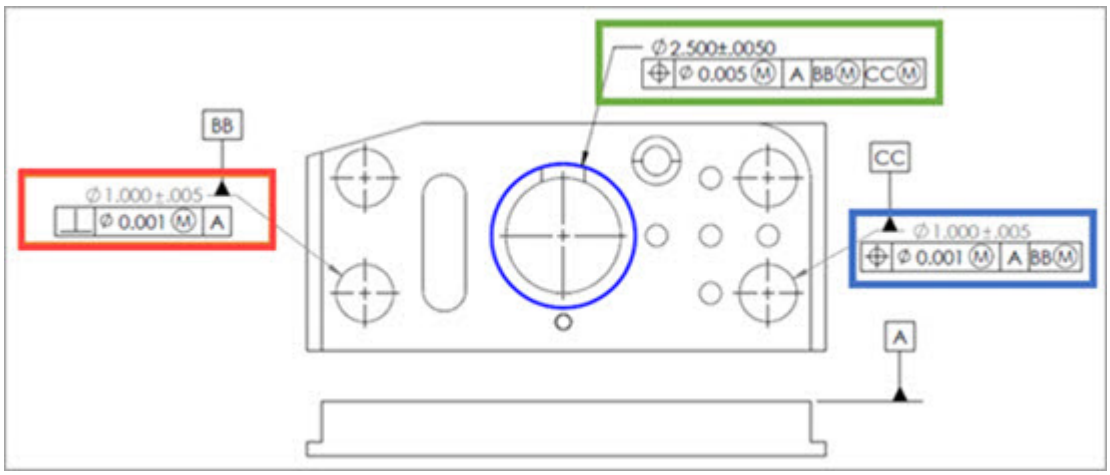


Example of the geometric tolerance for datum CC.

You must add the geometric tolerance for datum BB to the measurement routine ***first***. Once that geometric tolerance has been added, you can then add the geometric tolerance for datum CC.

Once you have defined the geometric tolerances for those datums, you can then add the geometric tolerance for the position feature to the measurement routine. PC-DMIS uses the geometric tolerances for datums BB and CC (applicable geometric tolerances) to determine the clearance (datum shift).

When you create the geometric tolerance for the position feature above for example, PC-DMIS reads up in the measurement routine to locate the applicable geometric tolerances for the datums with material condition modifiers. If PC-DMIS does not find any, it uses 0.0 @ MMC for the secondary and tertiary datums.



When PC-DMIS uses 0.0 @ MMC, in many cases you might see that PC-DMIS applies a limited datum shift. If the drawing specifies no applicable geometric tolerance, it may be a design or drawing error.

Even though PC-DMIS applies 0.0 @ MMC tolerance(s) to determine a conservative calculation for datum shifts, they do not appear in PC-DMIS reports. To confirm if the datums conform to these conservative tolerances, you can add the tolerance(s) to the measurement routine.

Using Modifiers on Datums and Their Reported Measurement Results

When you use modifiers on datums, PC-DMIS evaluates the available clearance in the datum reference frame (DRF) by using the applicable tolerances provided on the datums. When this occurs, it resolves to one of these two possible outcomes:

Result #1

The DRF **conforms** to the tolerances and PC-DMIS determines that there is clearance available in the DRF.

In this case, PC-DMIS optimizes the fit of the DRF, it applies the datum shift, and provides the measurement results.

Result #2

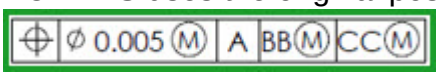
The DRF **does not conform** to the tolerances and PC-DMIS determines that there is **no clearance available** in the DRF.

If you attempt to use a functional gage for the part, **it would not fit**. This leaves PC-DMIS with two reporting options:

Report Option #1: PC-DMIS could generate a report with an error message but no measurement results.

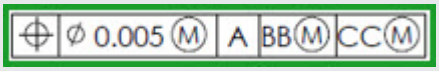
Report Option #2 : PC-DMIS could generate a report with a conservative measurement solution without the datum shift.

PC-DMIS utilizes Report Option #2 and still evaluates the Geometric Tolerance ***without*** using modifiers on the datums:

- PC-DMIS uses the original position tolerance 

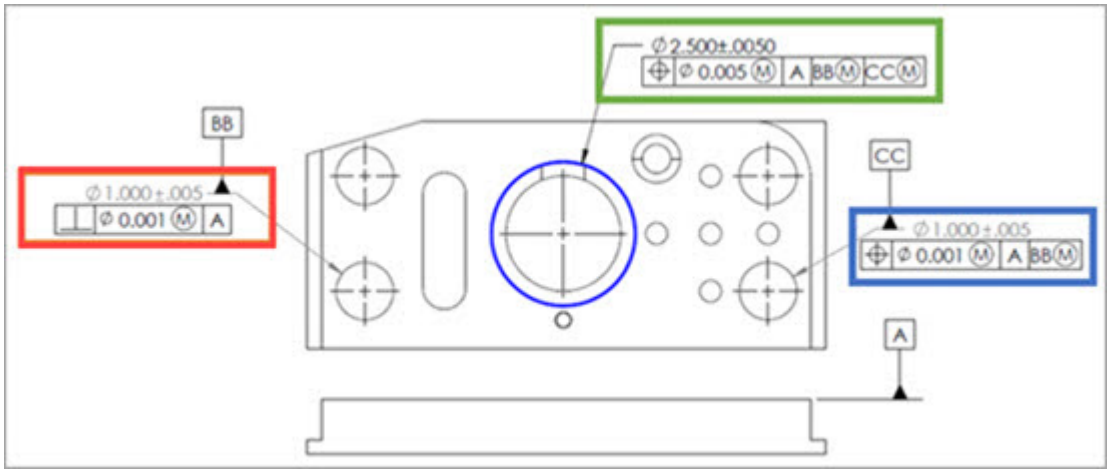
and evaluates this tolerance by removing the modifiers from the datum features. PC-DMIS would then complete the final evaluation as:

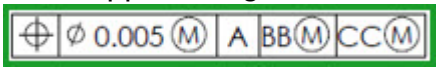
- PC-DMIS would not apply a datum shift

> PC-DMIS still indicates the original position tolerance requirement in the report:
>
> 
>

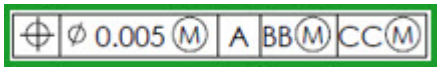
> Both results are possible even when no applicable geometric tolerances are found and PC-DMIS uses 0.0 @ MMC for the applicable geometric tolerances. For additional information, see the "Rules to Determine Geometric Tolerance" in this chapter of the PC-DMIS Core documentation.

Consider this example again:



If the datums conform to their applicable geometric tolerances, then PC-DMIS evaluates the tolerance 

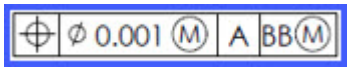
and reports the measurement results after the datum shifts as described in **Result #1** above.

However, if any of the datums do not conform to their applicable geometric tolerances, then PC-DMIS evaluates the original tolerance 

without the modifiers on any of the datums as

. PC-DMIS reports the measurement results ***without*** datum shift as described in **Result #2** above.

With Result #2, a review of the applicable geometric tolerances for the datums will show at least one of them as non-conforming. In this example, the non-conforming tolerances would either be

on datum BB or 

on datum CC.

Datums with Unconstrained Location Compared to Higher-Precedence Datums

An ASME datum with a translation modifier, or an ISO datum without a [DF] modifier, has unrestricted translation compared to higher-precedence datum simulators. Both the ASME and ISO standards are unclear about what this means when there are any datums referenced with a material modifier.

For these reasons we make the following recommendations:

- For ASME standard, when you use a modifier on a datum, we do not recommend using the translation modifier in the datum reference frame (DRF).
- For ISO standard, when you use a modifier on datum, we recommend that you always use the [DF] modifier in the DRF.

If you choose to ignore these recommendations, PC-DMIS solves the DRF as noted below and the PC-DMIS simulated gage fitting will most likely always have available clearance in the DRF.

Suppose we take a primary datum plane, a secondary datum circle at maximum material, and a tertiary datum circle without a material modifier but with translation allowed. There are two possible interpretations:

1. We could evaluate the datum reference frame independent of the position tolerance; we optimize the B-C distance so that B is optimally centered in its hole, and then we keep that distance constant while evaluating the position tolerance.
2. We could evaluate the datum reference frame simultaneously with the position tolerance; while optimizing the position tolerance we allow the B-C distance to vary until the position tolerance measured value is as small as possible.

In PC-DMIS, we use the first interpretation because it is more conservative (the measured values are larger).

Optimized Shift

Starting in PC-DMIS 2026.2, you can report optimized shift for the following position or profile tolerance cases, for both ISO 1101 and ASME Y14.5:

- Single-segment tolerances or the first segment of multi-segment tolerances whose datum reference frame (DRF) is fully constrained and includes a material condition modifier. ASME Y14.5 2018 section 7.11.11 calls this case "Datum feature Shift/ Displacement". In ISO GPS, you can consider the optimized shift as decomposition. The goal of decomposition is to translate measurement results into parameter values that you can use to adjust the manufacturing process (see ISO 20170:2019).

  The reported optimized shift information is relative to the datum reference frame's gage-centered pose (see below). The optimized shift values are always in the datum reference frame display coordinates.

- Lower segments of multi-segment tolerances.

  The reported optimized shift information is relative to the first segment's solved datum reference frame, in the datum reference frame display coordinates.

The Gage-Centered Pose

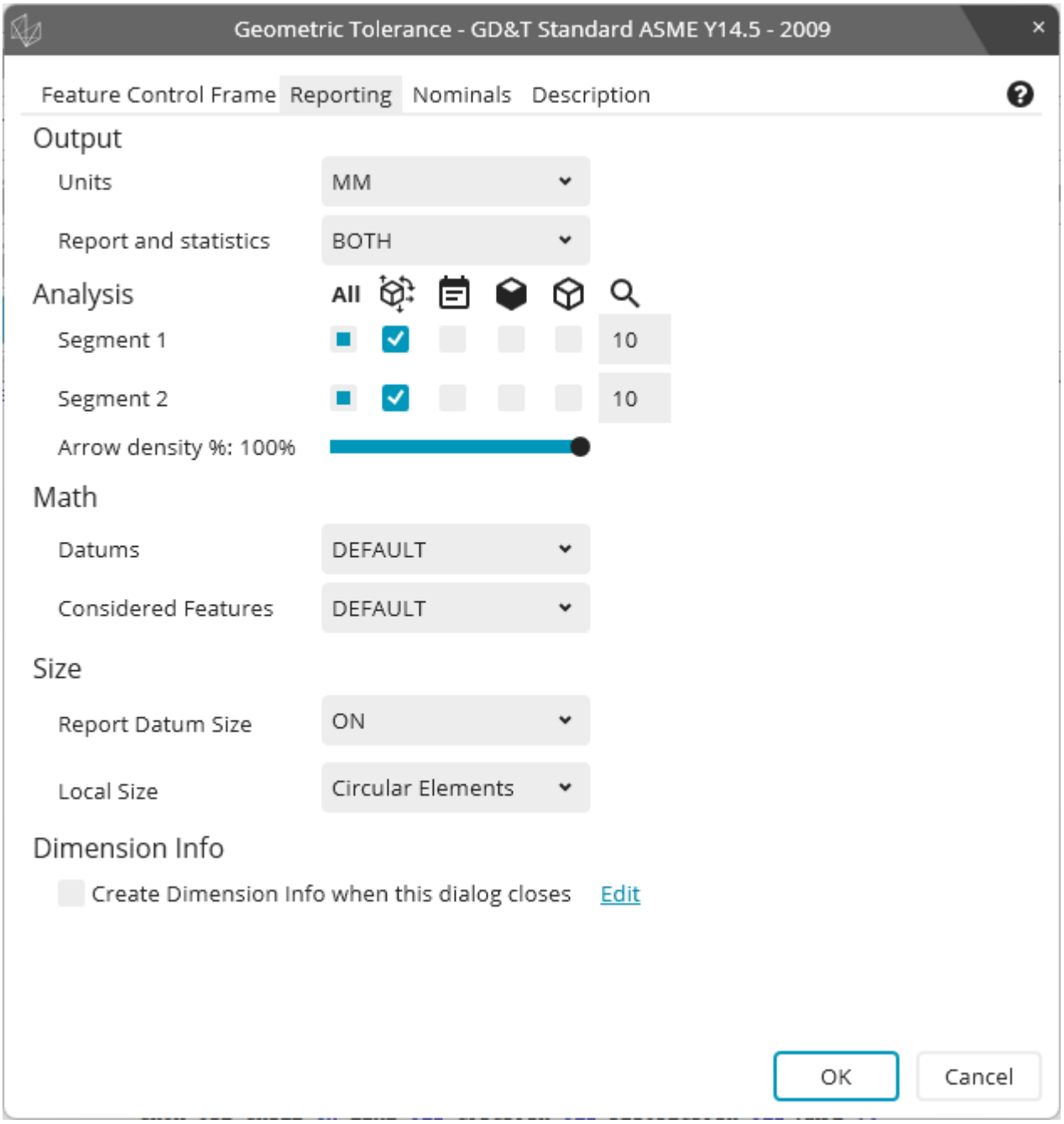
A pose is the combined position and orientation of the datum reference frame — the three translations and three rotations that appear in the report as Shift X, Shift Y, Shift Z, Rotation X, Rotation Y, and Rotation Z.

A datum feature referenced at maximum material boundary (MMB) or least material boundary (LMB) typically has play (movement) inside its gage. The gage-centered pose centers the datum features in their gages so that the gage clearance is as large as it can be, shared evenly when more than one gage competes for it.

The gage-centered pose is not the same as the pose the same datums produce with every material modifier removed, meaning all datums at regardless of material boundary (RMB) under ASME or no material requirement under ISO.

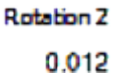
Enabling Optimized Shift

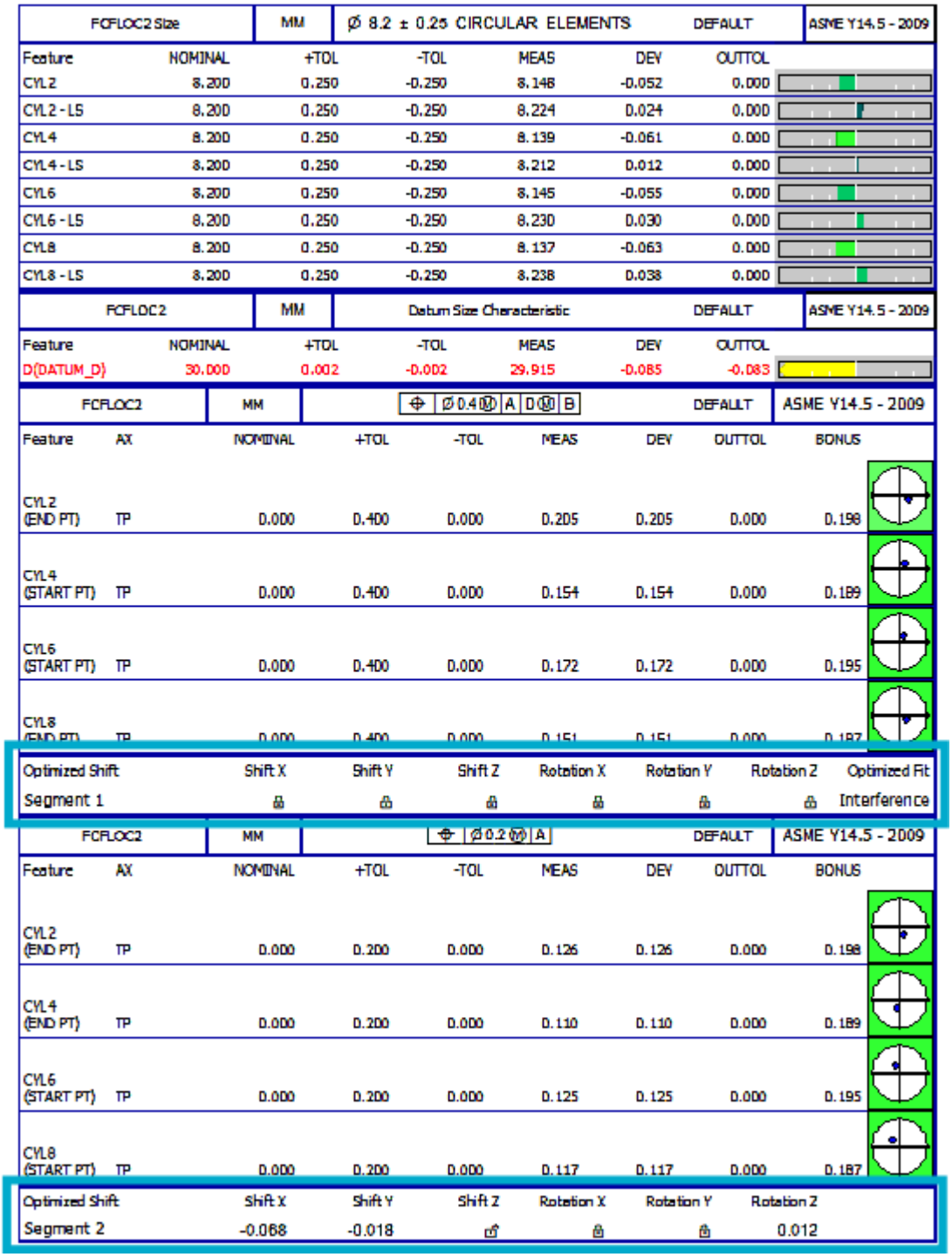
You can enable or disable optimized shift for each segment. Either select the applicable check box on the **Reporting** tab of the **Geometric Tolerance** dialog box, or toggle `SHOW_OPT_SHIFT` to `ON` in the applicable segment section of the command in Command Mode, as shown below:


Geometric Tolerance - GD&T Standard ASME Y14.5 - 2009
Feature Control Frame
Reporting
Nominals
Description
Output
Units
MM
Report and statistics
BOTH
Analysis
All
Segment 1
10
Segment 2
10
Arrow density %: 100%
Math
Datums
DEFAULT
Considered Features
DEFAULT
Size
Report Datum Size
ON
Local Size
Circular Elements
Dimension Info
Create Dimension Info when this dialog closes
Edit
OK
Cancel

```
FCFLOC2     =GEOMETRIC_TOLERANCE/STANDARD=ASME Y14.5 - 2009,SHOWEXPANDED=YES,
             DESCRIPTION=OFF,,
             FEATURE_MATH=DEFAULT,DATUM_MATH=DEFAULT,DISPLAY_COORDS=DRF,
             UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
             SIZE/NOMINAL=8.2,UPPER TOLERANCE=0.25,LOWER TOLERANCE=-0.25,
             REPORT_LOCAL_SIZE=ON,LOCAL_SIZE_METHOD=CIRCULAR_ELEMENTS,
               CYL1:
                 UAME SIZE:8.123,
                 LOCAL SIZE:8.220,
               CYL2:
                 UAME SIZE:8.148,
                 LOCAL SIZE:8.224,
               CYL3:
                 UAME SIZE:8.145,
                 LOCAL SIZE:8.242,
               CYL4:
                 UAME SIZE:8.139,
                 LOCAL SIZE:8.212,
               CYL5:
                 UAME SIZE:8.139,
                 LOCAL SIZE:8.236,
               CYL6:
                 UAME SIZE:8.145,
                 LOCAL SIZE:8.230,
               CYL7:
                 UAME SIZE:8.146,
                 LOCAL SIZE:8.233,
               CYL8:
                 UAME SIZE:8.137,
                 LOCAL SIZE:8.238,
             SEGMENT 1 POSITION,DIAMETER,0.4,MMC,__,<len>,A,D,MMB,<size>,__,B,__,
             SHOW_OPT_SHIFT=ON,TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF,MULT=10,
               MEASURED:
                 CYL1:0.147,
                 CYL2:0.153,
                 CYL3:0.153,
                 CYL4:0.120,
                 CYL5:0.093,
                 CYL6:0.115,
                 CYL7:0.131,
                 CYL8:0.137,
             SEGMENT 2 POSITION,COMPOSITE,DIAMETER,0.2,MMC,__,<len>,A,<dat>,<dat>,
             SHOW_OPT_SHIFT=ON,TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF,MULT=10,
               MEASURED:
                 CYL1:0.137,
                 CYL2:0.146,
                 CYL3:0.125,
                 CYL4:0.139,
                 CYL5:0.130,
                 CYL6:0.111,
                 CYL7:0.145,
                 CYL8:0.113,
             ADD
             DATUMS/REPORTDATUMSIZE=ON,
               D(DATUM_D):NOM=30,+Tol=0.2,-Tol=-0.2,
             FEATURES/CYL1,CYL2,CYL3,CYL4,CYL5,CYL6,CYL7,CYL8,,
```

When you enable optimized shift, the report in the Report window adds different optimized shift information, depending on the circumstances:

	This symbol means that optimized shift is not available. You see this symbol for unsupported cases, or for cases where PC-DMIS cannot display meaningful information (RMB cases, for example).
	This symbol means that the axis is locked. You see this symbol when optimization is available, but the degree of freedom for this axis cannot move or shift.
	This symbol means that the axis is unlocked. You see this symbol when an axis is fully free to optimize by any amount necessary. For example, this often occurs with a lower segment of composite callouts where a degree of translation can move (Z direction), but because the cylinder vectors are in the Z direction, this has no effect on the measurement results.
	Where explicit optimized shift data is available, you see a number that represents the amount of linear shift. These numbers are in the same units as the geometric tolerance command (mm or inches).
	Where explicit optimized shift data is available, you see a number that represents the amount of angular rotation. These numbers are in the measurement routine's angular units (decimal degrees or degrees minutes and seconds) for rotations (see "Display Angle Degrees").
INTERFERENCE	This appears in the final "Optimized Fit" column for cases where any of the datum features violate their material condition boundary (MMB or LMB). It means that there is interference between the datum simulator and the actual datum feature(s) – if you check the part using a physical gage, that gage does not fit.



FCFLOC2 Size	MM	Ø 8.2 ± 0.25 CIRCULAR ELEMENTS	DEFAULT	ASME Y14.5 - 2009

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2	8.200	0.250	-0.250	8.148	-0.052	0.000
CYL2 - LS	8.200	0.250	-0.250	8.224	0.024	0.000
CYL4	8.200	0.250	-0.250	8.139	-0.061	0.000
CYL4 - LS	8.200	0.250	-0.250	8.212	0.012	0.000
CYL6	8.200	0.250	-0.250	8.145	-0.055	0.000
CYL6 - LS	8.200	0.250	-0.250	8.230	0.030	0.000
CYL8	8.200	0.250	-0.250	8.137	-0.063	0.000
CYL8 - LS	8.200	0.250	-0.250	8.238	0.038	0.000

FCFLOC2	MM	Datum Size Characteristic	DEFAULT	ASME Y14.5 - 2009

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
D(DATUM_D)	30.000	0.002	-0.002	29.915	-0.085	-0.083

FCFLOC2	MM	⌖ Ø0.4Ⓜ A DⓂ B	DEFAULT	ASME Y14.5 - 2009

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL2 (END PT)	TP	0.000	0.400	0.000	0.205	0.205	0.000	0.198
CYL4 (START PT)	TP	0.000	0.400	0.000	0.154	0.154	0.000	0.189
CYL6 (START PT)	TP	0.000	0.400	0.000	0.172	0.172	0.000	0.195
CYL8 (END PT)	TP	0.000	0.400	0.000	0.151	0.151	0.000	0.187

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z	Optimized Fit
Segment 1							Interference

FCFLOC2	MM	⌖ Ø0.2Ⓜ A	DEFAULT	ASME Y14.5 - 2009

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL2 (END PT)	TP	0.000	0.200	0.000	0.126	0.126	0.000	0.198
CYL4 (END PT)	TP	0.000	0.200	0.000	0.110	0.110	0.000	0.189
CYL6 (START PT)	TP	0.000	0.200	0.000	0.125	0.125	0.000	0.195
CYL8 (START PT)	TP	0.000	0.200	0.000	0.117	0.117	0.000	0.187

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 2	-0.068	-0.018				0.012

Single-Segment Examples

Datum A is a plane (+Z), Datum E is a cylinder (+Z), and Datum CC is a plane (-X). We have a position dimension with two cylinders off location, with deviation values of 0.020 mm in two directions.

This is a fully constrained DRF, so there is no optimized shift and PC-DMIS displays the symbol:

FCFLOC1_EX1 Size	MM	Ø 16.4 ± 0.2	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL8	16.400	0.200	-0.200	16.400	0.000	0.000
CYL9	16.400	0.200	-0.200	16.400	0.000	0.000

FCFLOC1_EX1	MM	⌖ Ø0.005Ⓜ A E CC	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (START PT)	X	-0.000			-0.020	-0.020		
	Y	41.000			41.020	0.020		
	TP	0.000	0.005	0.000	0.057	0.057	0.000	0.200
CYL9 (END PT)	X	0.000			-0.020	-0.020		
	Y	-41.000			-40.980	0.020		
	TP	0.000	0.005	0.000	0.057	0.057	0.000	0.200

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 1						

However, if the print has a modifier added to Datum E ⌖ Ø0.005Ⓜ A EⓂ CC, we have the potential for optimized shift. The Geometric Tolerance command only allows optimized shift if the Datum E cylinder does not violate its MMB.

In the example below, our datum violates its MMB boundary, so the last column (Optimized Fit) appears and reports "Interference". The report also includes the datum size for clarity, which shows how the size violates the boundary. The Optimized Shift values for this case all show "locked" because we remove the MMB and report the RMB solution.

FCFLOC1_EX2 Size	MM	Ø 16.4 ± 0.2	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL8	16.400	0.200	-0.200	16.400	0.000	0.000
CYL9	16.400	0.200	-0.200	16.400	0.000	0.000

FCFLOC1_EX2	MM	Datum Size Characteristic	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
E(CYL_E1)	60.000	0.050	-0.050	59.949	-0.051	-0.001

FCFLOC1_EX2	MM	⌖ Ø0.005Ⓜ A EⓂ CC	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (END PT)	X	-0.000			-0.020	-0.020		
	Y	41.000			41.020	0.020		
	TP	0.000	0.005	0.000	0.057	0.057	0.000	0.200
CYL9 (END PT)	X	0.000			-0.020	-0.020		
	Y	-41.000			-40.980	0.020		
	TP	0.000	0.005	0.000	0.057	0.057	0.000	0.200

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z	Optimized Fit
Segment 1							Interference

Conversely, when the datum feature does not violate its material boundary, the report displays Optimized Shift values:

FCFLOC1_EX3 Size	MM	Ø 16.4 ± 0.2	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL8	16.400	0.200	-0.200	16.400	0.000	0.000
CYL9	16.400	0.200	-0.200	16.400	0.000	0.000

FCFLOC1_EX3	MM	Datum Size Characteristic	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
E(CYL_E1)	60.000	0.050	-0.050	59.960	-0.040	0.000

FCFLOC1_EX3	MM	⌖ Ø0.005Ⓜ A EⓂ CC	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (START PT)	X	-0.000			-0.016	-0.016		
	Y	41.000			41.016	0.016		
	TP	0.000	0.005	0.000	0.047	0.047	0.000	0.200
CYL9 (END PT)	X	0.000			-0.016	-0.016		
	Y	-41.000			-40.984	0.016		
	TP	0.000	0.005	0.000	0.047	0.047	0.000	0.200

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 1	0.004	-0.004	🔒	🔒	🔒	🔒

If sufficient clearance is available (allowing more shift), the measurement results can be zero:

FCFLOC1_EX4 Size	MM	Ø 16.4 ± 0.2	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL8	16.400	0.200	-0.200	16.400	0.000	0.000
CYL9	16.400	0.200	-0.200	16.400	0.000	0.000

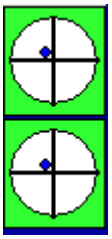
FCFLOC1_EX4	MM	⌖ Ø0.005Ⓜ A EⓂ CC	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (END PT)	X	-0.000			0.000	0.000		
	Y	41.000			41.000	-0.000		
	TP	0.000	0.005	0.000	0.000	0.000	0.000	0.200
CYL9 (END PT)	X	0.000			-0.000	-0.000		
	Y	-41.000			-41.000	-0.000		
	TP	0.000	0.005	0.000	0.000	0.000	0.000	0.200

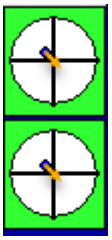
Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 1	0.020	-0.020	🔒	🔒	🔒	🔒

Direction of the Reported Shift Values

You can use different perspectives when reporting the Optimized Shift values. PC-DMIS takes the approach of having the simulated gage fixed in place and the part moving. As a result, if you look at the original position values (Dim FCFLOC_EX1) for CYL8 and CYL9, the position error is in this direction (viewing from +Z):



As we gain clearance, the part moves to the right and down on the simulated gage (see yellow arrows):



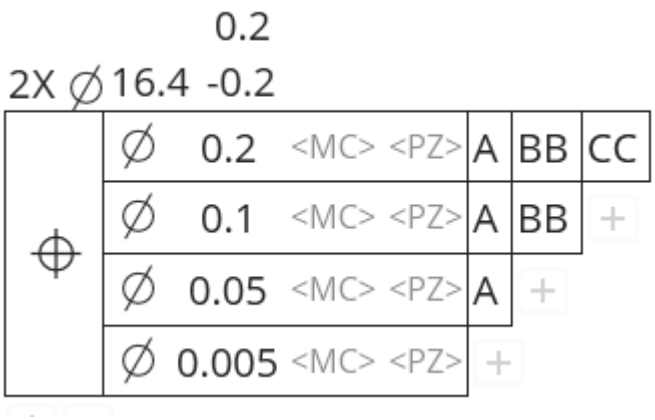
Our final result is a position whose deviation values are exactly at 0.0, with Optimized Shift values of X +0.020, Y -0.020.

When there is a modifier on the datum (MMB or LMB), PC-DMIS reports the optimized shift this way because the datum feature has play (movement) inside its gage. The starting point (coordinate system) is always the gage-centered pose.

Composite Callout Examples

Composite requirements for position and profile can get complicated. The example below shows datums without modifiers. If your first segment has MMB or LMB modifiers, PC-DMIS reports the optimized shift for that segment similar to the single segment considered above. Some of the report screenshots below show only a partial section of the entire report. The shift values reported in any lower segments compare the DRF for that segment to the DRF in the first segment.

Datum A is a plane (+Z), Datum BB is a plane (-Y), and Datum CC is a plane (-X). Consider the overall requirement for this example:



FCFLOC1_EX5 Size	MM	⌀ 16.4 ± 0.2	DEFAULT	ASME Y14.5 - 2018

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL8	16.400	0.200	-0.200	16.400	0.000	0.000
CYL9	16.400	0.200	-0.200	16.400	0.000	0.000

FCFLOC1_EX5	MM	⌖ ⌀0.2 A BB CC	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (START PT)	X	80.000			79.980	-0.020		
	Y	101.000			101.020	0.020		
	TP	0.000	0.200	0.000	0.057	0.057	0.000	0.000
CYL9 (END PT)	X	80.000			79.980	-0.020		
	Y	19.000			19.020	0.020		
	TP	0.000	0.200	0.000	0.057	0.057	0.000	0.000

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 1						

For Segment 1, PC-DMIS does not allow or report optimized shift.

FCFLOC1_EX5	MM	⌖ ⌀0.1 A BB	DEFAULT	ASME Y14.5 - 2018

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (END PT)	X	80.000			80.000	0.000		
	Y	101.000			101.000	0.000		
	TP	0.000	0.100	0.000	0.000	0.000	0.000	0.000
CYL9 (END PT)	X	80.000			80.000	-0.000		
	Y	19.000			19.000	0.000		
	TP	0.000	0.100	0.000	0.000	0.000	0.000	0.000

Optimized Shift	Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z
Segment 2	0.020	-0.020				

For Segment 2:

- The datums lock and control the rotational degrees of freedom
- PC-DMIS does not report a specific Shift Z because it can move infinitely for this case . It has no effect on the position calculation because the cylinder vectors are in the Z direction.

- SHIFT X and SHIFT Y have reported values. The shift values compare the DRF from Segment 2 to the DRF on Segment 1.

FCFLOC1_EX5	MM	⌖ Ø0.05 A			DEFAULT	ASME Y14.5 - 2018		
Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (END PT)	X	80.000			80.000	0.000		
	Y	101.000			101.000	-0.000		
	TP	0.000	0.050	0.000	0.000	0.000	0.000	0.000
CYL9 (END PT)	X	80.000			80.000	-0.000		
	Y	19.000			19.000	-0.000		
	TP	0.000	0.050	0.000	0.000	0.000	0.000	0.000
Optimized Shift		Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z	
Segment 3		0.020	-0.020				0.000	

For Segment 3:

- PC-DMIS allows the Rotation Z degree of freedom to shift. However, no rotational shift about this axis is required, so it reports zero.
- The datum locks and controls Rotation X and Y
- PC-DMIS does not report a specific Shift Z value because it can move infinitely for this case .
- SHIFT X and SHIFT Y have reported values. The shift values compare the DRF from Segment 3 to the DRF on Segment 1.

FCFLOC1_EX5	MM	⌖ Ø0.005			DEFAULT	ASME Y14.5 - 2018		
Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL8 (START PT)	X	80.000			80.000	0.000		
	Y	101.000			101.000	0.000		
	TP	0.000	0.005	0.000	0.000	0.000	0.000	0.000
CYL9 (START PT)	X	80.000			80.000	0.000		
	Y	19.000			19.000	0.000		
	TP	0.000	0.005	0.000	0.000	0.000	0.000	0.000
Optimized Shift		Shift X	Shift Y	Shift Z	Rotation X	Rotation Y	Rotation Z	
Segment 4		0.020	-0.020		0.000	0.000	0.000	

For Segment 4:

- PC-DMIS allows the Rotation X, Y, and Z degrees of freedom to shift. However, no rotational shift about these axes is required, so they all report zero.
- PC-DMIS does not report a specific Shift Z value because it can move infinitely for this case .
- SHIFT X and SHIFT Y have reported values. The shift values compare the DRF from Segment 4 to the DRF on Segment 1.

Feature Types With and Without Surface Data

In this Topic

[Feature Types With and Without Surface Data](#)
- [Introduction](#)
- [Planes](#)
- [Lines](#)
- [Points](#)
- [Cylinders](#)
- [Circles](#)
- [Widths](#)
- [Slots and Notches](#)
- [Cones](#)
- [Spheres](#)
- [Freeform Feature Types](#)
- [Reverse Features](#)

Introduction

The geometric tolerance command has detailed rules for feature types with surface data, for features types without surface data, and for features types that you cannot use in a geometric tolerance command at all. Unfortunately, some feature types are only allowed to be used with a few tolerance types, and that information is not detailed here. Also, there are a few exceptions to the rules detailed here that were omitted for clarity but are rare enough that we did not document them.

There are several types of features that you can use with the geometric tolerance command:

- Plane
- Cross section of a plane (surface line)
- Sample of a plane (surface point)
- Cylinder
- Cross section of a cylinder (surface circle)
- Surfaceless axis
- Width
- Cone
- Sphere
- Surfaceless 3D point

- Freeform

The documentation below talks about each PC-DMIS feature type and illustrates the kinds of PC-DMIS features that belong to the geometric tolerance command types.

For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points).

For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Planes

With Surface Data

Plane feature types with surface data include:

Measured planes, auto planes (any measurement strategy), constructed BF planes, and constructed BFRE planes

No Surface Data

Plane feature types without surface data include:

Alignment planes, cast planes, mid-planes, perpendicular planes, parallel planes, primary datum planes, offset planes, translated planes, and generic planes

Only Allowed for Datums

Some plane types you can use only as datums, but not as considered features. These features include:

Alignment planes, cast planes, perpendicular planes, parallel planes, primary datum planes, offset planes, translated planes, and generic planes

Lines

The Geometric Tolerance dialog box has an interpretation message whenever you use a constructed line as a constructed feature or as a datum, letting you know whether the constructed line is being treated as a line on a surface (cross section of a planar surface) or as an axis of revolution (surfaceless axis).

With Surface Data

The geometric command always treats a line feature on a surface as a cross section of a planar surface. While you can measure a line on a non-planar surface in PC-DMIS, the geometric tolerance command always handles lines on surfaces as coming from a planar surface. These line types have surface data. They include these features:

Measured lines, auto lines (any measurement strategy), constructed 2D BF lines, constructed BFRE lines, and scan segment lines

Constructed 3D BFRE lines frequently have incorrect nominal values, even when you program your routine from a CAD model. Specifically, the nominal workplane of the constructed 3D BFRE line contains both the nominal surface normal and the nominal line vector. It is often nominally misaligned with the drawing. If you use this as a datum, it can frequently cause the following error message:

"Datum feature <feature name> is 2D. It needs a higher-precedence datum to constrain its workplane."

In most cases, we recommend using constructed 2D BFRE lines instead of constructed 3D BFRE lines so the nominal workplane of the line is correct.

For information on how PC-DMIS handles line surface vectors for constructed 2D and 3D lines, see the "Constructed Line Surface Vectors" section of the "Migration from Prior Versions of PC-DMIS" topic of the PC-DMIS Core documentation.

No Surface Data

Some line features are cross sections of a plane, but they do not have surface data. These include:

Mid-lines, projected lines, and secondary datum lines

Some line features are surfaceless axes. These represent some axis of revolution and do not have a surface. These do not have surface data. These line features include:

Alignment lines, constructed 3D BF lines, cast lines, intersection lines, parallel lines, perpendicular lines, offset lines, and generic lines

Special Cases

Some line features deviate from the above interpretation in these special cases:

- Constructed 3D BF lines and offset lines are cross sections of a plane and have surface data when they are constructed from points measured on a surface.
- Parallel lines and perpendicular lines are cross sections of a plane, but do not have surface data when the resulting nominal surface normal matches the location reference (the second input to the construction).

Points

With Surface Data

A point feature on a surface is always treated as a sample of a planar surface. While you can measure a point on a non-planar surface in PC-DMIS, the geometric tolerance command always handles points on surfaces as coming from a planar surface. These point types have surface data. These point features include:

Measured points, vector points (default strategy, not the self-centering one), surface points, edge points, high points, mid-points, and tertiary datum points

No Surface Data

Some point features do not have a surface and are constrained in 3 dimensions just like the center-point of a sphere. These are surfaceless 3D points. These point features include:

Cast points, corner points, drop points, generic points, intersection points, offset points, origin points, projection points, and vector distance points

A pierce point is constrained in two dimensions, so it is a surfaceless axis having a length of zero. For information on surfaceless axes, see the "Lines" above.

Not Allowed

PC-DMIS does not allow auto angle point features as inputs to the geometric tolerance command, either as considered features or as datums.

Cylinders

With Surface Data

Cylinder feature types with surface data include:

Measured cylinders, auto cylinders (any measurement strategy), constructed BF cylinders, and constructed BFRE cylinders

No Surface Data

Cylinder feature types without surface data include:

Cylinders and generic cylinders

Not Allowed

Projection cylinders are not allowed as inputs to the geometric tolerance command, neither as considered features nor datums.

Circles

With Surface Data

Circle features are treated as a cross section of a cylindrical surface. While you can measure a circle on a non-cylindrical surface in PC-DMIS, the geometric tolerance command always handles circles as coming from a cylindrical surface (with one exception detailed in "Circular Runout").

Circle feature types with surface data include:

Measured circles, auto circles (except the gage scan calibration measurement strategy), constructed BF circles, constructed BFRE circles, and scan segment circles

No Surface Data

Circle feature types that do not have surface data include:

Intersection circles, projection circles, cylinder circles, cone circles, sphere circles, cast circles, tangent 2 lines circles, tangent 3 lines circles, tangent 3 circles, scan minimum circles, and generic circles

Not Allowed

Some circle feature types are not allowed as inputs to the geometric tolerance command, neither as considered features nor datums. These include:

Auto circles with the gage scan calibration measurement strategy

Additionally, sphere circles are allowed as a considered feature but are not allowed as a datum.

Widths

All width features have surface data.

Slots and Notches

With Surface Data

Slots and notches have surface data when you use them in a profile of a line tolerance.

No Surface Data

Slots and notches do not have surface data when you use them in a position tolerance or as a datum.

Be careful with slot and notch features.

You should only use them if you already know that the form of the features is very good. If you suspect that the manufactured form error might be significant, do not use a slot or notch command. Instead, measure a scan around the perimeter of the feature and then tolerance the form, orientation, and location of the feature with a profile of a line tolerance.

Cones

With Surface Data

Cone features with surface data include:

Measured cones, auto cones (any measurement strategy), constructed BF cones, and constructed BFRE cones

No Surface Data

Cast cones represent a conical surface but do not have surface data.

Generic cones don't have a diameter in PC-DMIS, and so they represent a surfaceless axis and do not have surface data.

Not Allowed

Projection cones are not allowed as inputs to the geometric tolerance command, neither as considered features nor as datums.

Spheres

With Surface Data

Sphere features with surface data include:

Measured spheres, auto spheres, constructed BF spheres, and constructed BFRE spheres

No Surface Data

Sphere features without surface data include:

Cast spheres and generic spheres

Not Allowed

Projection spheres are not allowed as inputs to the geometric tolerance command, neither as considered features nor as datums.

Freeform Feature Types

The geometric tolerance command allows Freeform features as considered features for profile tolerances. These features have a number of measured points with matching nominal points and nominal vectors.

When there is no CAD model present, or when you don't use the iterate and re-pierce option, PC-DMIS calculates the deviations simply as the vector deviation of the measured point to the nominal planar surface defined by the nominal point and the nominal vector. This is the "piecewise planar" approximation.

When your measurement routine uses a CAD model, and you use iterate and re-pierce, PC-DMIS calculates the deviations to the CAD model (there is no piecewise planar approximation). For more information, see "Profile of a Line" and "Profile of a Surface".

PC-DMIS considers these feature types as considered freeform features:

Scan commands, constructed sets, constructed filtered sets, constructed adjust filter objects, auto Profile2D features (vision), and tori

Possible Surface Data

PC-DMIS considers some polygons as freeform features (with surface data) when you use them in a profile tolerance, but it considers them as circle features (lacking surface data) when you use them in a position tolerance. These feature types aren't really designed for position tolerances, but they are included as circles (lacking surface data) for migration purposes.

PC-DMIS considers slots and notches as freeform features (with surface data) when you use them in a profile tolerance, but it considers them as width features (lacking surface data) when you use them in a position tolerance.

Not Allowed

These features or types of features are not allowed as inputs to the geometric tolerance command, neither as considered features nor as datums:

Measured sets, area scans, projected ellipses, projected notches, projected slots, auto blobs, flush and gap features, load fixture features, face features,

object features, constructed curve features, constructed surface features, and gage features

Reverse Features

Several types of features are "reverse" features. These include:

Reverse planes, reverse cylinders, and so on

Possible Surface Data

Reverse features might have surface data. It depends on the input to the reverse feature. If the input has surface data, so does the reverse feature.

Possibly Allowed

You might be able to use reverse features as a considered feature or datum feature. It depends on the input to the reverse feature. The surface handling of the reverse feature is the same as the input feature. For example, a reverse line may be a surfaceless axis or a plane cross section. It depends on the input to the reverse feature.

Not Allowed

You cannot use reverse features that are constructed out of a different type of feature than the reverse feature as inputs to the geometric tolerance command, neither as considered features nor as datums. For example, you cannot use a reverse line constructed from a cylinder as an input to the geometric tolerance command.

Evaluating Size with the Geometric Tolerance Command

Many geometric tolerance commands contain a size tolerance. This page describes how the geometric tolerance command computes measured size values and measured local size values.

Size Specifications

The geometric tolerance command supports only a few size specifications. There is a separate size command for more complicated size specifications. For information, see "" in the PC-DMIS Core documentation.

Under ASME, the geometric tolerance command uses the following size specification. The unrelated actual mating envelope (UAME) controls the feature surface in the plus-material direction, and the local sizes control the feature surface in the minus-material direction. The local sizes are opposed-point sizes unless the feature is a cylinder, and you have the

`CIRCULAR_ELEMENTS` local size option. That said, the geometric tolerance command does not report local sizes unless you turn on local size reporting, because many measurement systems have insufficient accuracy to verify that the local sizes comply with the size tolerance.

Under ISO, starting with PC-DMIS 2025.1, size math and feature math are independent of each other (for details, see the "Comparison to Past Practice - ISO Math Types" section in the "Structuring your Measurement Routine for Geometric Tolerances" topic of the PC-DMIS Core documentation). Most ISO size tolerances are envelope sizes as defined in ISO 14405-1. This means the plus-material direction is controlled by a mating envelope size, and the minus-material direction is controlled by opposed-point local sizes. However, when ISO 17450-3 applies, as discussed in "Deriving the Toleranced Feature", then the size tolerance is a no-modifiers (default) size. This means the size tolerance does not control any mating envelope, and the only sizes are the two-point local sizes.

Global Size

Anytime the size specification includes a mating envelope size, the size tolerance has a global size. ISO tolerances call this GLOBAL SIZE in the Edit window, while ASME tolerances call this UAME in the Edit window. The only time the size tolerance does not have a global size is when ISO 17450-3 applies as discussed above, or when you select an ISO (LP) size modifier.

If your considered feature does not have surface data, the global size is the `MEAS` size of the input feature. ISO size modifiers are not available for features that do not have surface data.

If your considered feature has surface data, and your ISO size math or the ASME feature math option is **LSQ** (least squares), the global size is the size of the (unconstrained) least squares best fit.

If your considered feature has surface data, and your ISO size math or ASME feature math option is **DEFAULT**, the global size is the size of the inscribed or circumscribed best fit, whichever of the two is external to the material. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

As discussed in "Specification Versus Verification" section of the "Introduction to Geometric Tolerances and Feature Control Frames" topic in the PC-DMIS Core documentation, PC-DMIS offers both **LSQ** and **DEFAULT** feature math options because different measurement systems have different amounts of measurement uncertainty. If your measurement system is precise and accurate enough to measure the feature's form error—if the measurement uncertainty is much smaller than the form error—then it makes sense to use **DEFAULT** math. If the measurement uncertainty is larger than the form error, then you should use **LSQ** math. For more information, see "Specification Versus Verification".

Local Size

If ISO 17450-3 applies as discussed above, then the size specification is the default ISO size specification (no modifiers), meaning there are only local sizes and no global size. The geometric tolerance command reports both the maximum and the minimum local size.

If ISO 17450-3 does not apply, and if your considered feature has surface data, you have the option to report local sizes. This is primarily useful when your feature math option is **DEFAULT**, because the geometric tolerance command reports only the *worst* local size in the internal to material direction. When combined with **DEFAULT** size math, the global size controls the surface in the external to material direction while the local size controls the surface in the internal to material direction. By contrast, the **LSQ** size math has a least-squares global size, which does not control the surface deviations in either direction. In this case, the surface will be uncontrolled in the external to material direction.

For ISO size tolerances, and for ASME size tolerances on spheres and widths, the local sizes are evaluated using opposed points. Each local size is essentially a two-point caliper measurement. Please ensure your measured points all have a directly opposed point, or the measurement accuracy may suffer. This is especially challenging on spheres.

For ASME size tolerances on cylinders, you have a choice of using the `OPPOSED_POINTS` interpretation or the `CIRCULAR_ELEMENTS` interpretation. These interpretations are specified in ASME Y14.5.1 - 2019. The opposed points interpretation behaves as just described.

The circular elements interpretation requires that the surface data was measured in circular cross sections. It best-fits a circle to each cross section; the sizes of the circles are the local sizes. When the feature math type is **LSQ**, the circles are computed using least squares. When the feature math type is **DEFAULT**, the circles are inscribed or circumscribed, whichever of the two is internal to the material. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

It does not make sense to report local sizes unless your measurement system is precise and accurate enough to measure the feature's form error.

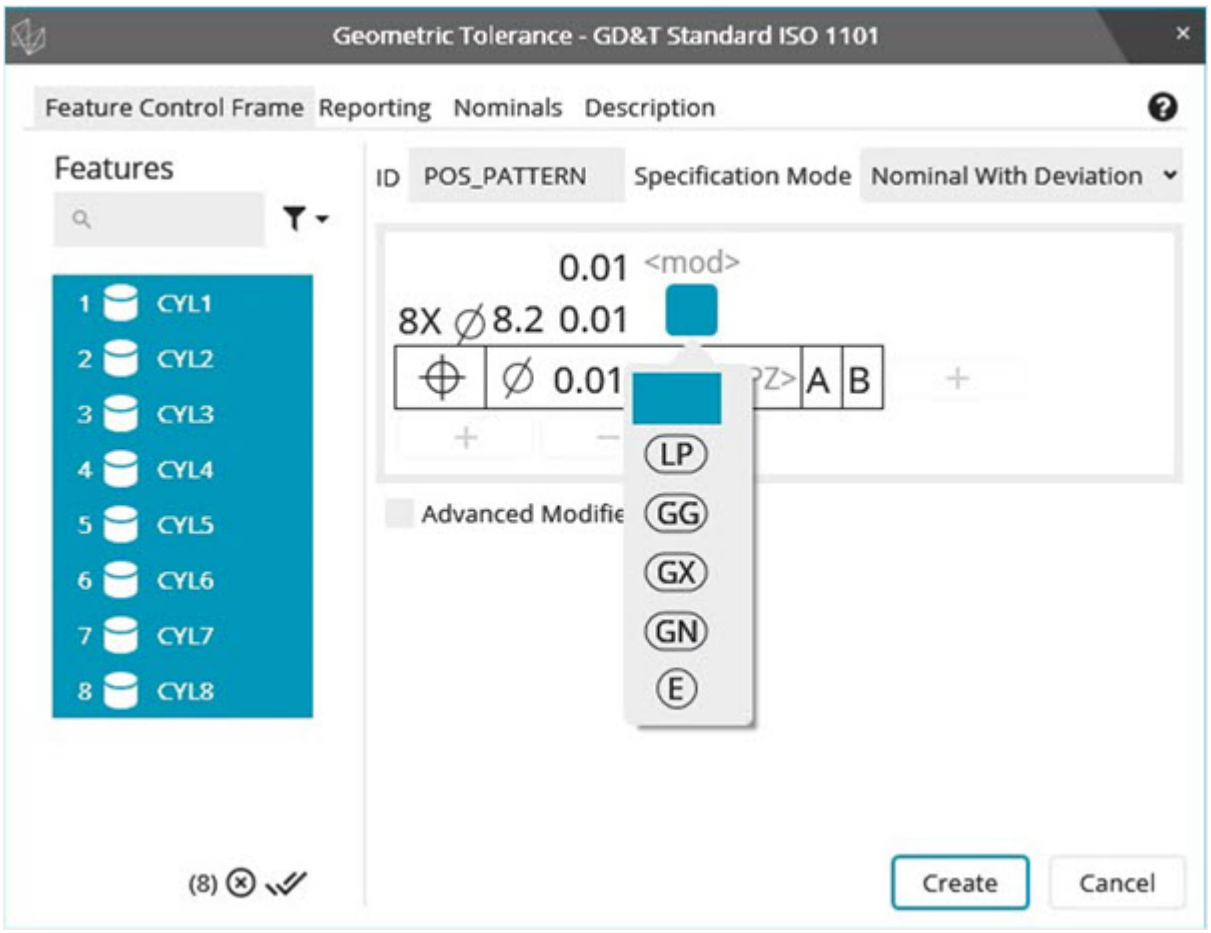
ISO Size Modifiers

Starting with PC-DMIS 2025.1, the Geometric Tolerance command supports some of the more common ISO 14405-1 size modifiers, such as:

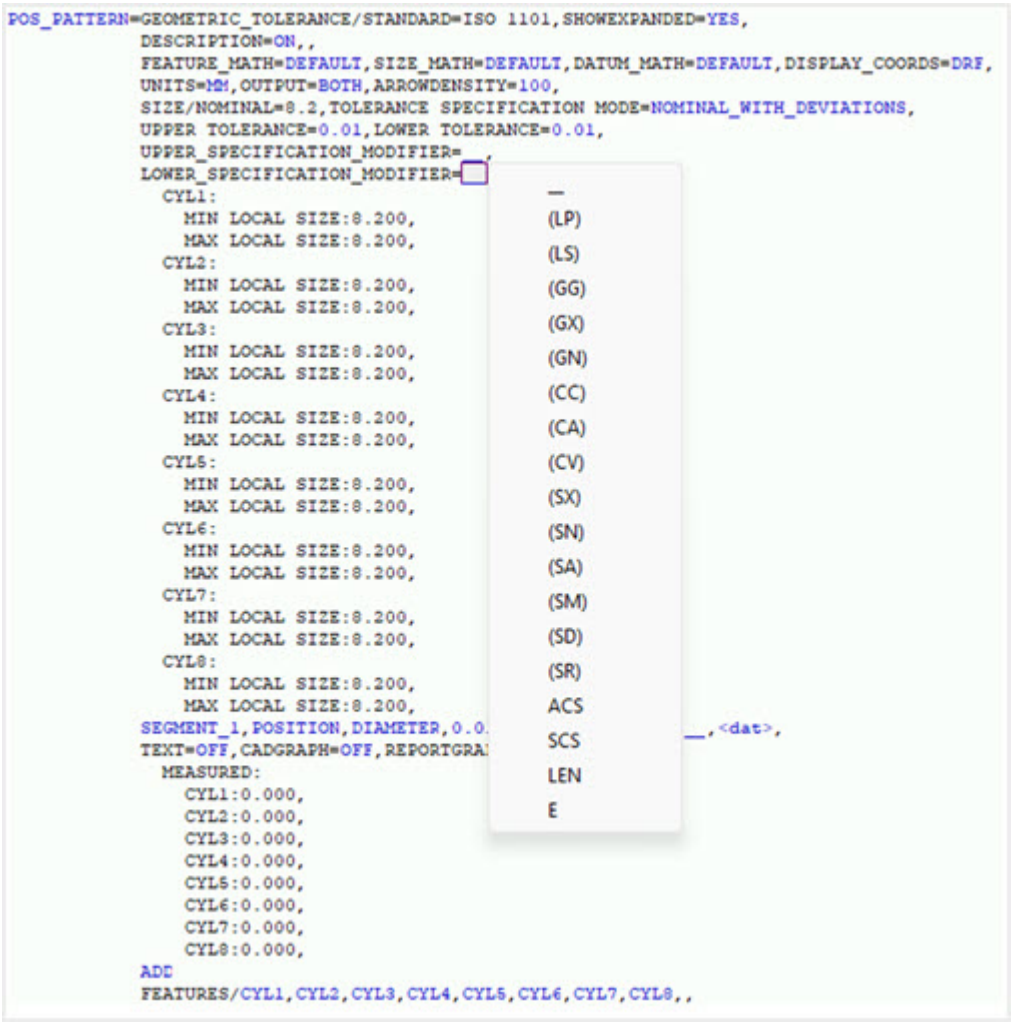
- (LP) - Two-point Size
- (GG) - Least-squares Association Criterion
- (GX) - Maximum Inscribed Association Criterion
- (GN) - Minimum Circumscribed Association Criterion

- (E) - Envelope Requirement

These modifiers are available from the **Geometric Tolerance** dialog box.



You can also add or remove size modifiers directly in the Edit window. You can either type it in or select it from a list as shown below:



Because the geometric tolerance command's Edit window `UPPER_SPECIFICATION_MODIFIER` and `LOWER_SPECIFICATION_MODIFIER` list is shared with the size command, all ISO size modifiers are shown. This doesn't mean, however, that they are all available for selection. You can only add modifiers which the geometric tolerance command supports. If you try to add an unsupported modifier, PC-DMIS ignores your selection and retains the applicable `LOWER_SPECIFICATION_MODIFIER` or `UPPER_SPECIFICATION_MODIFIER` 's existing selection.

You can see a typical example of how the information appears in the Edit window command and in the report when you select the option to report size modifiers:

```
FCFLOC1     =GEOMETRIC_TOLERANCE/STANDARD=ISO 1101,SHOWEXPANDED=YES,
             DESCRIPTION=OFF,,
             FEATURE_MATH=DEFAULT,SIZE_MATH=MODIFIER_SELECTED,DISPLAY_COORDS=DRF,
             UNITS=MM,OUTPUT=BOTH,ARROWDENSITY=100,
             SIZE/NOMINAL=20,TOLERANCE SPECIFICATION MODE=NOMINAL_WITH_DEVIATIONS,
             UPPER TOLERANCE=0.1,LOWER TOLERANCE=0.1,
             UPPER_SPECIFICATION_MODIFIER=(LP),
             LOWER_SPECIFICATION_MODIFIER=(GX),
               CYL2:
                 UPPER SIZE:20.033115,
                 LOWER SIZE:19.965135,
             SEGMENT_1,POSITION,DIAMETER,0.500039,__,__,<len>,<dat>,<dat>,<dat>,
             TEXT=OFF,CADGRAPH=OFF,REPORTGRAPH=OFF,MULT=10,
               MEASURED:
                 CYL2:0.015947,
             ADD
             FEATURES/CYL2,,
```

FCFLOC1 Size		MM	Ø 20 [+0.1 LP] - [-0.1 GX]			MODIFIER	ISO 1101
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
CYL2 - LP	20.000000	0.100000	0.100000	20.033115	0.033115	0.000000	
CYL2 - GX	20.000000	0.100000	0.100000	19.965135	-0.034865	0.000000	

FCFLOC1	MM	Ø0.500039	DEFAULT NONE	ISO 1101

Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
CYL2 (LEVEL # 1)	TP	0.000000	0.500039	0.000000	0.015947	0.015947	0.000000	0.000000

Bonus Calculations

Some geometric tolerances have a maximum material condition modifier Ⓜ (MMC) or least material condition modifier Ⓛ (LMC). This means that as the unrelated mating envelope size (or unrelated minimum material envelope size for LMC) deviates from the MMC (or LMC), additional tolerance or "bonus" tolerance is added to the tolerance in the feature control frame, yielding a total tolerance. For the **DEFAULT** feature math, the measured bonus tolerance is the difference between an inscribed or circumscribed best fit and one of the limits of size. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way. For the **LSQ** feature math, the measured bonus tolerance is the difference between the least squares global size and one of the limits of size.

Measured bonus tolerance is calculated in the following way.

- For MMC tolerances on outer features, the bonus is the upper limit of size (the MMC size) minus the circumscribed size (external to material envelope) or the least squares size (depending on the feature math type).
- For MMC tolerances on inner features, the bonus is the inscribed size (external to material envelope) or the least squares size (depending on the feature math type) minus the lower limit of size (the MMC size).
- For LMC tolerances on outer features, the bonus is the inscribed size (internal to material envelope) or the least squares size (depending on the feature math type) minus the lower limit of size (the LMC size).

- For LMC tolerances on inner features, the bonus is the upper limit of size (the LMC size) minus the circumscribed size (internal to material envelope) or the least squares size (depending on the feature math type).

In every case, the bonus is limited so it is never negative, and never exceeds the total size tolerance (the upper boundary of size minus the lower boundary of size).

The selection of ISO size modifiers does not affect the bonus calculation. Bonus values are always calculated as described above.

Report

Without Local Size

When you are not reporting local size, the size label of the report looks like this:

FCFLOC5 Size		MM	SØ 31.75 +0.025/-0.025		LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
SPH1	31.750000	0.025000	-0.025000	31.751629	0.001629	0.000000
SPH2	31.750000	0.025000	-0.025000	31.748836	-0.001164	0.000000

The header bar shows the dimension ID of the tolerance, the dimension units (MM or IN), the size specification, the math type (**LSQ** in this case), and the standard (ASME Y14.5 in this case). The table below shows the measured sizes of each sphere.

With Global and Local Sizes

When you are reporting global sizes and local sizes, the size label has extra rows, with LS added as a suffix for the worst local sizes. For example, "SPH1 - LS". For ASME tolerances on cylinders, the header bar also says whether the local size interpretation is OPPOSED or CIRCULAR. When reporting global sizes and local sizes, the report looks like this:

FCFLOC5 Size		MM	SØ 31.75 +0.025/-0.025 OPPOSED		LSQ	ASME Y14.5 2018
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
SPH1	31.750000	0.025000	-0.025000	31.751629	0.001629	0.000000
SPH1 - LS	31.750000	0.025000	-0.025000	31.731449	-0.018551	0.000000
SPH2	31.750000	0.025000	-0.025000	31.748836	-0.001164	0.000000
SPH2 - LS	31.750000	0.025000	-0.025000	31.734986	-0.015014	0.000000

Without Global Size

When ISO 17450-3 applies, there is no global size reported. Instead, MIN and MAX are added as a suffix for the worst local sizes in both directions. The report looks like this:

LOC12 Size		MM	Ø 152.4 +/- 5		DEFAULT	ISO 1101
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL2 - MIN	152.400	5.000	-5.000	152.477	0.077	0.000
CYL2 - MAX	152.400	5.000	-5.000	152.830	0.430	0.000
CYL3 - MIN	152.400	5.000	-5.000	152.490	0.090	0.000
CYL3 - MAX	152.400	5.000	-5.000	152.848	0.448	0.000

With ISO 14405 Size Modifiers

When ISO 14405 size modifiers are applied, the details are listed in the header with the appropriate suffix added to the feature name as shown below:

FCFLOC3 Size		MM	Ø 30 [+0.1 LP] - [-0.1 GX]		MODIFIER	ISO 1101
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CYL1 - LP	30.000000	0.100000	0.100000	29.957540	-0.042460	0.000000
CYL1 - GX	30.000000	0.100000	0.100000	29.919873	-0.080127	0.000000

Deriving the Toleranced Feature

In this Topic

[Deriving the Toleranced Feature](#)
- Introduction
- ISO Associated Toleranced Feature Modifiers (ATFM)
- ISO Reference Feature Association Modifiers (RFAM)
- Feature Math Types
- Spherical Features With Surface Data
- Cylindrical Features with Surface Data under ASME Y14.5
- Cylindrical Features with Surface Data under ISO 1101
- Circular Features with Surface Data under ASME Y14.5
- Circular Features with Surface Data under ISO 1101
- Conical Features With Surface Data under ASME Y14.5
- Conical Features with Surface Data under ISO 1101
- Width Features with Surface Data under ASME Y14.5
- Width Features with Surface data under ISO 1101
- Features without Surface Data
- Slots and Notches
- Tangent plane modifier

Introduction

For most specification types, the toleranced feature is the considered feature's surface data. However, with location and orientation geometric tolerances (position, concentricity, symmetry, perpendicularity, parallelism, and angularity) the toleranced feature is derived from the considered feature's surface data. This applies to circles, cones, cylinders, notches, slots, spheres, and widths. This also applies to the tangent plane modifier planar features. Each type of considered feature is handled differently. This topic covers features that have surface data, features without surface data (including mid-planes and mid-lines),

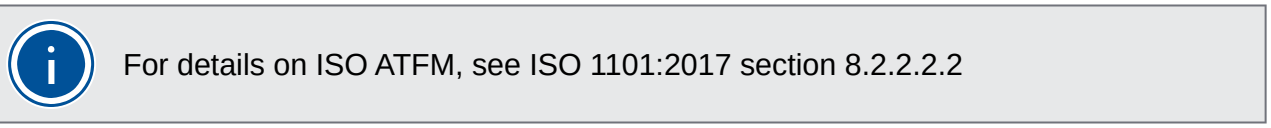
and finally the tangent plane modifier. For information on the command types that have and do not have surface data, see "Feature Types With and Without Surface Data".

In several places below, we discuss the sample plane. Circle, cone, and cylinder Auto features can have a sample plane:

- When the feature has a sample feature, the sample feature is the sample plane.
- When the feature has one sample hit, the sample plane passes through that hit and is nominally oriented to the measured datums.
- When the feature has three or more sample hits, the sample plane is the least squares plane for those sample hits.
- When the feature has no sample hits nor sample feature selected, there is no sample plane.

When there is a sample plane, there is also a start plane that intersects the axis at the start point. The nominal sample plane is allowed to be offset from the nominal start plane, because the nominal sample plane might not intersect the nominal axis at the start point. The measured start plane is parallel to the measured sample plane and is nominally offset from it.

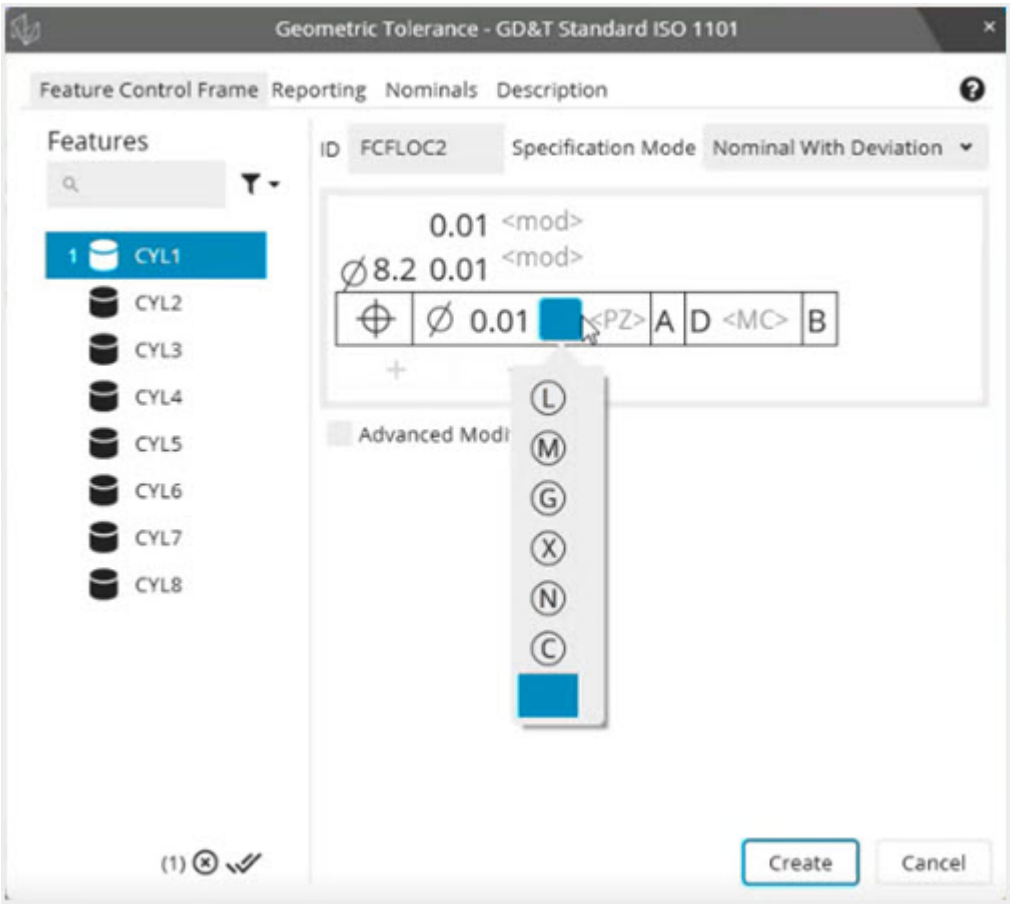
ISO Associated Toleranced Feature Modifiers (ATFM)



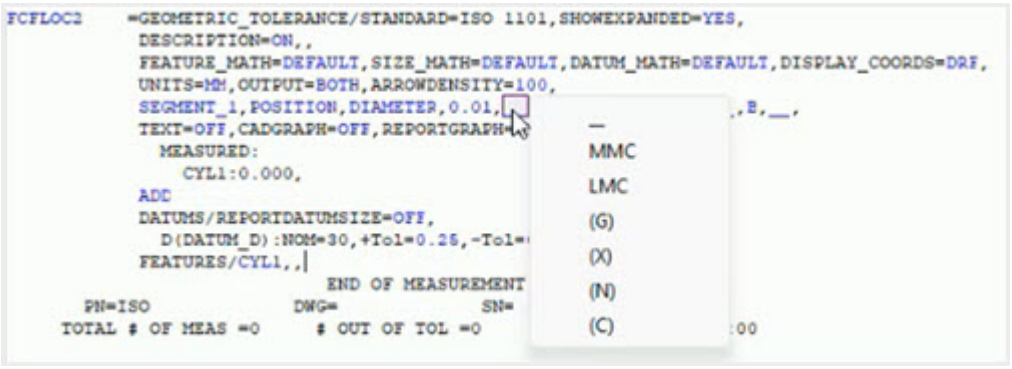
Starting with PC-DMIS 2025.1, the geometric tolerance command supports these ISO Associated Toleranced Feature Modifiers (ATFM):

- Ⓒ **MinMax (Chebyshev)** feature modifier - This feature modifier is available for circle, cone, cylinder, line, plane, and width considered feature types.
- Ⓖ **Least Squares (Gaussian)** feature modifier - This feature modifier is available for circle, cone, cylinder, line, plane, and width considered feature types.
- Ⓝ **Minimum Circumscribed** feature modifier - This feature modifier is available for circle, cylinder, line, plane, and width considered feature types.
- Ⓧ **Maximum Inscribed** feature modifier - This feature modifier is available for circle, cylinder, line, plane, and width considered feature types.
- Ⓣ **Tangent** feature modifier - This feature modifier is only available for plane considered feature types.

You can add the associated tolerance feature modifier to the tolerance section of the Feature Control Frame in the **Geometric Tolerance** dialog box.



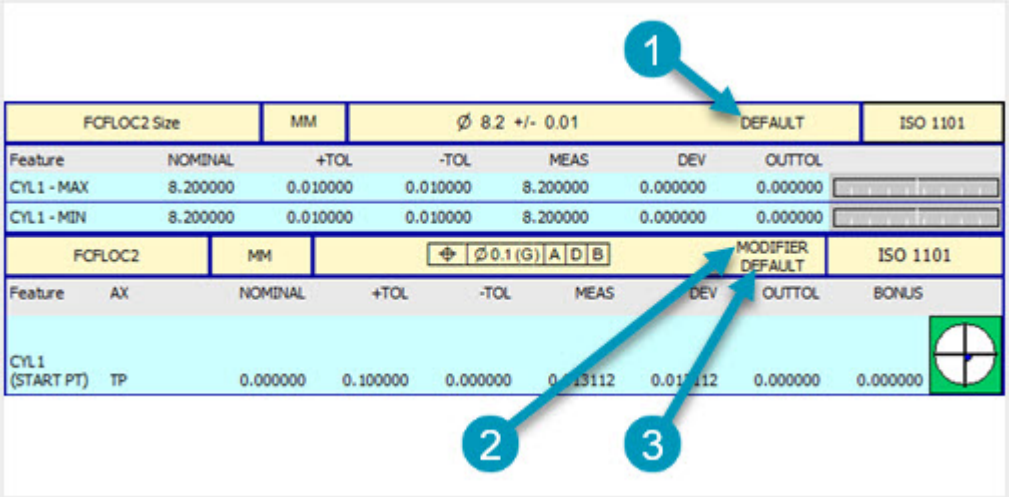
You can also select the associated tolerance feature modifier in the Edit window when in Command Mode:



When you select the modifier from the Edit window, the FEATURE_MATH shows "MODIFIER_SELECTED", and the selected modifier overrides the usual **DEFAULT** or **LSQ** feature fitting.

> **!** You cannot combine an associated toleranced feature modifier with a material condition modifier (MMC or LMC).

The resulting report shows the selected modifiers and applicable math types:



1. *SIZE Math Type*
2. *FEATURE Math Type (indicating a modifier was selected)*
3. *DATUM Math Type*

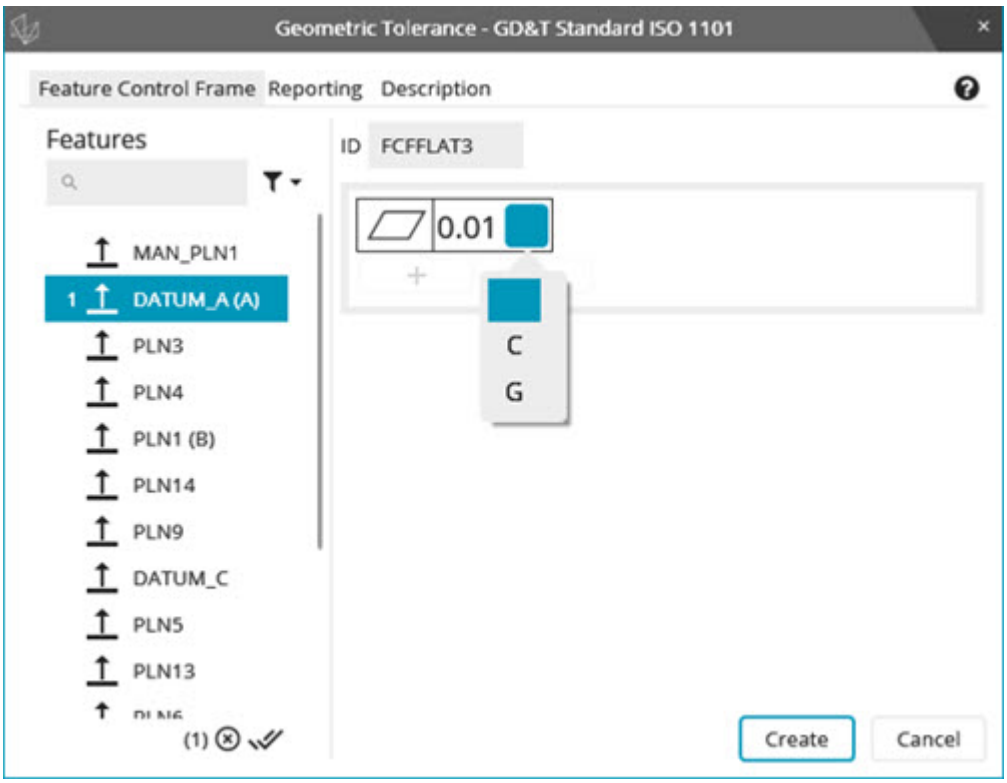
ISO Reference Feature Association Modifiers (RFAM)

> For details on the ISO Reference Feature Association Modifiers (RFAM), see ISO 1101:2017 section 8.2.2.3.1.

Starting with PC-DMIS 2025.1, the Geometric Tolerance command supports these ISO Reference Feature Association Modifiers (RFAM):

- **C** - **MinMax (Chebyshev)** feature without Constraint: This reference feature association modifier is available for circularity, cylindricity, flatness, and surface straightness tolerances.
- **G** - **Least Squares (Gaussian)** feature without Constraint: This reference feature association modifier is available for circularity, cylindricity, flatness, and surface straightness tolerances.

ISO RFAM are not applicable for straightness of an axis. You can add an RFAM to the tolerance section of the Feature Control Frame in the **Geometric Tolerance** dialog box as shown here:

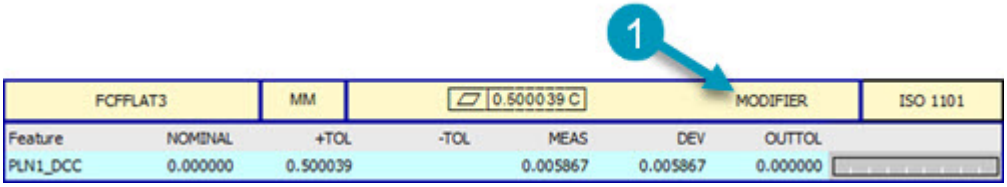


You can also select the associated RFAM in the Edit window when in Command Mode:

When you select the modifier from the Edit window, the TOLERANCE_ZONE_MATH shows "MODIFIER_SELECTED", and the selected modifier overrides the usual **DEFAULT** or **LSQ** feature fitting.

> **!** You cannot combine a reference feature modifier with a material condition modifier (MMC or LMC).

The resulting report shows the selected modifiers and applicable math types:



1. *TOLERANCE_ZONE_MATH (indicating a modifier was selected)*

Feature Math Types

As discussed in the "Specification Versus Verification" section of the "Introduction to Geometric Tolerances and Feature Control Frames" topic, we offer multiple math types for toleranced feature computation. PC-DMIS provides two such math types for features having measured surface data: **DEFAULT** and **LSQ**. What they do is detailed below. For the most part, **DEFAULT** is a good choice when the measurement uncertainty of the surface data is much less than the form error of the surface, because the mathematics is similar to the specification.

The **LSQ** feature math option does a plain least squares best fit to the surface data. This algorithm is mathematically quite different than the specification, but it is a better choice than **DEFAULT** when the measurement uncertainty of each point is much larger than the form error of the surface.

For more details, see "Specification Versus Verification".

Spherical Features With Surface Data

The toleranced feature is a 3D point when the considered feature is a sphere. When the spherical feature has surface data, the toleranced feature is constructed in the following way:

A fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and based on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Thus, the DEFAULT math type usually produces the unrelated actual mating envelope (UAME) unless the modifier is LMC. In that case, the math type produces the unrelated actual minimum material envelope (UAMME). Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

The center-point of the fitted sphere is the toleranced feature.

Cylindrical Features with Surface Data under ASME Y14.5

The toleranced feature is an axis when the considered feature is a cylinder. Under ASME Y14.5, when a cylindrical feature has surface data, the toleranced feature is constructed in the following way:

First, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is

chosen to be internal to the material. Thus, the **DEFAULT** math type usually produces the unrelated actual mating envelope (UAME) unless the modifier is LMC. In that case, the math type produces the unrelated actual minimum material envelope (UAMME). Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

Second, an extrapolation is selected. It is based on whether a sample plane is available and whether a projected-zone modifier is present:

- When there is no sample plane, and there is no projected zone modifier, the best-fit axis is extrapolated to the nominal end-faces of the cylinder. The nominal end-faces of the cylinder are nominally oriented and located to the measured datums. If there is a projected zone modifier, the extrapolation starts at the nominal start-face of the cylinder. It continues away from the end-face, until it hits the nominal projection plane parallel to the start plane at the projected distance away from it.

- When there is a sample plane, the extrapolation starts at the measured start plane.

The extrapolated best-fitted axis is the toleranced feature.

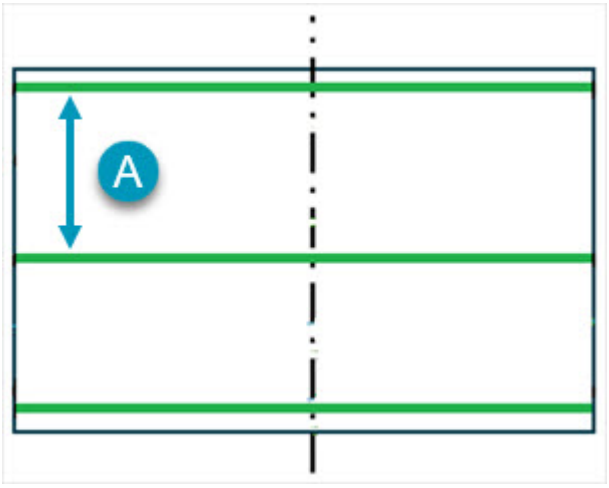
Cylindrical Features with Surface Data under ISO 1101

The toleranced feature is an axis when the considered feature is a cylinder. Under ISO 1101, when a cylindrical feature has surface data, the toleranced feature is constructed in the following way:

First, PC-DMIS determines whether ISO 17450-3 : 2016 applies or not. To PC-DMIS, it applies when there is no material modifier, no associated tolerance feature modifier, no projected-zone modifier, and the feature math type is **DEFAULT**.

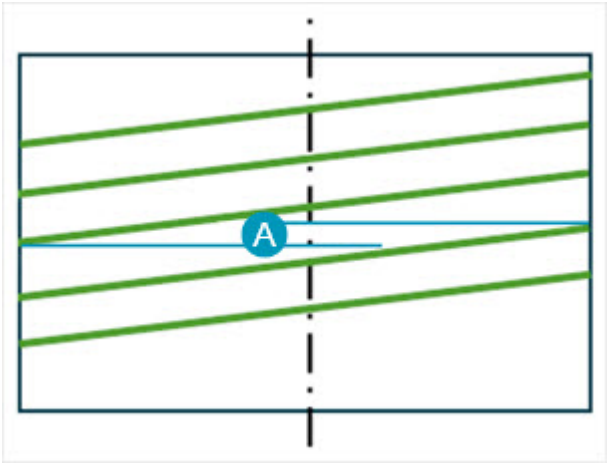
When ISO 17450-3 applies, and the surface data were measured in cross sections, the toleranced feature is an imperfect axis. Each cross section has a least-squares circle fitted to it. The vector of each circle is the vector of the least-squares axis of the entire cylinder. The center-points of the circles form the toleranced feature. This process matches closely with the specification in ISO 17450-3. When the surface data were not measured in cross sections, PC-DMIS attempts to automatically section the data in the following way:

- If the cylinder was measured in clean cross sections, meaning multiple distinct levels with an easily detectable gap between each level, then PC-DMIS uses those clean cross sections.



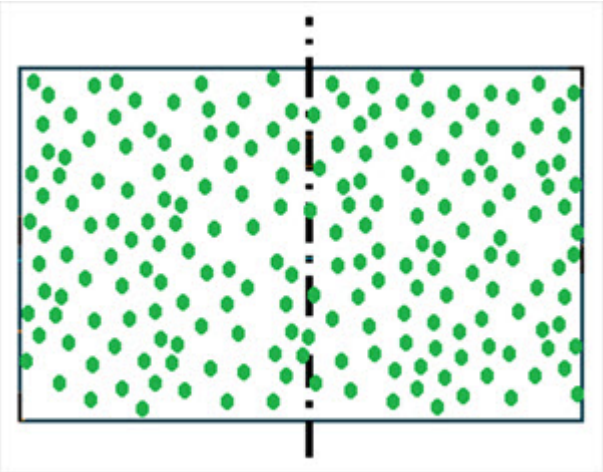
***A** - Gap*
Easily detectable gap between the end of one level and the start of the next. Three cross sections are returned.

- If the cylinder was not measured in clean cross sections, but the data follows a clean spiral pattern, then PC-DMIS divides the spiral into several cross sections, equal to the number of revolutions in the spiral (so long as the spiral has more than two revolutions).



***A** - No Gap*
You can see in this example one continuous spiral or overlapping start and end points. Five cross sections are returned.

- If the cylinder was not measured in either clean cross sections or a clean spiral pattern, then PC-DMIS performs a best effort to divide the data into cross sections.



You can see in this example that there is no discernible pattern. The number of returned cross sections depends on the point distribution and density.

After PC-DMIS divides the cylinder into cross sections, it rejects any section that has less than 90 degrees of arc. The corresponding circle center is not included in the toleranced feature.

In some cases, the above procedure produces no circle centers. For example, spiral scans with two or fewer revolutions or all cross sections were found to have less than 90 degrees of arc. In such cases, it is not possible to adhere so closely to the description in ISO 17450-3, and so PC-DMIS employs an approximation. Specifically, the toleranced feature is the axis of the least-squares cylinder, extrapolated to the endpoints of the measured surface data.

When ISO 17450-3 does not apply, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and based on the material modifier. The LSQ math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

When ISO 17450-3 does not apply, and after a fit has been computed, an extrapolation is selected. The extrapolation is based on whether a sample plane is available, and based on whether a projected-zone modifier is present:

- When there is no sample plane, and there is no projected zone modifier, the best-fit axis is extrapolated to the nominal end-faces of the cylinder. The nominal end-faces of the cylinder are nominally oriented and located to the measured datums. If there is a projected zone modifier, the extrapolation starts at the nominal start-face of the cylinder. It continues away from the end-face, until it hits the nominal projection plane parallel to the start plane at the projected distance away from it.
- When there is a sample plane, the extrapolation starts at the measured start plane.

The extrapolated best-fitted axis is the toleranced feature.

Circular Features with Surface Data under ASME Y14.5

The toleranced feature is a 2D point when the considered feature is a circle. Under ASME Y14.5, when a circular feature has surface data, the toleranced feature is constructed in the following way:

First, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Thus, the **DEFAULT** math type usually produces the unrelated actual mating envelope (UAME) unless the modifier is LMC. In that case, the math type produces the unrelated actual minimum material envelope (UAMME). Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

Second, a projection is selected, based on whether there is a sample plane:

- When there is no sample plane, the best-fit center-point is projected to the nominal start-plane of the circle. The nominal start-plane is nominally oriented and located to the measured datums.
- When there is a sample plane, the best-fit center-point is projected to the measured start plane.

The projected point is the toleranced feature.

Circular Features with Surface Data under ISO 1101

The toleranced feature is a 2D point when the considered feature is a circle. Under ISO 1101, when a circular feature has surface data, the toleranced feature is constructed in the following way:

All of the possibilities involve fitting a circle.

First, PC-DMIS determines whether ISO 17450-3 : 2016 applies or not. To PC-DMIS, it applies when there is no material modifier, no associated tolerance feature modifier, and the feature math type is **DEFAULT**.

When ISO 17450-3 applies, the toleranced feature is the center-point of the least-squares circle.

When ISO 17450-3 does not apply, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit.

When the material modifier is MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

The axis vector of the best-fit circle is nominally oriented to the measured datums. If there is a sample plane, it controls the axis vector of the best-fit circle to the extent that the measured datums do not control the axis vectors.

Next, a projection is selected, based on whether there is a sample plane:

- When there is no sample plane, the best-fit center-point is projected to the nominal start-plane of the circle. The nominal start-plane is nominally oriented and located to the measured datums.
- When there is a sample plane, the center-point is projected to the measured start plane. The orientation of the measured start plane depends on the number of sample hits taken. If there is only one sample hit, the measured start plane inherits the nominal orientation. If there are three or more sample hits, the measured start plane inherits its orientation from the measured sample plane.

The projected point is the toleranced feature.

Conical Features With Surface Data under ASME Y14.5

The toleranced feature is an axis when the considered feature is a cone. Under ASME Y14.5, when a conical feature has surface data, the toleranced feature is constructed in the following way:

First, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**). The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit, chosen to be external to the material. Thus, the **DEFAULT** math type produces the unrelated actual mating envelope (UAME). Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way. In all of these fits, the cone angle is allowed to optimize from the nominal angle.

Second, an extrapolation is selected. It is based on whether sample hits are available:

- When there is no sample plane, the best-fit axis is extrapolated to the nominal end-faces of the cone. The nominal end-faces of the cone are nominally oriented and located to the measured datums.
- When there is a sample plane, the extrapolation starts at the measured start plane.

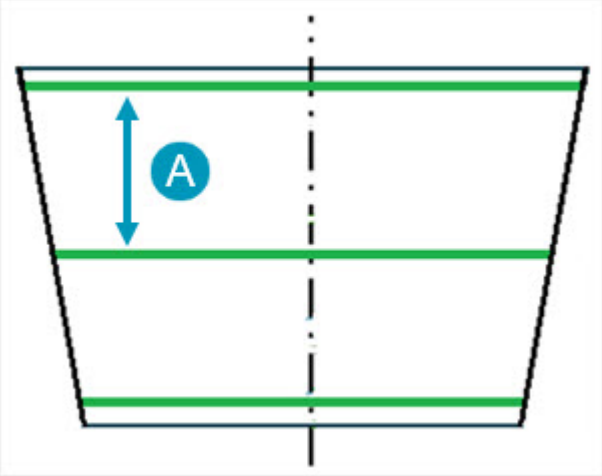
The extrapolated best-fitted axis is the toleranced feature.

Conical Features with Surface Data under ISO 1101

The toleranced feature is an axis when the considered feature is a cone. Under ISO 1101, when a cylindrical feature has surface data, the toleranced feature is constructed in the following way:

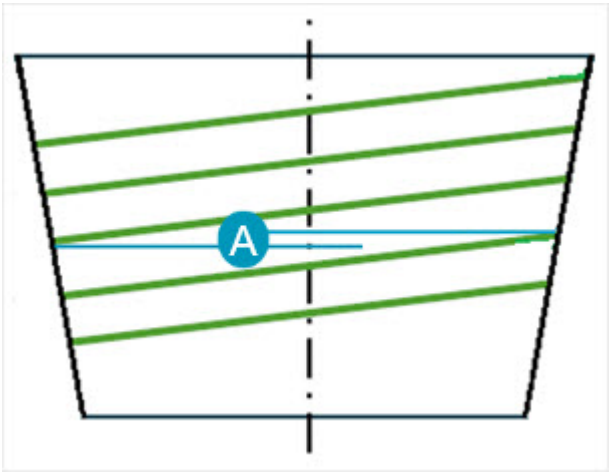
When the feature math type is **DEFAULT**, PC-DMIS decides that a generalization of ISO 17450-3 : 2016 applies so the toleranced feature is an imperfect axis derived from multiple cross sections. Each cross section has a least-squares circle fitted to it. The vector of each circle is the vector of the least-squares axis of the entire cone. The center-points of the circles form the toleranced feature. This process matches closely with the specification in ISO 17450-3. PC-DMIS determines the cross sections in the following way:

- If the cone was measured in clean cross sections, meaning multiple distinct levels with an easily detectable gap between each level, then PC-DMIS uses those clean cross sections.



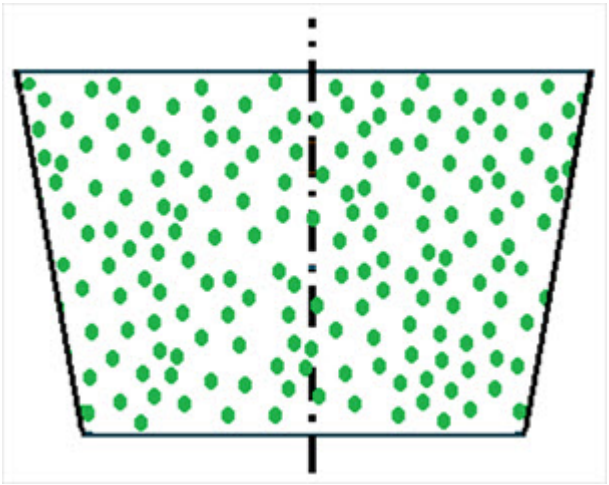
A *- Gap*
Easily detectable gap between the end of one level and the start of the next. Three cross sections are returned.

- If the cone was not measured in clean cross sections, but the data follows a clean spiral pattern, then PC-DMIS divides the spiral into several cross sections, equal to the number of revolutions in the spiral (so long as the spiral has more than two revolutions).



A *- No Gap*
You can see in this example one continuous spiral or overlapping start and end points. Five cross sections are returned.

- If the cone was not measured in either clean cross sections or a clean spiral pattern, then PC-DMIS performs a best effort to divide the data into cross sections.



You can see in this example that there is no discernible pattern. The number of returned cross sections depends on the point distribution and density.

After PC-DMIS divides the cone into cross sections, it rejects any section that has less than 90 degrees of arc. The corresponding circle center is not included in the toleranced feature.

In some cases, the above procedure produces no circle centers. For example, spiral scans with two or fewer revolutions or all cross sections were found to have less than 90 degrees of arc. In such cases, it is not possible to adhere so closely to the description in ISO 17450-3, and so an approximation is employed. Specifically, the toleranced feature is the axis of the least-squares cone, extrapolated to the endpoints of the measured surface data.

When the feature math type is **LSQ**, PC-DMIS decides that ISO 17450-3 : 2016 does not apply. A least-squares fit is computed to produce a least-squares axis. The cone angle is

allowed to optimize from the nominal angle. Next, an extrapolation is selected, based on whether sample hits are available:

- When there is no sample plane, the best-fit axis is extrapolated to the nominal end-faces of the cone. The nominal end-faces of the cone are nominally oriented and located to the measured datums.
- When there is a sample plane, the extrapolation starts at the measured start plane.

The extrapolated best-fitted axis is the toleranced feature.

Width Features with Surface Data under ASME Y14.5

The toleranced feature is a plane when the considered feature is a width. Please note that all PC-DMIS width features have surface data. Under ASME Y14.5, when the considered feature is a width, the toleranced feature is constructed in the following way:

First, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Thus, the **DEFAULT** math type usually produces the unrelated actual mating envelope (UAME) unless the modifier is LMC. In that case, the math type produces the unrelated actual minimum material envelope (UAMME). Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

Second, the surface points are all projected to the center-plane of the fitted width. The toleranced feature is then the convex polygon that describes the perimeter of those projected surface points. Mathematically, the toleranced feature is the convex hull of the projected surface points.

Width Features with Surface data under ISO 1101

The toleranced feature is a plane when the considered feature is a width. Please note that all PC-DMIS width features have surface data. Under ISO 1101, when the considered feature is a width, the toleranced feature is constructed in the following way:

First, PC-DMIS decides whether ISO 17450-3 : 2016 applies or not. To PC-DMIS, ISO 17450-3 : 2016 applies when there is no material modifier and the feature math type is **DEFAULT**.

When ISO 17450-3 applies, the toleranced feature is an imperfect plane. The toleranced feature is the center-points of the opposed two point sizes as described in ISO 17450-3 and ISO 14405-1. This process matches closely with the specification in ISO 17450-3.

When ISO 17450-3 does not apply, a fit type is selected, based on the feature math type (**DEFAULT** or **LSQ**) and on the material modifier. The **LSQ** math type always does a least-squares best fit. The **DEFAULT** math type does an inscribed or circumscribed best fit. When the material modifier is RFS (no material modifier) or MMC, the inscribed or circumscribed fit is chosen to be external to the material. When the material modifier is LMC, the inscribed or circumscribed fit is chosen to be internal to the material. Since traditional inscribed and circumscribed fits are notoriously unstable, PC-DMIS uses a constrained least squares algorithm to compute the inscribed or circumscribed fit in a stable way.

When ISO 17450-3 does not apply, and after a fit has been computed, the surface points are all projected to the fitted plane. The toleranced feature is then the convex polygon that describes the perimeter of those projected surface points. Mathematically, the toleranced feature is the convex hull of the projected surface points.

Features without Surface Data

Several types of considered features do not have surface data (for information, see "Feature Types With and Without Surface Data"). When the considered feature does not have surface data, the feature math type is unavailable in the geometric tolerance command. In most cases, you should not use features that do not have surface data. This is because the geometric tolerance command is unable to construct the toleranced feature from the surface data in a way that complies with the ASME Y14.5 or ISO 1101. Instead, you are responsible to define the toleranced feature according to any applicable standards.

For axial or linear features without surface data, the toleranced feature is the line segment from the measured start point to the measured end point. For circular, spherical, and point features without surface data, the toleranced feature is the measured centroid of the feature.

3D constructed BF lines have more complex handling. PC-DMIS interprets them as features without surface data. Instead, it interprets the input points as circle centers of circular cross sections. Under ISO 1101, this interpretation complies with ISO 17450-3 : 2016, and the toleranced feature is the set of centroids. However, under ASME Y14.5, PC-DMIS interprets 3D constructed BF lines in the same way as other axial or linear features without surface data. In that case, the toleranced feature is the start-to-end line segment (except for straightness of an axis tolerances which use all the input centroids).

The mid-plane is the only planar feature without surface data that is allowed as a considered feature. For the most part, you should use 3D widths instead of mid-planes (and 2D widths instead of mid-lines, and 1D widths instead of mid-points). The mid-plane command is still supported so that old programs continue to work after they migrate to PC-DMIS 2020 R2 or later. Since PC-DMIS still supports mid-plane for these legacy applications, its interpretation in the geometric tolerance command is similar to XactMeasure's interpretation. Specifically, PC-DMIS interprets mid planes to have four

corners that lie in the mid-plane, and the toleranced feature consists of the rectangle between those four corners.

Slots and Notches

Slots and notches are treated as 2D widths without surface data. That is, the toleranced feature is a line centered on the feature's centroid. For slots, users can choose whether the slot is considered widthwise or lengthwise, as discussed in "Lengthwise versus Widthwise Slots":

> For a widthwise slot, the size of the slot is the slot's width, and the tolerance zone controls the position in the width direction. That means the toleranced feature line is parallel to the slot's length and is as long as the slot's length.
>
> For a lengthwise slot, the size of the slot is the slot's length, and the tolerance zone controls the position in the length direction. That means the toleranced feature line is parallel to the slot's width and is as long as the slot's width.
>
> For notches, the size of the notch is the notch's length, and the tolerance zone controls the position in the length direction. That means the toleranced feature line is parallel to the notch's width and is as long as the notch's width.

Tangent plane modifier

Most of the time, for plane-type considered features, the toleranced feature is the considered feature's surface data. However, the tangent plane modifier makes the toleranced feature different from the surface data. Angularity, parallelism, perpendicularity, and position tolerances on planes are allowed to use the tangent plane modifier. The toleranced feature is derived in the following way.

First, an external-to-material constrained least squares plane is fitted in a way that removes the influence of surface voids. This is the same way that primary datum planes are fitted under ASME Y14.5 with **DEFAULT** datum math, and the same way primary datum planes are fitted under ISO 1101 with **CL2** datum math. We use this math because (1) it is external to the material, (2) it mimics the behavior of a surface plate quite well, and (3) it is stable compared to other external-to-material fits.

Next, the surface points are all projected to the constrained least squares plane. The toleranced feature is then the convex polygon that describes the perimeter of those projected surface points. Mathematically, the toleranced feature is the convex hull of the projected surface points.

Simultaneous Tolerances

Many geometric tolerances are meant to be considered simultaneously. For example, when position and/or profile of a surface specifications reference the same partially

constrained datum reference frame, you should usually consider them simultaneously. For more details, see the following standards:

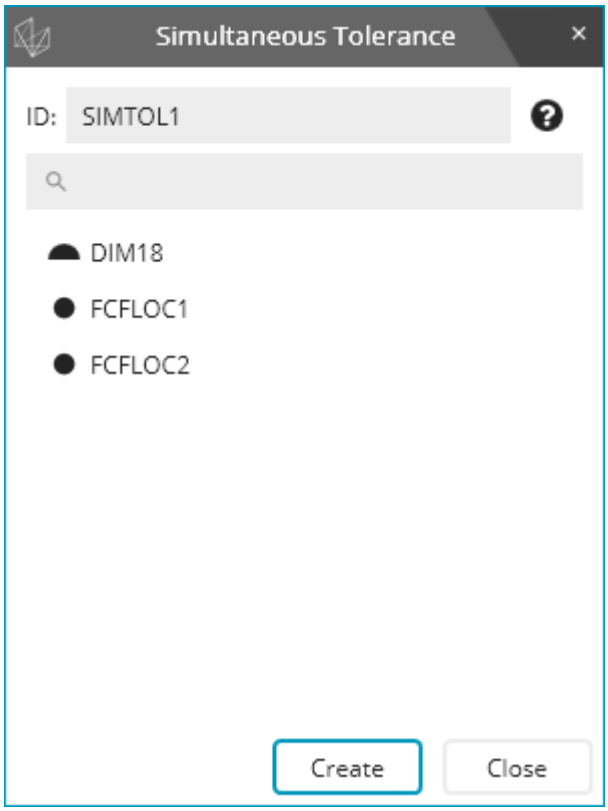
- ASME Y14.5 - 1994 sections 4.5.12 and 5.3.6
- ASME Y14.5 - 2009 section 4.19
- ASME Y14.5 - 2018 section 7.19
- ISO 5458 - 1998

You are responsible to choose which geometric tolerances to consider simultaneously. To do this, you create a simultaneous tolerance command for each set of simultaneous geometric tolerances.

PC-DMIS can consider profile of a line specifications simultaneously so long as they have at least one datum, but this is not recommended. For more details, see "Profile of a Line".

Defining a Simultaneous Tolerance

1. From the menu, choose **Insert | Dimension | Simultaneous Tolerance** to open the **Simultaneous Tolerance** dialog box:



2. Modify the dimension ID if needed.
3. Select which geometric tolerance commands belong to the simultaneous tolerance.

4. After you select a geometric tolerance command, the list filters itself so you only see dimensions with the same datum reference frame as the first tolerance command you selected.

Command Mode Syntax

In the Edit window, the Command Mode syntax looks like this:

```
SIMTOL2 =SIMULTANEOUS_TOLERANCE/
FCFLOC2,FCFLOC5,FCFPROF12,,
```

As in the dialog box, the only controls in the Edit window are for the dimension ID and which geometric tolerance commands belong to the simultaneous tolerance.

Behavior

Consider the simultaneous tolerance command as an annotation or directive. Nothing happens if you execute it. It also does not appear in the report. Instead, the simultaneous tolerance command changes how the geometric tolerances are evaluated.

During execution, the geometric tolerance commands know that they belong to the simultaneous tolerance command. They also know which of the other geometric tolerance commands are also part of the simultaneous tolerance command. If you've measured all of the inputs to all of the simultaneous tolerances (so the tolerance is ready to evaluate), the simultaneous calculation takes place. If you haven't measured all the input features, then the tolerance is not ready to evaluate. In this case, PC-DMIS temporarily shows a "Waiting for evaluation" message in all of the geometric tolerance commands in the simultaneous set. The message identifies which geometric tolerance is causing the delay, and any unmeasured features. Your report temporarily shows "Waiting" as well, instead of a measured value. Once you measure all input features, PC-DMIS updates the Edit window and report with the measured values.

The report for a tolerance that belongs in a simultaneous set looks like this:

FCFLOC8		MM	⌖ ∅0.1Ⓜ D NⓂ OⓂ : SIMTOL2 LSQ					ASME Y14.5 2018
Feature	AX	NOMINAL	+TOL	-TOL	MEAS	DEV	BONUS	OUTTOL
	X	57.150000			57.211490	0.061490		
	Z	101.600000			101.529825	-0.070175		
DAT_Y1_Z1	TP	0.000000	0.100000	0.000000	0.186606	0.186606	0.024326	0.062280
	X	-209.550000			-209.559500	-0.009500		
	Z	-25.400000			-25.493130	-0.093130		
DAT_Y2	TP	0.000000	0.100000	0.000000	0.187227	0.187227	0.024740	0.062487

The report is essentially the same as any other geometric tolerance. PC-DMIS shows the label in the report in the same place it would be if there were no simultaneous tolerance. You can know if a tolerance was considered simultaneously by the text in the header of the

label. For example, in the image above, SIMTOL2 is that text. It indicates which simultaneous tolerance command the geometric tolerance command belongs to.

PC-DMIS does not consider lower segments of composite position or profile tolerances simultaneously. The upper segment is simultaneous with other tolerance commands, but PC-DMIS considers any lower segments independently of everything else.

Measurement Routine Structuring Recommendations

Be aware that during execution, when PC-DMIS executes each geometric tolerance in the simultaneous set, it displays a temporary "Waiting" message until all tolerances belonging to the simultaneous set have been executed. The easiest way to avoid this temporary "Waiting" message is to measure all your features first, then put all your tolerances after all your features. This ensures that all tolerances are ready to evaluate, and you never get a "Waiting" message.

If you use expressions with your geometric tolerance commands, place them late enough in your measurement routine so that you don't get a "Waiting" message.

Finally, while your measurement routine does not need any special structure to use simultaneous tolerances, it is safest to have the same “active alignment” for all the individual Geometric Tolerance commands and the “Simultaneous Tolerance” command. Where possible, place the “Simultaneous Tolerance” command directly after the final Geometric Tolerance command for that group of FCFs (don’t wait until the end of the measurement routine). If there are alignment changes in the measurement routine, measured values will be unaffected, but it may be necessary to add “Recall Alignment” commands to prevent problems with the XYZ summary information.

Comparison to Past Practice

PC-DMIS 2020 R2 introduced the simultaneous tolerance (`SIMULTANEOUS_TOLERANCE`) command. This command works with the new geometric tolerance command. Prior to this, PC-DMIS had the simultaneous evaluation (`SIMULTANEOUS_EVALUATION`) command. It was designed to work with the XactMeasure command. The new simultaneous tolerance command works differently from the simultaneous evaluation command:

Simultaneous Evaluation (2020 R1 and Earlier Behavior)	Simultaneous Tolerance (2020 R2 and Later Behavior)
XactMeasure tolerances were unmarked for execution (blue in the Command mode edit window).	Geometric tolerances are marked for execution (white in the Edit window in Command mode).
XactMeasure tolerances had nothing in the report.	Geometric tolerances report as usual, and a small note in the report states that they belong to a simultaneous set.
The simultaneous evaluation report had all of the results from all of the tolerances in the simultaneous set.	Simultaneous tolerance produces nothing in the report

Migration

If you open a program from version 2020 R1 or earlier, PC-DMIS does the following:

- Converts any simultaneous evaluation commands to simultaneous tolerance commands
- Converts XactMeasure tolerances to geometric tolerance commands
- Marks the geometric tolerance commands for execution (white in the Edit window in Command mode)
- If any profile of a line specifications are included in a simultaneous evaluation command, that do not reference any datums, PC-DMIS converts them to profile of a surface.

For additional information on migration, see "Migration from Prior Versions of PC-DMIS".

Outputting Results from Geometric Tolerances

There are many ways to output results from geometric tolerance commands.

Statistical Data

Statistical data is one of the most common ways to access results from geometric tolerances. For more information, see the "Tracking Statistical Data" chapter.

Only the `STATS/ON,Datapage+` or `STATS/ON,QDAS` methods of statistical output support Geometric Tolerance commands.

Excel Output

Excel output is another common way to access results from geometric tolerances. For more information, see "Output to an Excel File" in the "Using Basic File Options" chapter of the PC-DMIS Core documentation.

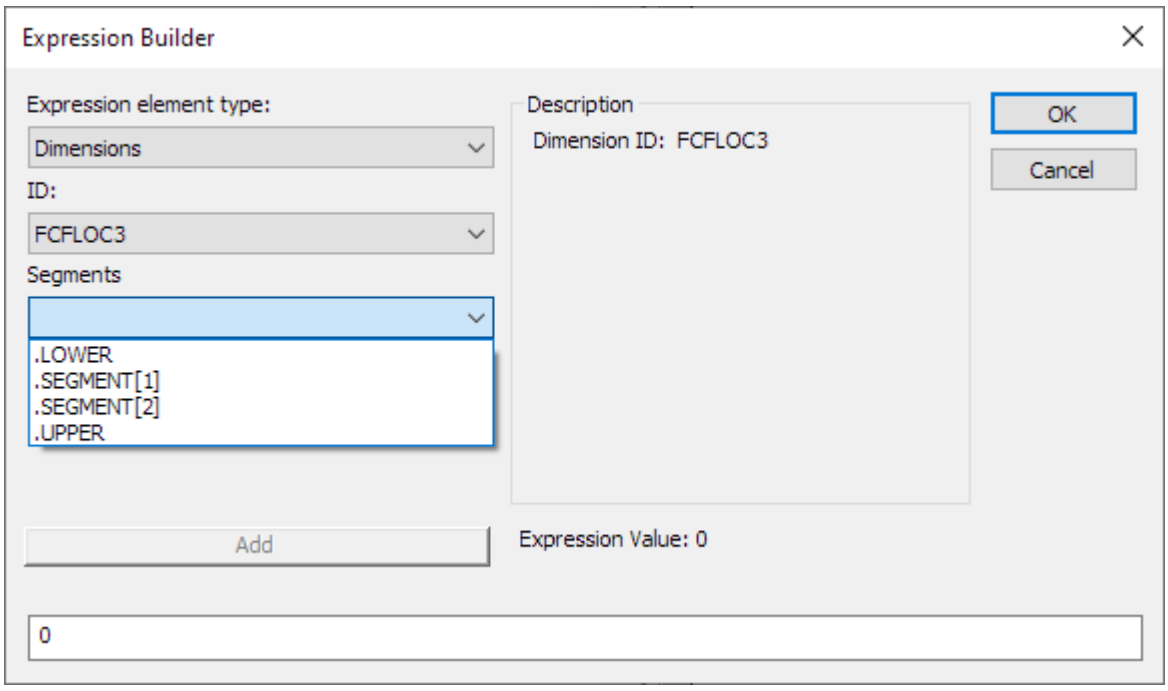
You can also export content with the Excel Form Report (**Insert | Report Command | Excel Form Report**). For details, see "Using the Excel Form Report Command" section of the PC-DMIS Toolkit Modules documentation.

Expressions

Expressions are also a common way to access results from geometric tolerances. For a general overview of how expressions work, see the "Using Expressions and Variables" chapter.

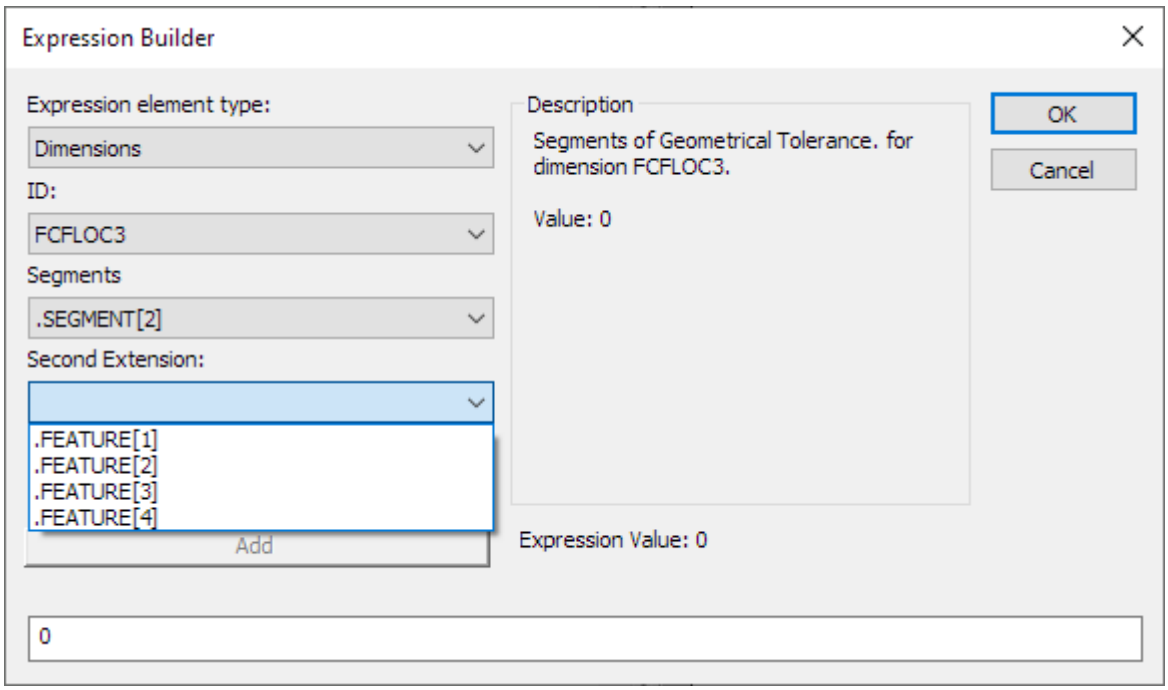
The easiest way to construct an expression that references a geometric tolerance is with the **Expression Builder** dialog box.

1. Choose the **Edit | Expression** menu option to access the **Expression Builder** dialog box. If the menu option is not visible, your Edit window cursor must be on a field that can accept an expression, such as the value of a variable assignment.
2. From the **Expression element type** list, select **Dimensions**.
3. From the **ID** list, select the dimension's ID name.
4. From the **Segments** list, select the segment to use in your expression. You will see all of the geometric tolerance command's segments listed here:



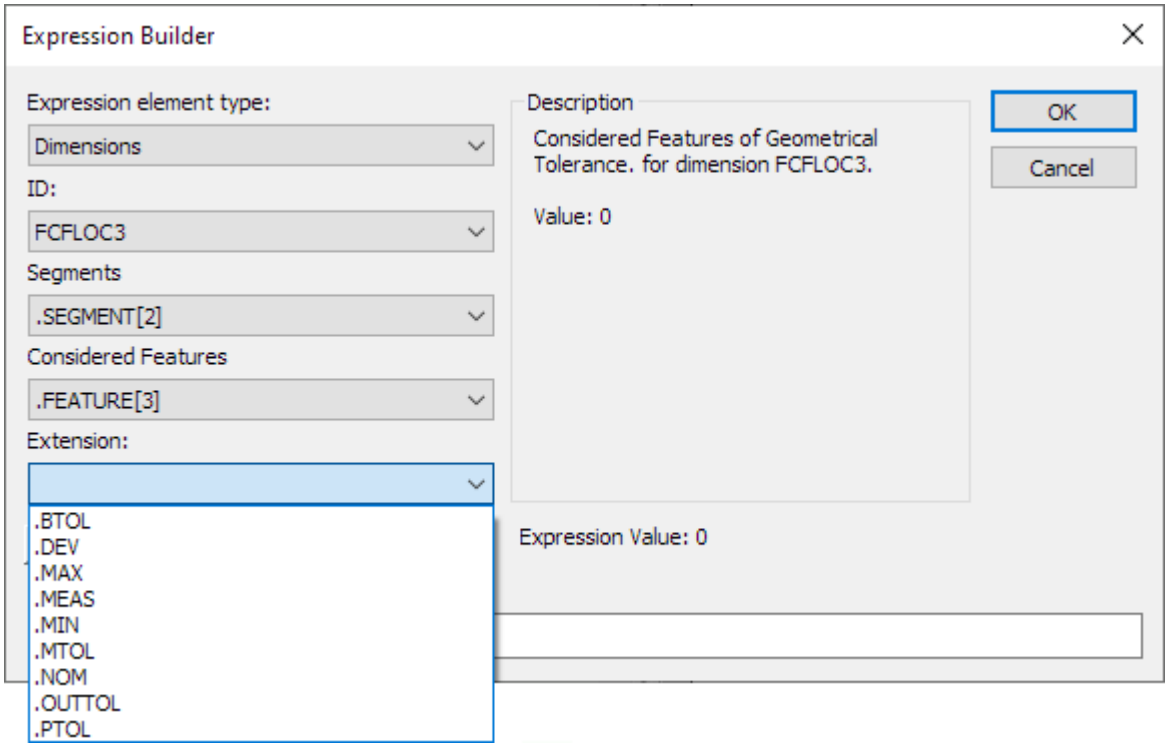
- **.LOCAL_SIZE** accesses the local size
- **.UAME** accesses the unrelated mating envelope size
- **.SEGMENT[1]** accesses the first segment
- **.SEGMENT[2]** accesses the second segment

5. After you select your segment, the **Second Extension** list appears. From that list, choose the feature you want to know the results from:



PC-DMIS shows all of the geometric tolerance command's features listed here.

6. After you choose your feature, the **Extension** list appears. From that list, choose the extension.



   For information on these extensions and what information they contain, see "Valid Extensions for Dimension Reference of Type Double by Example" in "References of Type Double" in the "Using Expressions and Variables" chapter.

7. Click **OK** to insert the expression into the Edit window.

Notes on Geometric Tolerance Report Labels

GEOTOL_SUMMARY labels

Due to the design of the Geometric Tolerance GEOTOL_SUMMARY labels (GEOTOL_SUMMARY.lbl, GEOTOL_SUMMARY1.lbl, GEOTOL_SUMMARY2.lbl, GEOTOL_SUMMARY3.lbl, and GEOTOL_SUMMARY4.lbl), you should not customize the vertical and horizontal alignment.

The reason you should not customize GEOTOL_SUMMARY labels is that they use a single row for each feature. Because of this, the content (for example, X, Y, Z, PR, PA, and TP) uses non-trimmable carriage returns to define each row of data. If you try to set a horizontal alignment to the GEOTOL_SUMMARY label, it only affects the first row. If you try to set a vertical alignment to the GEOTOL_SUMMARY label, it produces a stair step report since +Tol and -Tol are not shown for X, Y, Z, PR, and PA.

SIZE_GEOTOLERANCE.LBL label

The Geometric Tolerance SIZE_GEOTOLERANCE.LBL labels are individual rows but they are unique in that they group the upper and lower sizes for each feature. For this reason, you should not customize the SIZE_GEOTOLERANCE.LBL label.

Text Mode Dimension Reporting does not support Geometric Tolerance commands. For details on editing text in Geometric Tolerance reports, see "Edit Text Reporting" in the "Changing the Report Window's Content" section of the PC-DMIS Core documentation.

Migration from Prior Versions of PC-DMIS

In this Topic Hide

Introduction

When you open a measurement routine from a prior version, PC-DMIS attempts to migrate the commands. Depending on the version the routine was previously saved as, the types of Geometric Tolerance commands it contains and the features they reference, you may encounter a migration report. The migration report details any errors that were encountered, and any changes that needed to be applied to make your routine compatible with PC-DMIS 2026.2.

Prior to migration, when you open your routine in this version of PC-DMIS, the software creates a backup of your measurement routine in this folder:

C:\Users\Public\Documents\Hexagon\PC-DMIS\2026.2\MigrationBackup

Never open this measurement routine from this backup location. If you want to use the backed-up measurement routine, copy it to another folder first.

If you click the **Cancel** button on the migration report, PC-DMIS discards the migrated measurement routine and automatically restores the original version.

You can control whether PC-DMIS creates a backup of the measurement routine or not with the Settings Editor `MigrationBackup` setting. This setting is set to **True** by default. If you set this to **False**, PC-DMIS still presents you with a migration report, but it does not create a backup of the measurement routine. Because of this, PC-DMIS does not show the **Cancel** option on the migration report since no backup is available to perform a restore.

For details on the `MigrationBackup` setting, see the "MigrationBackup" topic in the "FileMan" section of the PC-DMIS Settings Editor documentation.

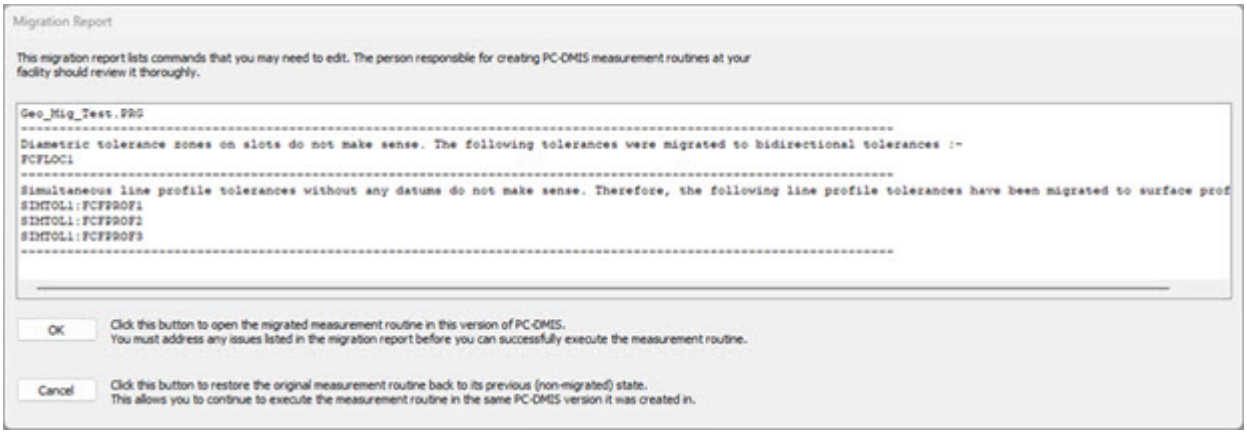
Suggested Workflow

Migration is mostly automatic, but in some cases, you may need to customize the migration to your needs. You can use some options to control how the migration takes place. We suggest the following workflow to migrate your PC-DMIS measurement routines from a prior version:

- Back up your old measurement routines to a safe place and never open them with 2020 R2 or later.
- Make a copy of your backup measurement routines in a folder you are willing to experiment with.
- Open the experimental copies of your measurement routines in this version of PC-DMIS.
- Carefully examine the results of the migration. Verify that the migration worked as desired and that the new measured values fit your needs.
- In rare cases, a migration might not work for you in a small number of places. In these cases, manually edit those places in your program to update the commands.
- In rare cases, the migrated math types might not work for you in a large number of places. If this happens, adjust your migration options, make new copies from your backup measurement routines, and place the copies in the experimental folder. Then open these new copies with this version.
- Repeat this workflow until all your measurement routines are working for you.

Migration Report

When PC-DMIS encounters issues during migration, the migration tool creates a report in a **Migration Report** dialog box. The migration report looks like this:



Migration Report dialog box with a migration report.

PC-DMIS automatically saves the migration report in the same location as the measurement routine with the same name as the measurement routine but with _migratedReport.txt appended to it.

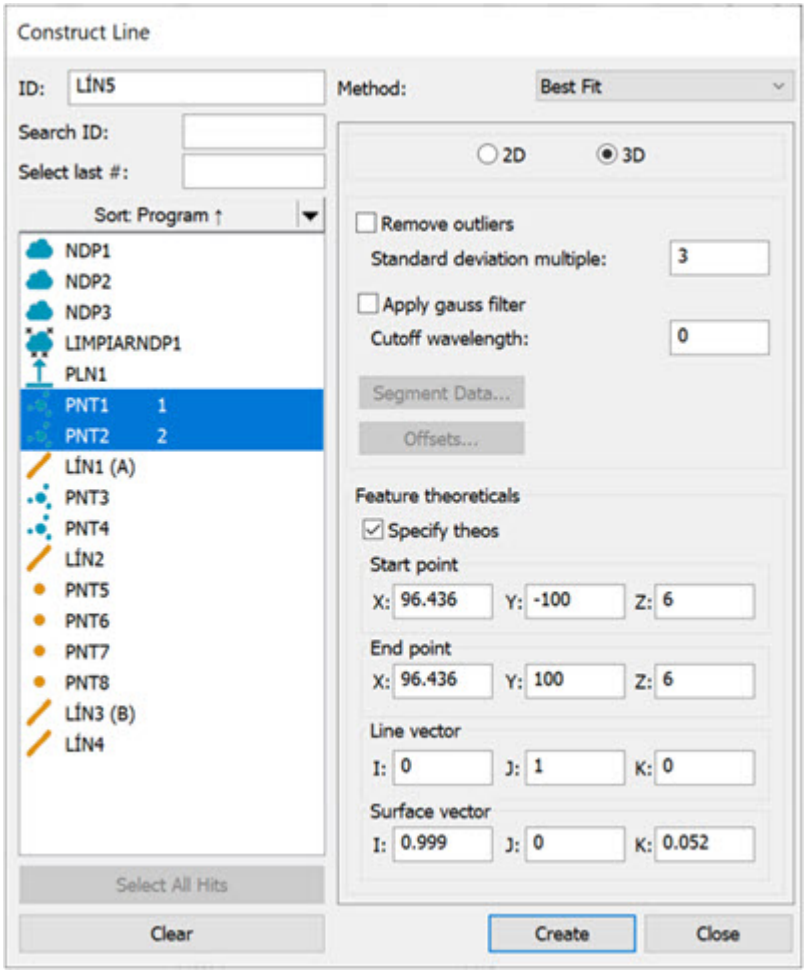
Most migration reports are much simpler than the example in the image above. In the image above, the report conveys these main pieces of information:

- PC-DMIS changed FCFLOC1 from a single, diametric position of a slot to two separate planar positions of the slot – one for the slot's length-wise orientation and one for the slot's width-wise orientation. This is because support for diametric position of slots was removed in PC-DMIS 2020 R2 SP1.
- PC-DMIS detected multiple profile of a line tolerances to no datums were previously being evaluated simultaneously. Support for simultaneous line profile tolerances without any datums was removed in PC-DMIS 2022.1 and the following service packs: 2020 R2 SP11, 2021.1 SP8, and 2021.2 SP3. For details, see the "Simultaneous Profile of a Line Tolerances" section in this topic.

Important Notes

Constructed Line Surface Vectors

All constructed BF and BFRE lines have a line vector and a surface vector which are important to the geometric tolerance command. However, while you can see and edit the line vector in the edit window when in Command mode, the surface vector information is only accessible through the **Specify Theos** section of the line's **Construct Line** dialog box.



If you manually edit the line vector, PC-DMIS automatically adjusts the Surface vector so that it remains orthogonal. In older versions of PC-DMIS, this was not always the case, and the surface vector may have been incorrect (where THEO line vectors had been corrected in the edit window after manually teaching points on the machine or when a measurement routine was created without a CAD model for example). XactMeasure did not use the line's surface vector, so this was previously not a problem. For the Geometric Tolerance command however, correct surface vectors are important in many cases. Therefore, starting with PC-DMIS version 2024.1, PC-DMIS now checks for lines with non-orthogonal line and surface vectors when it opens measurement routines. If any error is found, PC-DMIS automatically normalizes the surface vector and informs you with a warning message in the migration report.

Simultaneous Profile of a Line Tolerances

If a set of profile of a line tolerances are part of a simultaneous evaluation command, and if those profile of a line tolerances do not reference any datums, then they are migrated to profile of a surface. This is because it does not make sense (from a standards compliance perspective) to simultaneously evaluate profile of a line tolerances. The Migration Report announces the migration when it happens. For more details, see "Profile of a Line" and "Simultaneous Tolerances".

Options to Control Migration

When you migrate measurement routines from earlier versions of PC-DMIS that contain Geometric Tolerance commands, PC-DMIS attempts to automatically select the appropriate GD&T standard using the following rules:

- If the combined total of ISO Size and Geometric Tolerance commands ***is greater*** than the combined total of ASME Size and Geometric Tolerance commands, then ***all*** of the migrated Size and Geometric Tolerance commands use ISO 1101 (see note).
- If the combined total of ISO Size and Geometric Tolerance commands ***is less*** than the combined total of ASME Size and Geometric Tolerance commands, then ***all*** of the migrated Size and Geometric Tolerance commands use ASME Y14.5, subject to additional rules for selecting the year (see note).
- If the combined total of ISO Size and Geometric Tolerance commands ***is equal to*** the combined total of ASME Size and Geometric Tolerance commands, then ***all*** of the migrated Size and Geometric Tolerance commands use the GDTStandard entry in the Dimensions section of the PC-DMIS Settings Editor. For details, see "GDTStandard" in the PC-DMIS Settings Editor documentation (see note).

Additional rules for determining which year of ASME Y14.5 a routine should migrate to:

- If the ASME Geometric Tolerance commands include concentricity or symmetry tolerances, the migrated measurement routine will use ASME Y14.5 – 2009.
- If the ASME Geometric Tolerance commands do not include any concentricity or symmetry tolerances, the migrated measurement routine will use ASME Y14.5 – 2018.

Starting with PC-DMIS version 2023.2, you cannot have different GD&T standards in a single measurement routine. Therefore, PC-DMIS does not allow you to migrate a 2023.2 (or newer) measurement routine from ISO to ASME or vice vera.

ASME Y14.5 – 1994 does not allow the use of the translation modifier, dynamic profile modifier, customized datum reference frames or a specified material boundary size. It also uses the older, two-value definition of profile (see ASME Y14.5.1M – 1994).

ASME Y14.5 – 2009 does not allow the use of the dynamic profile modifier.

ASME Y14.5 – 2018 does not allow the use of concentricity or symmetry.

In some cases, you may not want to use the standard that PC-DMIS chooses during migration. You can control which migrated standard to apply. To do this, create a file named fcfMigrationPreferences.json and place it in "*C:\ProgramData\Hexagon\PC-DMIS\2026.2*" folder. Note that the C:\ProgramData\ folder is hidden by default.

The "*fcfMigrationPreferences.json*" file must be in the JSON format (see https://en.wikipedia.org/wiki/JSON).

Here is an example file:

```
{
"default standard migrates to":"ASME Y14.5-2018"
}
```

"default standard migrates to" is the only valid key for the "*fcfMigrationPreferences.json*" file. When the "*fcfMigrationPreferences.json*" file does not exist (or it exists but it is not valid a valid JSON file), then PC-DMIS selects the migrated math options according to the rules described above. When the "*fcfMigrationPreferences.json*" file exists, and PC-DMIS determines that it is a valid JSON file, and it has the valid key and values for controlling the migration, then PC-DMIS uses the math options defined by the "*fcfMigrationPreferences.json*" file.

Invalid keys in the "*fcfMigrationPreferences.json*" file are allowed, but they are ignored. If the "*fcfMigrationPreferences.json*" file exists, and PC-DMIS determines that it is a valid JSON file, but that it lacks the valid key or value, then the behavior for that key is the same as if there were no "*fcfMigrationPreferences.json*" file.

The valid values for the "default standard migrates to" key for the "*fcfMigrationPreferences.json*" file are:

- "ASME Y14.5-1994"
- "ASME Y14.5-2009"
- "ASME Y14.5-2018"
- "ISO 1101"

The valid keys and values in the "*fcfMigrationPreferences.json*" file must be in English.

Dimensioning Size

The Size command (**Insert | Dimension | Size**) facilitates computing and reporting of local and global sizes per the ISO 14405-1 or ASME Y14.5 standards. The ISO 14405-1 and the ASME Y14.5 standards define local and global actual sizes.

For ASME Y14.5, the allowed features are cylinders, opposed parallel planes (also known as 3D widths), spheres, circles (cylinder cross sections), 2D widths (3D width cross sections). ASME Y14.5 defines the unrelated actual mating envelope and the local size.

For ISO 14405-1, the allowed features are cylinders, 3D widths, and their cross sections. Over twenty modifiers are defined in ISO 14405-1. You can combine these modifiers in various ways to create a specification operator for size. Separate specification operators are allowed for the upper and lower size limits. This means that thousands of different kinds of size calculations exist.

Command Modes

The Size command defines four different modes. These modes allow you to enter, calculate, and report different kinds of size calculations:

1. **ASME Y14.5**

   The Size command in this mode does the following:

   - It records a nominal size, an upper deviation limit, and a lower deviation limit.
   - It calculates the unrelated actual mating envelope. The command also calculates a long list of local sizes.
   - It reports the actual mating envelope. The command also reports the worst of the local sizes.

2. **ISO 14405-1, Nominal with Deviation**

   The Size command in this mode records a nominal size, an upper deviation limit, and a lower deviation limit.

   - If there is a single specification operator, the operator applies to both limits.
   - If there are two specification operators, each operator only applies to one limit.
   - If a specification operator is a global size, that global size is computed, and the report compares the global size to the applicable limit or limits.
   - If a specification operator is a local size, a long list of those local sizes is computed, and the worst local size is reported for each applicable limit.

3. **ISO 14405-1, ISO Code**

   The Size command in this mode records a nominal size and an ISO code. This information, together with ISO 286-1, define the limits of size.

   The ISO 286-1 standard defines hundreds of tolerance codes that look somewhat like "E9" and "H7". These tolerance codes are all supported by the size command. Reporting is much like the ISO nominal-with-deviations mode.

4. **ISO 14405-1, Range of Sizes**

You should use this mode with the range of sizes modifier defined by ISO 14405-1 (the text SR enclosed in an oval).

The Size command in this mode does the following:

- It only records one specification operator.
- It does not record a nominal size, nor limits on deviations.
- It records a single tolerance value, because the range of sizes modifier produces specification operators that are similar to form tolerances. For an example, see Figure 17 in the ISO 14405-1 standard. Reporting in this mode is similar to reporting for form tolerances.

Input Features

The Size command allows only one input feature at a time.

In the ASME Y14.5 and ISO 14405-1 modes, the valid input feature types are 1D widths, 2D widths, 3D widths, circles, cylinders, and spheres.

Any input features that you use with the Size command should contain enough points to represent well the actual surface. To best mimic the intent of the standards, those points should be opposing points.

Limits on Input Features for Section Sizes

Several ISO specification operators for size are referred to as "section sizes". For example, "(GG) ACS" and "(LP) ACS (SX)" are section sizes. Section sizes are local sizes where each cross-section of the cylinder or of the 3D width has a size. Then, the worst section size is reported for each applicable limit.

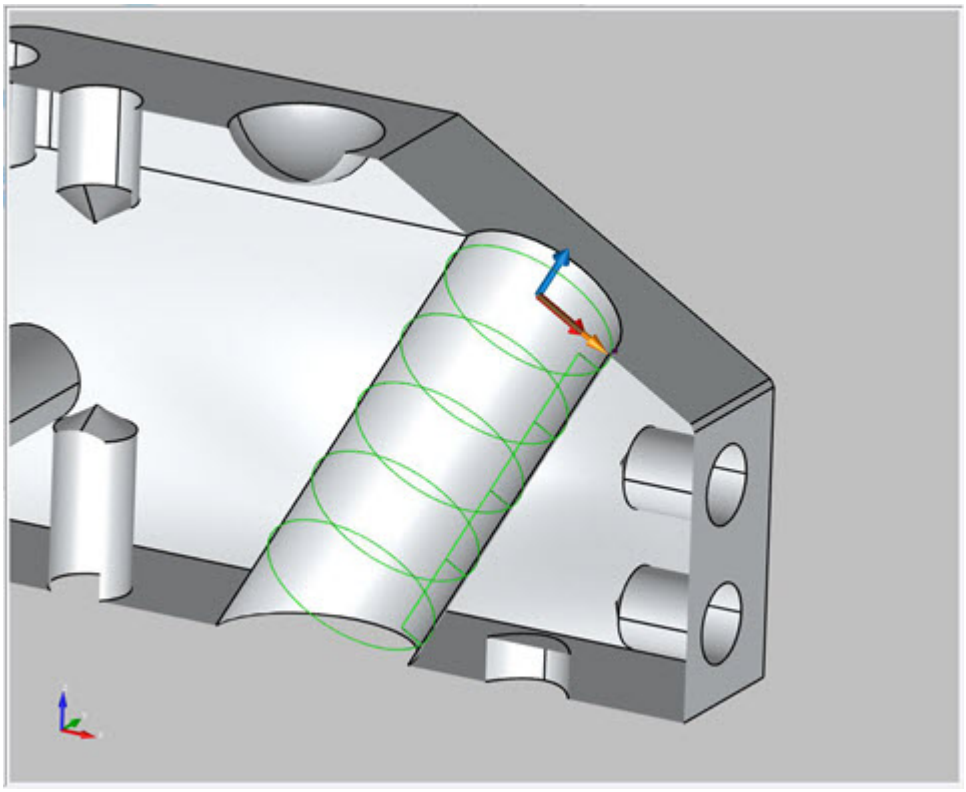
One possibility for producing data in sections is to use 2D cross section features like circles and 2D widths. The other possibility is to use a measurement strategy that produces data in sections, such as "Default PC-DMIS Measurement Strategy" or "Adaptive Cylinder Concentric Circle Scan". For more information on measurement strategies, see the "Working with Measurement Strategies" topic in the "PC-DMIS CMM" documentation.

When evaluating the section sizes, the command partitions the data into sections, according to these rules:

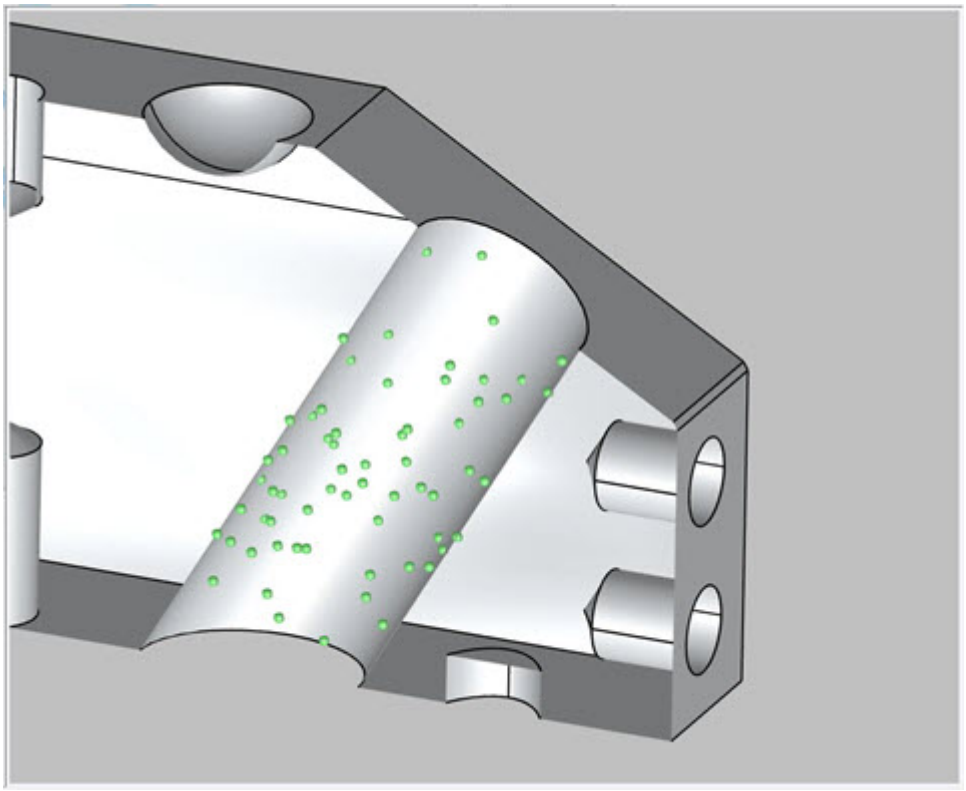
- For circle features and 2D widths, the command uses the data without partitioning, because the feature is already a cross section.
- For cylinders, the command attempts to partition the data into circular cross sections. The data must be arranged in circles or the command will fail.
- For 3D widths, the command fails.

When partitioning cylindrical data, the command first projects the points onto the cylinder's axis. It then identifies clusters of projected points as belonging to the same cross section.

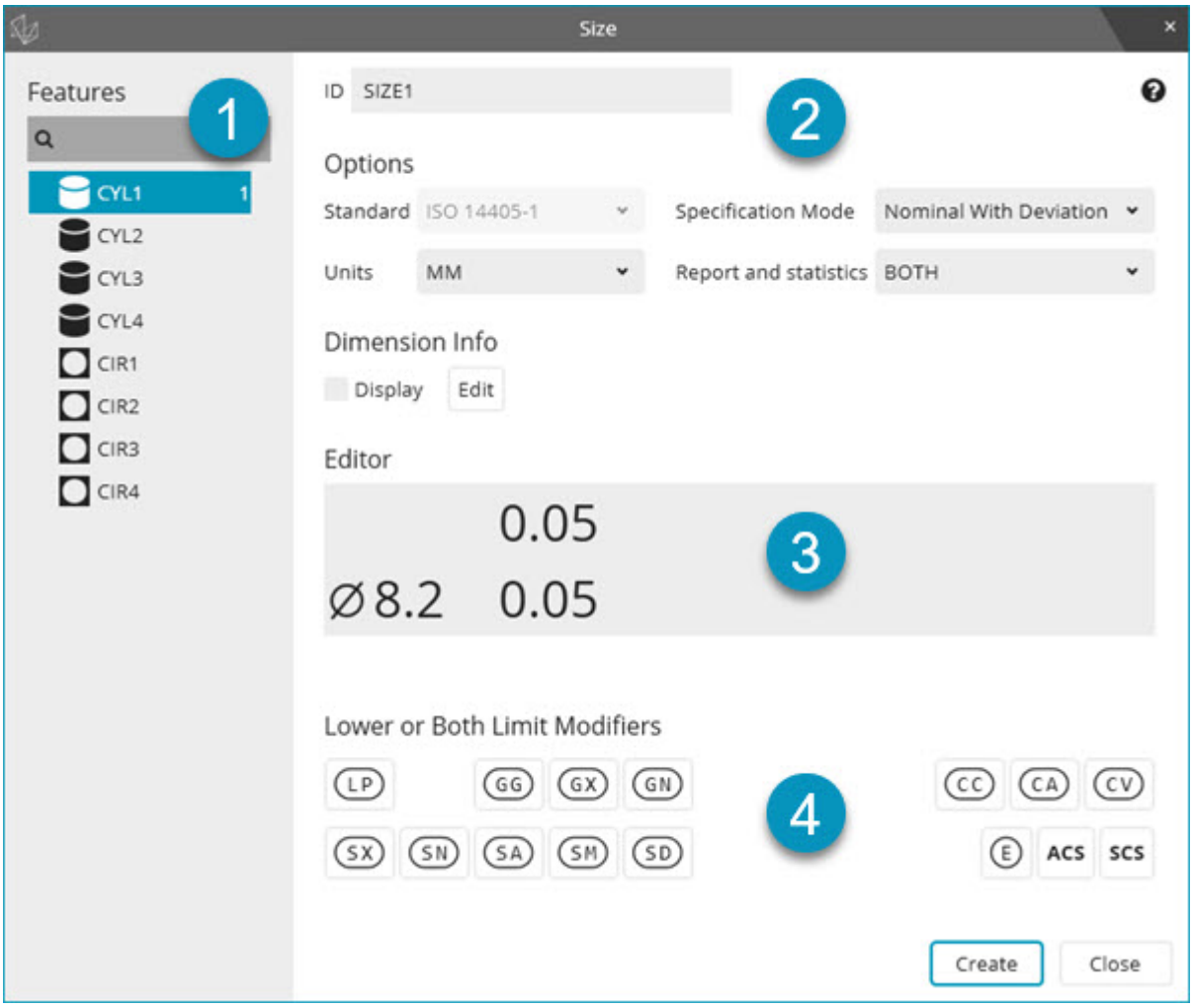
Example of Good Inputs



Example of Bad Inputs



To Dimension a Feature Using the SIZE Option



The main components of the Dimension Size dialog box are:

- ***1** - Feature List area*
- ***2** - Options area*
- ***3** - Editor area*
- ***4** - Limit Modifier area*

For information on how to display dimensional information, see the "Display Dimension Information" topic in the PC-DMIS Core documentation.

To dimension a feature using the SIZE option, do this:

1. Select **Insert | Dimension | Size** to show the **Size** dialog box.
2. The **Standard** list in the **Options** area shows the current standard to use in the tolerance. It should match the standard used in your print. PC-DMIS supports prints based on ASME Y14.5 and ISO 1101.
3. From the **Options** area, choose your options:
    a. The **Standard** list, select the standard to use to calculate the size.

b. If you use the ISO 14405-1 standard, from the **Specification Mode** list, select the Command mode. For information, see "Command Modes". The print requirement determines the selection to use.

c. If you use the ASME Y14.5 standard, a **Local Size options** list appears where **Specification Mode** is. Choose the type of local size, either **Opposed Points** or **Circular elements** (default).

You can find details on the Opposed Points and Circular Elements interpretations of Local Size in the "Local Size" section of the "Evaluating Size with the Geometric Tolerance Command" topic in the PC-DMIS Core documentation.

d. Set the **Units** and **Report and Statistics** as needed. For information on these options, see the "Common Dimension Dialog Box Options" topic in the "Using Legacy Dimensions" chapter.

4. In the **Editor** area, use the steps below to define the tolerance.

The **Editor** area adjusts to fit the mode requirements that you choose from the **Standard** and **Specification** lists. This means any existing changes you made to the dialog box are lost when you select a new specification mode.

a. Select the region you want to edit and type in the value.

b. To add an upper specification operator, click on the upper tolerance value. From the **Lower or Both Limit Modifiers** area, you can then click the modifier buttons to add modifiers into the upper specification operator.

c. To add a lower specification operator, click on the lower tolerance value. From the **Lower or Both Limit Modifiers** area, you can then click the modifier buttons to add modifiers into the lower specification operator. When there's only one specification operator, put the modifiers in the lower specification operator.

d. To remove a modifier, select the button again to clear the highlight.

Example showing selected modifiers.

5. Click the **Create** button to create the Size command in the Edit window.

Reading the Report

ISO 14405-1 : Nominal with Deviations

With the NOMINAL_WITH_DEVIATIONS mode, PC-DMIS compares the upper specification operator measured size against the upper limit of size. It also compares the lower specification operator measured size against the lower limit of size. Therefore, the Size command generates two measured values for a given feature of size:

SIZE1-CYL1			MM		Ø 25.4 [+0.15 GN] - [-0.15 GX]	
MODIFIER	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL
GN	25.500	25.400	0.150		0.100	0.000
GX	25.300	25.400		0.150	-0.100	0.000

ISO 14405-1 : Tolerance Codes

With the TOLERANCE_CODE mode, the PC-DMIS report mimics NOMINAL_WITH_DEVIATIONS mode, except that it displays the tolerance code in the dimension header:

SIZE2-CYL1			MM		Ø 25.4 JS14 [GN] - [GX]	
MODIFIER	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL
GN	25.500	25.400	0.260		0.100	0.000
GX	25.300	25.400		0.260	-0.100	0.000

ISO 14405-1 Range of Sizes

With the RANGE_OF_SIZES mode, PC-DMIS compares a maximum measured size against a minimum measured size and reports the difference. This mode requires a single measured value. The software compares that measured value against an upper tolerance.

SIZE3-CYL1			MM		Ø 0.25 SR	
MODIFIER	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL
SR	0.200	0.000	0.250		0.200	0.000

ASME Y14.5 - Local Size

With ASME Y14.5, two size characteristics are reported, the Unrelated Actual Mating Envelope (UAME) and Local Size.

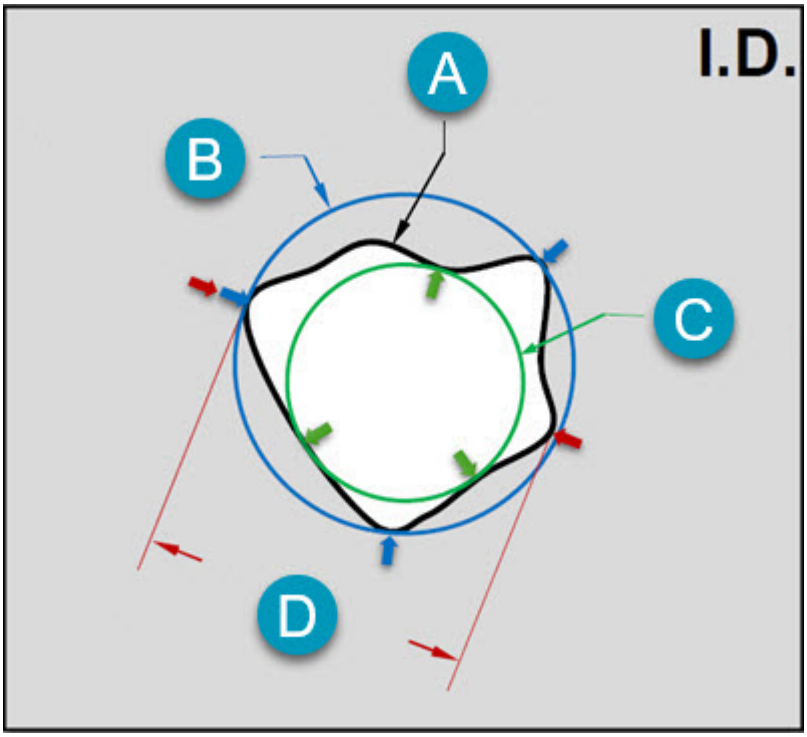
The UAME is the minimum circumscribed size for external features (e.g. pin) and the maximum inscribed size for internal features (e.g. hole).

Local Size:

1. **Circular Elements** (default) report the smallest circumscribed circular element (internal feature/ hole) or largest inscribed circular element (external feature/pin) amongst all local sizes.
2. **Opposed Points** reports the largest opposed point distance (internal feature/hole) and smallest opposed point distance (external feature/pin) amongst all local sizes.

SIZE4-CYL1			MM		Ø 25.4 +0.15/-0.15 OPPOSED POINTS	
MODIFIER	MEAS	NOMINAL	+TOL	-TOL	DEV	OUTTOL
UAME	25.300	25.400	0.150	0.150	-0.100	0.000
Local Size	25.500	25.400	0.150	0.150	0.100	0.000

ASME Y14.5 Local Size Example



A. **True shape of the feature's cross section**

B. **Local Size (Circular Elements) - ⌀ 44.2659**

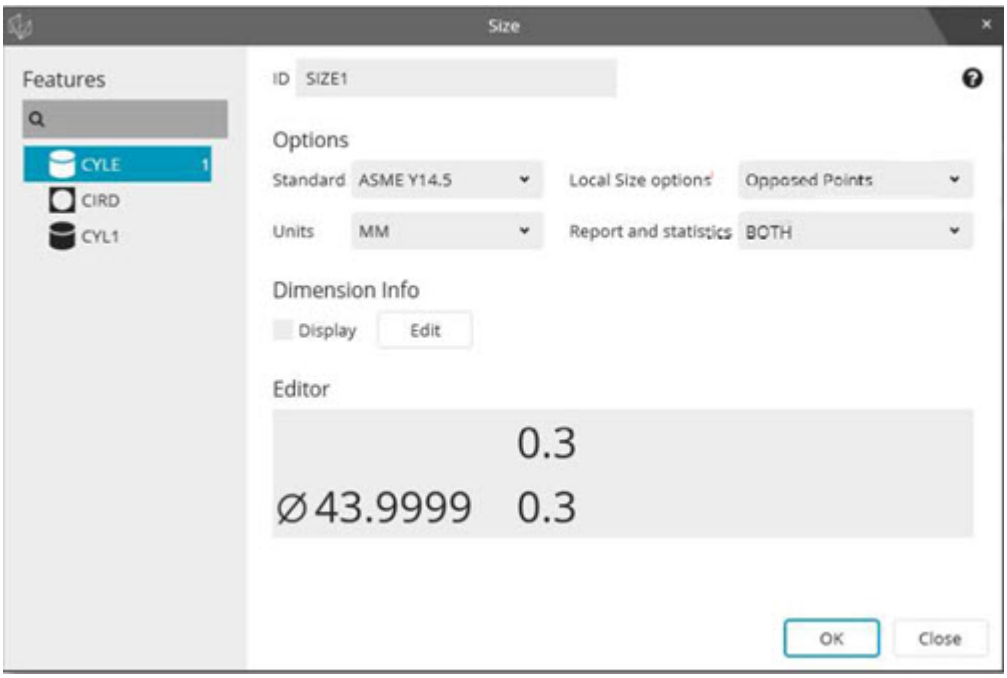
C. **UAME - ⌀ 43.8849**

D. **Local Size (Opposed Points) - ⌀ 44.2656**

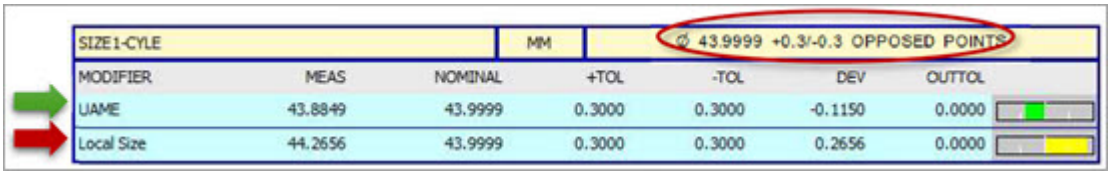
> The symbol "⌀" denotes **Diameter**.

For an inner diameter (ID) feature using the ASME Y14.5 standard, you can see from the above image that:

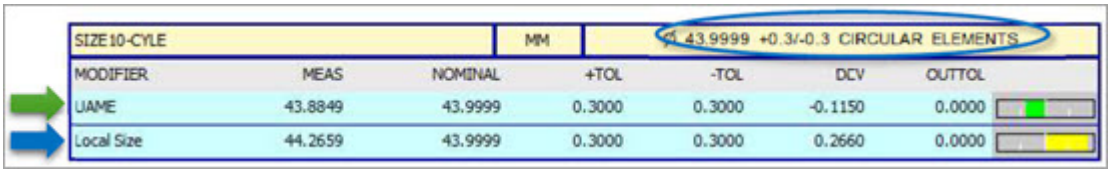
- The Unrelated Actual Mating Envelope (UAME) is the largest possible inscribed circle.
- The Local Size is based on which option you select from the **Local Size options** list of the Geometric Tolerance **Size** dialog box (see descriptions above).



If you select **Opposed Points** from the **Local Size options** list as shown in the Geometric Tolerance **Size** dialog box image above, PC-DMIS reports the result as:



If you select the **Circular Elements** option, PC-DMIS reports the feature as:

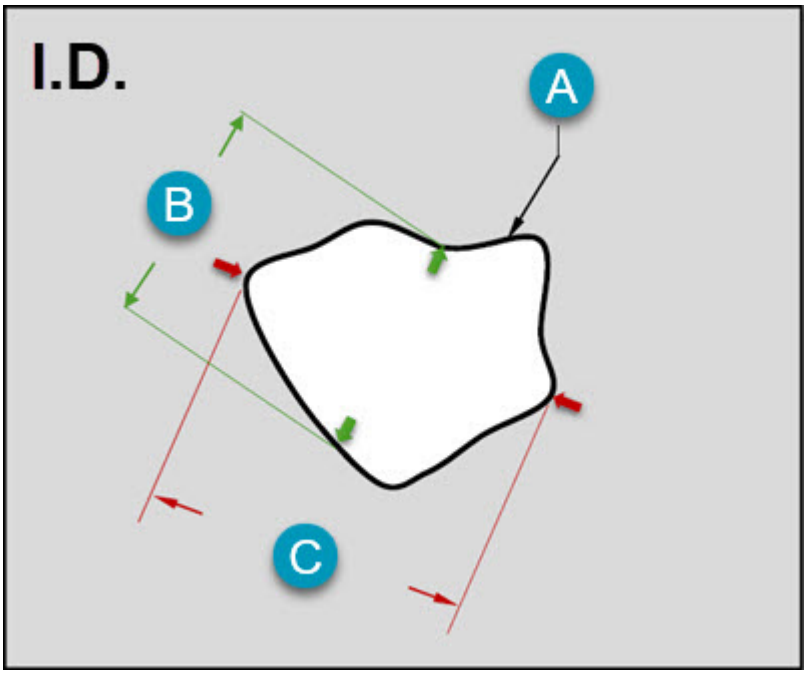


SIZE 10-CYLE		MM	Ø 43.9999 +0.3/-0.3 CIRCULAR ELEMENTS			
MODIFIER	MEAS	NOMINAL	+TOL	-TOL	DCV	OUTTOL
UAME	43.8849	43.9999	0.3000	0.3000	-0.1150	0.0000
Local Size	44.2659	43.9999	0.3000	0.3000	0.2660	0.0000

In all cases where details related to the ASME standards are concerned, you should consult the primary source at The American Society of Mechanical Engineers (ASME) website.

ISO 1101 - Local Size

When PC-DMIS reports Size using the ISO standard, it is based on Local Size only. Because of the Independency Rule, it is not based on the unrelated or related mating envelope. For details on the ISO Independency Rule, see ISO 8015: 2011 section 5.5 “Principle of Independence”.

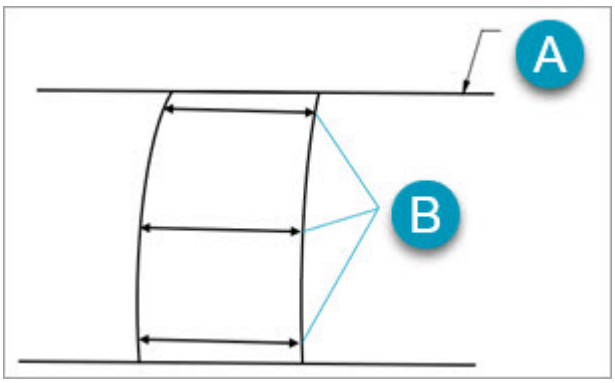


A. ***True shape of the feature's cross section***

B. ***Minimum Local 2-Point Size***

C. ***Maximum Local 2-Point Size***

For example, when you measure a cylinder feature at multiple levels, PC-DMIS evaluates each cross section individually and then reports the Maximum and Minimum 2-point size, similar to performing a caliper check.



A. *Datum feature simulator of datum feature A (Plane)*

B. *Actual local sizes (Any individual distance at any cross section of a feature of size)*

> **!** In all cases where details related to the ISO 1101 standards are concerned, you should consult the primary source at The International Organization for Standardization (ISO) website.

Supported ISO 14405-1 Modifiers

The Size command supports the following modifiers that are defined in the ISO 14405-1 standard:

- (LP) - Two-point size
- (GG) - Least-squares association criterion
- (GX) - Maximum inscribed association criterion
- (GN) - Minimum circumscribed association criterion
- (CC) - Circumference diameter (calculated size)
- (CA) - Area diameter (calculated size)
- (CV) - Volume diameter (calculated size)
- (SX) - Maximum size
- (SN) - Minimum size
- (SA) - Average size
- (SM) - Median size
- (SD) - Mid-range size
- (SR) - Range of sizes
- (E) - Envelope requirement
- ACS - Any cross section or sections
- SCS - Specific fixed cross section or sections

Using GD&T Selection Modes to Create FCFs

PC-DMIS provides these ways to get Feature Control Frames (FCFs) into PC-DMIS from GD&T tolerances in other sources:

GD&T Selection Mode (from CAD) ( ) - This enables you to import GD&T callouts embedded in your CAD model. In this mode you can box select or click on callouts to import them. For more information on this option, see "Using GD&T Selection Mode (from CAD)".

> You can access **GD&T Selection Mode (from CAD)** From the **Graphic Modes** toolbar or the **QuickMeasure** toolbar.

GD&T Selection Mode (from File) ( ) - PC-DMIS lets you import a drawing and decide what GD&T tolerances to import. This process uses Optical Character Recognition (OCR). For information on this option, see "Using GD&T Selection Mode (From File)". For more information, see "About Optical Character Recognition (OCR) to Create FCFs" below.

You can access **GD&T Selection Mode (from File)** from one of these:

- **Graphic Modes** toolbar
- **QuickMeasure** toolbar
- **File | Import** menu

Using GD&T Selection Mode (from CAD)

GD&T Selection Mode (from CAD) ( ) imports selected GD&T callouts from CAD models that contain them as dynamically-generated Geometric Tolerance dimensions or datum definitions.

For information on how to do this, see the "Importing CAD GD&T Callouts" heading within the "Working with CAD GD&T Callouts" topic of the "Editing the CAD Display" chapter.

Using GD&T Selection Mode (from File)

PC-DMIS can recognize and import GD&T tolerances on blueprints into your measurement routine with the **GD&T Selection Mode (from File)** ( ) option.

PC-DMIS uses Optical Character Recognition (OCR) to import the GD&T tolerances. PC-DMIS can import a .pdf file or an image file.

You can find sample blueprint .pdf files that correspond to the Hexagon demo-block CAD models, in the **Training** subfolder where you installed PC-DMIS.

When you import a blueprint file, the software parses the content of the file. It then shows the **GD&T from Capture** window and applies an orange highlight to all the supported items that you can import.

You then decide what supported items to import:

- One item at a time - To do this, click on an orange-colored tolerance.
- Multiple items - To do this, box-select multiple orange-colored tolerances.
- All items on a page - To do this, from the toolbar on the **GD&T from capture** window, click the **Process whole page** ( ) option.

If you choose to process multiple supported items or all supported items on a page, the software opens the OCR widget and cycles through each GD&T tolerance from the selection.

Be aware that OCR recognition does not handle every item in your file. For information on what items are supported and unsupported, see the "About Optical Character Recognition (OCR) to Create FCFs" topic in the PC-DMIS Core documentation.

PC-DMIS also parses measurement units and default tolerances from the title block and applies a light-blue highlight to them. For more information on the title block see, "About Optical Character Recognition (OCR) to Create FCFs". If PC-DMIS does not correctly determine the default tolerance values from the title block, you can use the pause button on the widget and manually fix a tolerance on any created item. For information on how to use the pause button, see "OCR Widget".

Procedure

This procedure explains how to import one or more callouts.

1. From the **Graphic Modes** toolbar, or the **QuickMeasure** toolbar, click the **GD&T Selection Mode (from File)** ( ) option.
2. Import a CAD model into the Graphic Display window that corresponds with your blueprint.

3. From the **Open** dialog box, navigate to an electronic blueprint file. This can be an image file or a .pdf file. OCR detection works best with an image that has at least 300 DPI. Lower resolutions may produce less accurate results.

4. Select the file, and click **Open** to parse the file and show all the recognized contents as orange-highlighted items in the **GD&T from Capture** window.

> Note that with this window open, the Edit window is not initially available for selection. This means you cannot delete features that you create unless you create them with the OCR widget or unless you click the pause button on the widget to pause the import process.

5. Do one of the following to import one or more items into PC-DMIS and to show the OCR widget:

   - Click on a single callout. This can be a GD&T tolerance or a basic linear distance, angle distance, or location dimension.
   - Box-select multiple callouts on the page.
   - From the toolbar on the GD&T from capture window, click **Process whole page** ( ).

6. Follow the instructions in the OCR widget.

> During the procedure, you need to pick features for datums or input features for the geometric tolerances. You can create these features with QuickFeatures. You can also select existing features from the Graphic Display window if you click on their feature IDs. PC-DMIS does not support clicking in the Edit window to select features.

Characteristic IDs

If your imported file has characteristic IDs, and you enabled the **Use Characteristic ID** check box from the **General** tab of the **Setup Options** dialog box, PC-DMIS automatically assigns an ID in the OCR widget that matches the characteristic ID. Any features that you then define with the OCR widget for the callout next to that ID use this naming convention:

<BalloonID>__1, <BalloonID>__2, and so on.

For example, if you have a characteristic ID of 3 in your file, and you need to create two features for the tolerance with that ID, PC-DMIS gives the feature that you create the IDs of 3__1 and 3__2.

If a callout has a multiplier, such as a callout for a bolt-hole pattern, PC-DMIS assigns the characteristic ID for the first feature or dimension in the pattern. Any other features or dimensions in that pattern follow the default naming convention for that feature or dimension.

Datums Target Definition

If the OCR widget asks you to define datum target points, follow this procedure:

1. Press Ctrl + Shift and click on the CAD to create a vector point QuickFeature for each target.
2. As you define each datum target, the vector point feature appears in the Edit window.
3. Once you define the targets for a datum feature, it shows "Completed" for that datum feature. The OCR parsing of the file may not determine the number of targets needed. If that happens, you can create additional vector point features even if the targets for a datum feature show "Completed".
4. When you're done creating all the points for a target, click **Next** to go to the next set of datum targets.
5. Continue to define datum targets until the text on the widget says "Datum Target Definition Complete".
6. At this point, you can use the **Pause** button ( ) on the widget to pause the GD&T import process so that you can work with PC-DMIS to finish defining the datums. This can be a complex process that involves alignments, constructed features, or doing other work in PC-DMIS.
7. Use the Edit window, and press F9 on each Datum Definition (DATDEF command in Command Mode) to show the **Datum Definition** dialog box.
8. Link the datum targets to the datum letter.
9. Once you use PC-DMIS to define any datums from the targets, click **Resume** ( ) on the widget.

Datums Definition

If the OCR widget shows **Datums definition**, you need to use QuickFeatures to create datum features for those Geometric Tolerance commands. (If you've already created a datum feature, you can click on its feature ID from the Graphic Display window to select it.)

1. Create or select a datum feature or click **Skip current feature definition** ( ) if you want to skip over single datum definition for now.
2. Click **Next** and select additional datum features.

3. Continue repeating the above steps until the widget shows "Datums Definition Completed".

4. Then click **Apply** to define the remaining GD&T information.

Tolerances Definition

If the OCR widget shows **GEOTOL1**, you need to define the remaining GD&T information. If you skipped any datum definitions from the previous procedure, the widget asks you to define them now during the GD&T definition.

1. Once datums are completely defined, you need to create or select the actual considered feature or features. Use QuickFeatures to create the minimum number of features from the CAD model for the current step. (If you've already created a considered feature, you can click on its feature ID from the Graphic Display window to select it).

2. If you don't want to import a GD&T tolerance, click **Skip GEOTOL1** ( ). You can go back to a skipped item with the **Back** button ( ) as long as it is not on an earlier .pdf page.

3. Once you define a feature or features from the CAD model for a step, from the widget, click **Next** ( ). This continues with the next step of the GD&T definition. The **Next** button becomes enabled once you've defined the minimum number of features.

4. Once you define the features, and the widget informs you that the tolerances definition is completed, click **Apply** ( ). This accepts your changes and moves onto the next GD&T tolerance in the GD&T from Capture window.

5. Continue repeating the above steps until you have defined all selected GD&T tolerances.

Basic Linear Distance, Angle Distance, or Location Definitions

When you import basic dimensions, such as linear distance, angle distance, or location, you need to select or create features as usual to fulfill the definition requirements for these basic dimensions.

For the linear distance or angle distance dimensions, this can be a complex process that involves alignments, constructed features, or doing other work in PC-DMIS. In those cases, use the available **Pause** button to help you. Here's the procedure for those:

1. For those two dimension types, select the distance or angle features.

2. On the widget, click **Next** ( ). This continues with the next step of the definition and the widget says "Tolerances definition completed".
3. At this point, if you need to do something in PC-DMIS to fulfill the definition, click the small **Pause** button ( ) on the OCR widget. This temporarily stops the import process and gives you nearly full control over PC-DMIS.
4. Do whatever work you need to do in PC-DMIS to fulfill the callout definition.
5. On the widget, click the **Resume** button ( ) to resume the import process and click **Apply** ( ) to accept the definition. If the callouts have tolerance values defined in the blueprint, those values are also parsed and assigned to any created dimension in the Edit window.

Best Practices

OCR detection works best with an image that has at least 300 DPI. Lower resolutions may produce less accurate results.

When you have imported the desired GD&T tolerances, double-check the geometric tolerance commands that PC-DMIS generated.

Related Topics:

[About Optical Character Recognition (OCR) to Create FCFs]

[GD&T from Capture Window]

[OCR Widget]

About Optical Character Recognition (OCR) to Create Geometric Tolerances

The **GD&T Selection Mode (from File)** ( ) method uses OCR to identify the GD&T tolerance information to import.

For more information, see the "[Using GD&T Selection Mode (from File)]" topic.

Resolution

OCR detection works best with an image that has at least 300 DPI. Lower resolutions may produce less accurate results.

Supported Geometric Tolerance Types

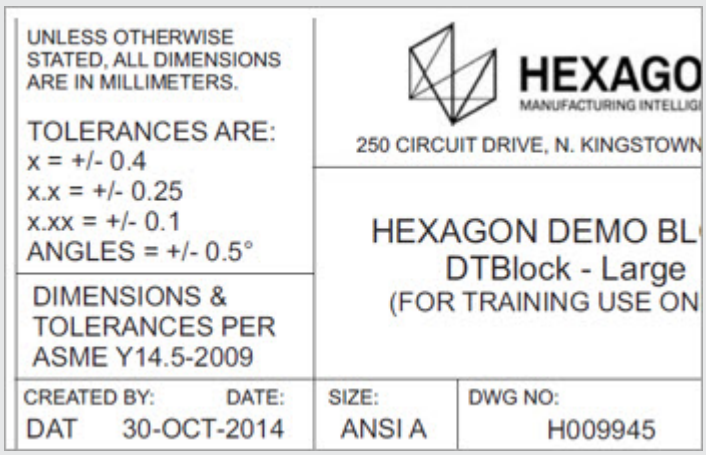
The described OCR methods support these Geometric Tolerance types:

- The 14 characteristic symbols
- The Diameter symbol

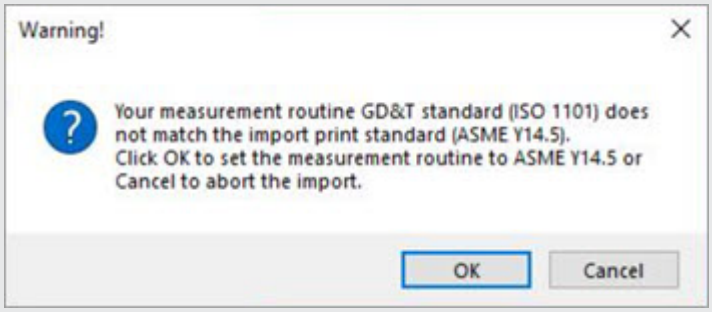
- M, L, P, and other modifier symbols bounded by circles
- Data within the boxes and size data above boxes
- Single-line tolerances, multi-line non-composite tolerances, composite tolerances
- Linear, Angular, and Location dimensions

PC-DMIS highlights supported items you can interact with in orange.

When you import GD&T through OCR, PC-DMIS checks the title block text to see if a GD&T standard is declared.



If there is, PC-DMIS compares it to what is defined in the measurement routine. If the standards do not match, PC-DMIS displays a warning message asking if you want to continue or cancel the import.

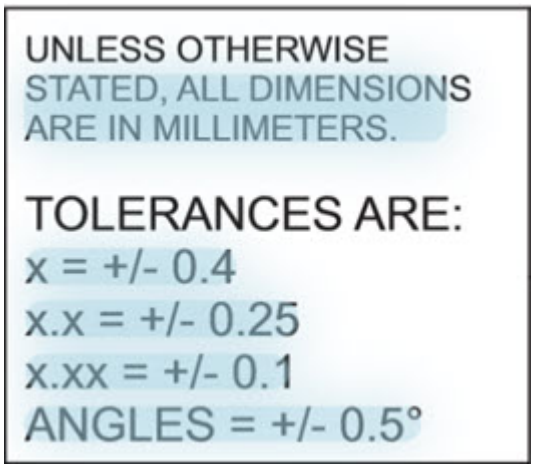


Example Warning message showing mismatch between current and imported GD&T standards

- If you click **OK**, PC-DMIS applies the GD&T standard called out in the title block and sets all existing Geometric Tolerance commands in the measurement routine to be invalid. You can use the invalid commands to recreate new versions of the commands based on the updated GD&T standard. You can then delete the invalid commands after you recreate the new commands.
- If you click **Cancel**, PC-DMIS cancels the import, and nothing changes in your measurement routine.

Other Supported Items

- OCR detection also parses some default tolerances and measurement units from the title block. This happens automatically, and PC-DMIS visually indicates that it has been parsed by highlighting the title block text in a light blue color, like this:



Example of a title block with parsed text (Highlighted in blue).

PC-DMIS compares the measurement units on the blueprint against the measurement units of the routine. If they are different, PC-DMIS converts the blueprint's tolerance values into the units used by the routine. If the blueprint doesn't clearly state the measurement units, PC-DMIS assumes that the tolerances match the measurement units of the routine. PC-DMIS applies the parsed default tolerances to any dimension on the drawing that doesn't have tolerances already explicitly defined.

- OCR detection can also support characteristic IDs. For that to work, you need to enable the **Use characteristic ID naming** check box from the **General** tab of the **Setup Options** dialog box prior to performing the OCR detection. For more information on that check box, see "Use Characteristic ID Naming" in the "Setting Your Preferences" chapter. For information on OCR detection of characteristic IDs, see "Using GD&T Selection Mode (from File)" where the topic discusses characteristic IDs.

Unsupported Items

The OCR methods above do not support some complex elements. These include items like square brackets, extended characters, double-headed arrows, and so on.

FAQ

Question: What happens if my measurement routine already has a datum defined with the same label as a datum defined in the drawing?

Answer: If a datum definition with the same name already exists in your measurement routine, PC-DMIS does not request that you to create a new datum feature of the same name. Instead, PC-DMIS uses the existing datum feature from your measurement routine.

Question: Do I need to use OCR to recognize a datum all by itself?

Answer: No. Since datums are always defined for a GD&T tolerance, the OCR does not recognize them as separate GD&T notes. You only need define the GD&T tolerances and PC-DMIS prompts you to select the datum features as needed.

Question: What happens if there is size text immediately above the GD&T tolerance?

> *Answer: PC-DMIS parses this additional line and assigns the data to the created Geometric Tolerance.*

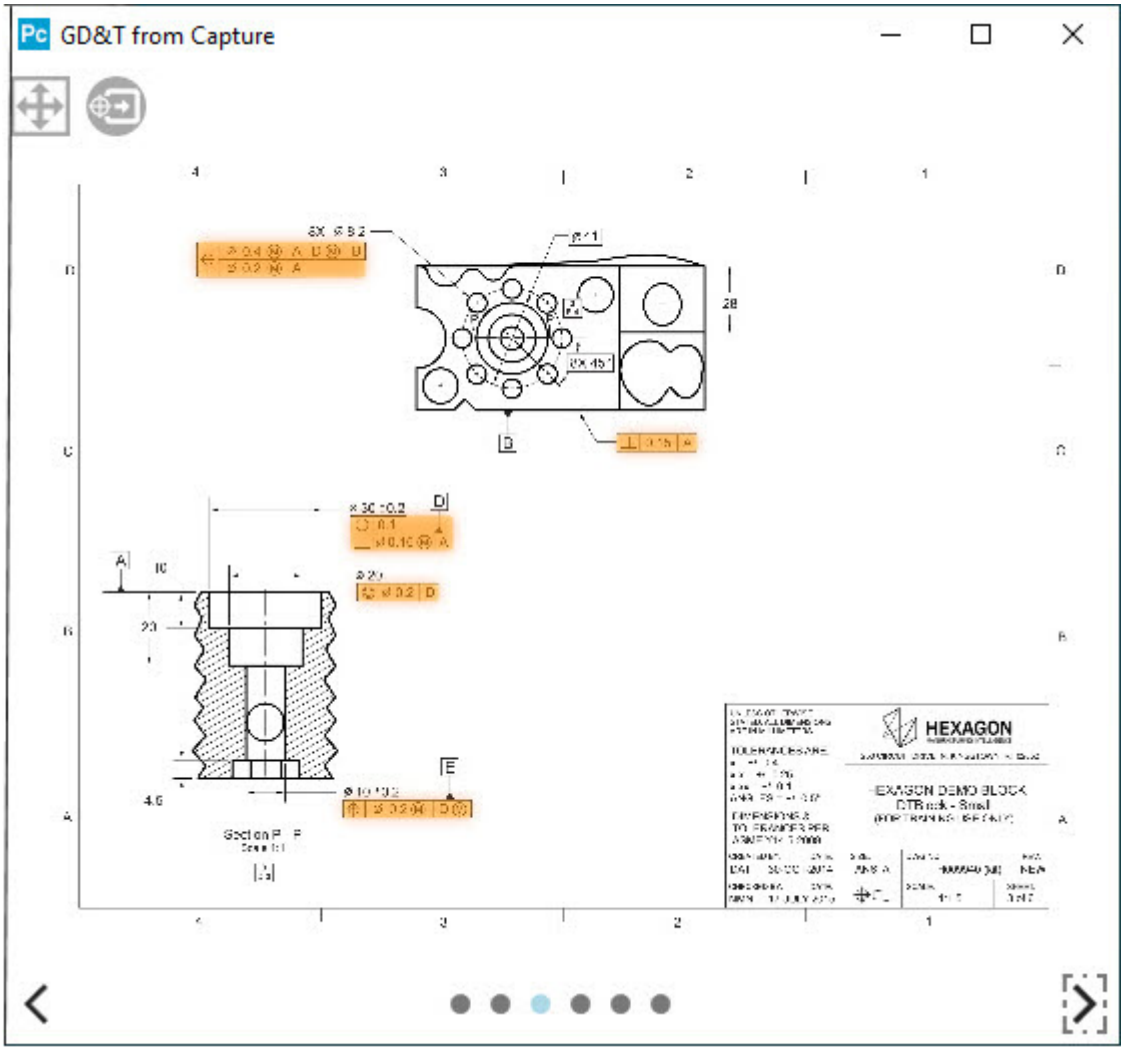
Question: What happens to multi-line non-composite GD&T tolerances?

> *Answer: PC-DMIS imports these as two separate Geometric Tolerance commands.*

GD&T from Capture Window

When you import an image or .pdf file that contains GD&T tolerance information with the **GD&T Selection Mode (from File)** ( ) option, PC-DMIS uses Optical Character Recognition (OCR) to process the file.

PC-DMIS then displays the **GD&T from Capture** window and the detected tolerances:



GD&T from Capture window with detected GD&T tolerances in orange

This is a re-sizeable and movable window. You can double-click the title bar to maximize the window or to restore it to its previous size. The image on each page of the file scale to fit the window.

Screen Elements

Fit to page - This element sizes the contents of the current page to the dimensions of the window.

Process whole page - This element processes all highlighted orange GD&T tolerances on all pages.

Orange highlight - A GD&T tolerance with this color means it has been identified by OCR and can be imported into the routine.

Yellow highlight - A GD&T tolerance with this color means it is the active tolerance and that the OCR widget is waiting for you to take action to proceed. The software creates a temporary GD&T callout in the Graphic Display window, and the OCR widget provides some brief instructions on how to proceed.

Green highlight - A GD&T tolerance with this color has been processed, and an appropriate Geometric Tolerance or other similar command exists for it the routine.

- The gray dots at the bottom of the window show the total pages. The blue dot shows which page is the current page.

- The buttons at the bottom of the window let you go to next and previous pages.

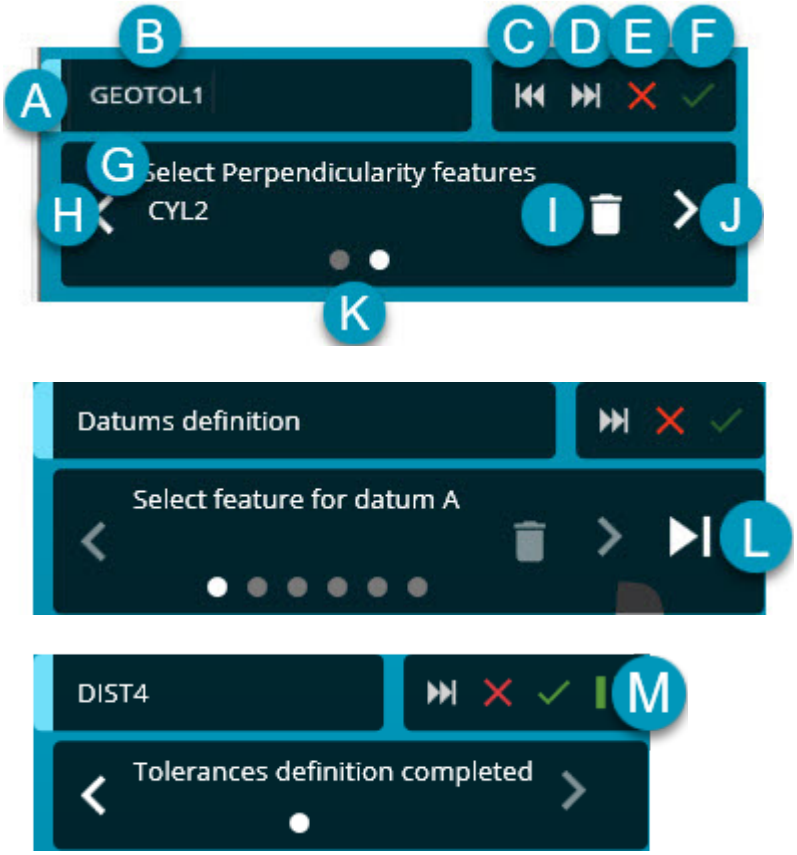
Panning and Zooming

Just like in the Graphic Display window, you can use the mouse wheel to zoom in and out on the drawing. You can right-mouse drag to pan the drawing.

OCR Widget

PC-DMIS uses this widget to import GD&T tolerances from a drawing with Optical Character Recognition (OCR). The widget appears if you select **GD&T Selection Mode (from File)** ( ), and you then select one or more GD&T tolerances to import from the GD&T from Capture window.

There are often multiple steps with instructions on the OCR widget for a given GD&T tolerance definition. For example, you often need to first select a datum feature (or features) from the CAD model and later select the considered feature (or features). These instructions and your selections occur across multiple steps.



Example of the OCR Widget for the highlighted GD&T tolerance.

A. This handle lets you drag and reposition the widget.

B. **Feature** - This box defines the name of the GD&T tolerance or step.

C. **Back** - This button only appears if there are multiple GD&T tolerances on a .pdf page or a capture of multiple tolerances, and you have skipped one or more of them with the **Skip** button. If you then click **Back**, the software returns to the skipped tolerance and highlights it in the **Selection Preview** window.

D. **Skip** - This button appears with a .pdf import or with a capture of more than one GD&T tolerance. By default, for each tolerance, you define the datums first (or datum targets). In this case, this button skips the current datum definition (or datums target definition). Once you move onto defining the rest of the GD&T tolerance, this button skips the highlighted GD&T tolerance. It then highlights the next available GD&T tolerance on a .pdf page or capture. For a file import, if there's only one GD&T tolerance on a page, the software asks if you want to proceed to the next page. If there are no more .pdf pages with GD&T tolerances, the software asks if you want to exit the GD&T import process. For an imported file, if you need to go back to a skipped item on a previous page, you must start the file import again.

E. **Cancel** - This button cancels the OCR process and closes the OCR widget and the **Selection Preview** window.

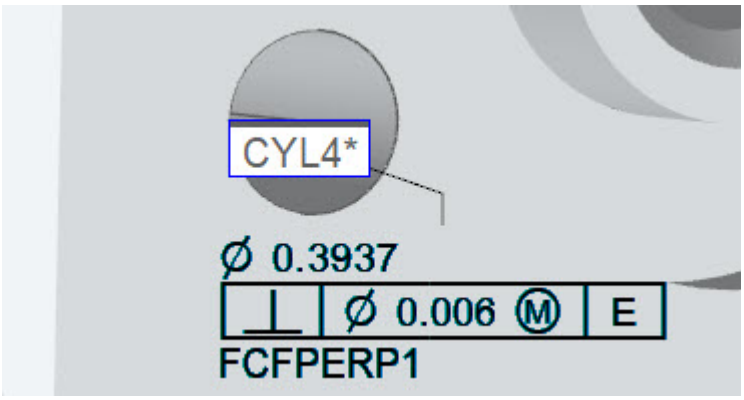
F. **Apply** - This button accepts the selected datum targets or features for the GD&T tolerance and moves onto the next part of the import or onto the next tolerance.

G. This field shows the instructions for the current step. You need to use QuickFeatures to select the indicated feature in the Graphic Display window.

H. **Previous** - This button goes back one step.

I. **Remove** - This button removes the selected feature from the current step.

J. **Next** - This button accepts the selection for the step and goes to the next step in the definition. The software enables this button once you define the minimum number of features. The temporary callout in the Graphic Display window updates as you progress through the definition:



Example of a temporary callout.

K. These dots show the number of steps to define one or more datums or to define a single GD&T tolerance. The white dot indicates the current step.

L. **Skip current feature definition** - This skips over the current datum definition.

M. **Pause** / **Resume** - The **Pause** button ( ) only appears if you are processing these types of callouts:

- Datum targets
- Linear distance dimensions
- Angle distance dimensions

To define these items can be a complex process. You may need to work with alignments, constructed features, or do other work in PC-DMIS. The **Pause** button is useful because it pauses the import process so that you can have nearly full access to PC-DMIS to finalize the callout's definition.

For Datum targets, the software enables this button once you have defined all the various datum target points, and the widget shows "Datums Target Definition Completed".

For the distance dimensions, it appears once you have defined both features, and the widget shows "Tolerances Definition completed".

Once you fulfill the callout's definition, you can click **Resume** ( ) to continue the import process.

Troubleshooting Error Messages and Warnings

PC-DMIS helps you to properly create your geometric tolerance command by providing many error messages and warnings. These messages may help you understand problems with your measurement routine. This topic provides additional details on many of these messages.

Message	Description	Solution
Warning: 2D primary datums do not constrain orientation so the datum feature's view will be used as an implied primary datum.	This is a warning message and not an error. Circles, lines, and 2D widths are 2D features, and are not able to constrain the orientation sufficiently. They are not recommended as primary datums. When you use a 2D feature as a primary datum, PC-DMIS levels to the view of the primary datum. In other words, the view becomes an implicit primary datum, and the specified primary datum becomes a secondary datum. For more information, see "How PC-DMIS Solves and Uses Datums".	You can continue to use geometric tolerances normally with this warning message. However, we recommend that you use a 3D primary datum instead, such as a plane, cylinder, 3D width, or sphere.

Warning: this type of constructed point does not have surface information. Therefore, it is treated similarly to the center of a sphere. Please use caution to ensure the results represent the specification.	This is a warning message and not an error. This warning is visible during these conditions: • When the constructed point is treated as a surfaceless 3D point. • When the point is used as a datum or as a considered feature. As discussed in "Structuring your Measurement Routine for Geometric Tolerances", in most cases we do not recommend using surfaceless 3D points, because you are taking over phases 2 and 3 of the conceptual evaluation process. If you do so, it is your responsibility to construct the feature according to appropriate standards. Because the point is treated like a sphere center, it is easy to get undesired behavior. (More Info) For more information about datum handling with surfaceless 3D points, see How PC-DMIS Solves Datums.	You can continue to use geometric tolerances normally with this warning message. However, in most cases we recommend that you use a feature that preserves the surface information. This way the geometric tolerance command can guarantee compliance to the applicable standard.

	For more information about considered feature handling with surfaceless 3D points, see Deriving the Toleranced Feature. For more information about the conceptual evaluation process, see Introduction to Geometric Tolerances and Feature Control Frames.	
Warning: simultaneous profile of a line tolerances are treated the same as profile of a surface.	This is a warning message and not an error. Profile of a line tolerances have a different meaning than profile of a surface tolerances, and it doesn't make sense (strictly speaking) to consider profile of a line tolerances simultaneously. For more information, see "Profile of a Line". However, PC-DMIS permits profile of a line tolerances to be included in simultaneous tolerances. It does this by treating them the same as profile of a surface.	You can continue to use geometric tolerances normally with this warning message. However, in most cases we recommend the following: • When cross sections are considered individually, use profile of a line specifications. • When several profile tolerances need to be considered simultaneously, use profile of a surface.

Warning: the [DF] modifier does not exist in ISO 5459:2011. The [DF] modifier (distance fixed) adds a location constraint to the datum. No [DF] modifier removes the location constraint.	This is a warning message and not an error. The [DF] modifier is not standardized as described in "Modifiers_for_Datums". However, it is functionally necessary in certain kinds of datum reference frames, and so PC-DMIS permits using this non-standardized modifier. For more details about how [DF] behaves, please refer to "Modifiers_for_Datums".	You can continue to use geometric tolerances normally with this warning message. However, in most cases we recommend that you do the following: • Contact your country's representatives to the ISO TC/213 family of standards, letting them know you need the [DF] modifier to be standardized in ISO 5459. • Verify that the functionality achieved with the [DF] modifier satisfies your functional needs.

Composite position specifications require more than one feature.	Composite position specifications are intended to control the location of a pattern to itself. It doesn't make sense to specify composite position of a single feature.	We suggest that you either: • Ensure your composite position specification includes at least two features. • Change your composite position specification to be two separate position specifications. • Or replace the lower segment of the composite position specification with an orientation specification instead, because the lower segments of composite position specifications do not have location constraints to the datum reference frame.

Simultaneous profile of a line tolerances without any datums are not supported.	As discussed in "Profile of a Line", it doesn't make sense (strictly speaking) to include profile of a line tolerances in simultaneous tolerance commands. If there are no datums referenced, then it makes even less sense because there are no datums or workplanes that you can select to control the orientation of the cross sections. Therefore, profile of a line specifications, without any datums inside simultaneous tolerance commands, are not supported.	We suggest that you either: • Change your profile of a line specification to be profile of a surface. • Or ensure that your profile of a line specifications are not considered simultaneously.

Datum feature <feature name> is 2D. It needs a higher-precedence datum to constrain its workplane.	**Case 1:** As discussed in "How PC-DMIS Solves and Uses Datums", some feature types are 2D, and their workplane needs to be constrained by higher-precedence datums. **Case 2:** Sometimes this error occurs because a constructed 3D BFRE line was used as a secondary datum. It is very common for the THEO line vector of these lines to be non-parallel to the primary datum plane. This means that the nominal workplane of the line is not parallel to the primary datum plane, and so the primary datum plane does not constrain the workplane of the secondary datum line.	For **Case 1**, the two most common solutions to this problem are: (1) measure the datum feature as a 3D feature, and (2) use one or more higher-precedence datums to constrain the datum feature's workplane. For **Case 2**, change the constructed 3D BFRE line to be a constructed 2D BFRE line, so that the nominal workplane of the line is parallel to the primary datum plane.

		For Best Fit (BF) or Best Fit Recompensate (BFRE) constructions, while you can use any feature type for your input features, BF and BFRE fit types are typically used with point features or point sets (a scan of points, a feature set with points, or an expression that resolves to an array of points). For details on using the Best Fit and Best Fit Recompensate methods to construct features, see the "Understanding Best Fit (BF) and Best Fit Recompensate (BFRE) Constructions" topic in the PC-DMIS Core documentation.

Feature <feature name> has too few points to fit uniquely.	This error indicates that the considered feature doesn't have enough points for a unique fitted shape. For example, PC-DMIS cannot uniquely fit a cylinder with only four surface points. Features with less than the absolute minimum number of points produce this error. (More Info) Generally, we recommend that you measure features as densely as practical. However, the absolute minimum number of points for each kind of feature are listed here: • Plane: 3 points • Surface line: 2 points • Surface circle: 3 points • Cylinder: 5 points • Cone: 6 points • Sphere: 4 points • 3D width: 4 points • 2D width: 3 points	Increase the number of points measured.

	Features with the absolute minimum number of points have all deviations equal to zero (with some exceptions too rare and complex to document here). Therefore, such features have zero measured form error.	

Datum <datum reference> has too few points to fit uniquely.	The most common way to get this error is when the surface points of the feature are not placed in a way that constrain any degrees of freedom. (More Info)	Measure the datum feature as a full 3D feature (plane, cylinder, cone, sphere, 3D width, and so on) instead of as a 2D or 1D feature (surface line, surface circle, 2D width, surface point, 1D width). If that is not possible, ensure that the datum feature and its surface points actually do constrain the necessary degrees of freedom.

		Suppose a datum reference frame has a primary datum plane, with surface normal Z-plus. The secondary datum is a cylinder with axis vector Z-plus, and the tertiary datum is a surface point. The orientation of the tertiary surface point determines whether this error occurs or not. If the surface normal of the point is parallel to the vector between the cylinder and the surface point, you will get this error because the point will not constrain rotation about the cylinder. However, if the surface normal of the point is any other direction, you will not get this error.

	You can also get this error if your datum feature has less than the absolute minimum number of points, but that is not a common way to get this error.	

Freeform feature(s) need points in more places to fit uniquely	You get this error when the geometric tolerance command cannot determine the degrees of freedom that are optimizable for the freeform-considered features, subject to the datum reference frame. To know those feature command types that are considered freeform, see the "Feature Types With and Without Surface Data" topic of the PC-DMIS Core documentation. One of the ways you can get this error is if you don't measure enough of the full surface to help the geometric tolerance command understand the surface. Perhaps you only measured a single point on the surface, or only one cross section.	You can fix the error in these ways: • Sample more of the surface. For example, if you measured only in one cross section, measure in more than one cross section. • Make sure all of the nominal vectors of your measured points are exactly correct. • If the nominal surface is almost symmetric, such that the degrees of freedom are unclear, constrain the uncertain degrees of freedom with additional datums in the datum reference frame. • If you don't have any datums referenced, ensure that the chosen workplane (under the **Reporting** tab of the **Geometric Tolerance** dialog box) matches the 2D considered feature's workplane.

	Another way you can get this error is when the nominal vectors of your measured points are not theoretically correct. For example, if you measure a plane, but the nominal vectors are not exactly planar. In this case, the geometric tolerance command determines that the surface isn't quite planar, but it is not able to determine whether the surface is cylindrical, spherical, conical, or complex. A third way you can get this error is when the nominal surface is almost symmetric in some way, but not quite. For example, perhaps it is almost planar, or almost cylindrical. In these cases, the geometric tolerance command cannot determine the optimal degrees of freedom. Finally, this error can also occur for profile of a line to no datums when you select an incorrect workplane.	

Freeform feature(s) need(s) points in more places to fit uniquely.	You get this error when the geometric tolerance command cannot determine the degrees of freedom that are optimizable for the freeform considered features, subject to the datum reference frame. To know those feature command types that are considered freeform, see the "Feature Types With and Without Surface Data" topic. One of the ways you can get this error is if you don't measure enough of the full surface to help the geometric tolerance command understand the surface. Perhaps you only measured a single point on the surface, or only one cross section. Another way you can get this error is when the nominal vectors of your measured points are not theoretically correct. For example, if you measure a plane, but the nominal vectors are not exactly planar. In this case, the geometric tolerance command thinks the surface isn't quite planar, but it won't be able to tell whether the surface is cylindrical, spherical, conical, or complex.	You can fix the error in these ways: • Sample more of the surface. For example, if you measured only in one cross section, measure in more than one cross section. • Make sure all of the nominal vectors of your measured points are exactly correct. • If the nominal surface is almost symmetric, such that the degrees of freedom are unclear, constrain the uncertain degrees of freedom with additional datums in the datum reference frame.

	A third way you can get this error is when the nominal surface is almost symmetric in some way, but not quite. For example, perhaps it is almost planar, or almost cylindrical. In these cases, the geometric tolerance command cannot determine the optimal degrees of freedom.	
Profile of a line tolerances need their workplane defined by the datum reference frame.	Profile of a line tolerances have 2D considered features. Their workplane needs to be constrained by the datum reference frame.	*If you have one or more datums referenced*, ensure that the datum reference frame constrains the workplane of the 2D considered features. *If you don't have any datums referenced*, ensure that the chosen workplane (under the Reporting tab in the dialog box) matches the 2D considered features' workplane.
This tolerance requires the data to be measured in circular cross sections.	Some tolerances, such as circularity of a cylinder, or the CIRCULAR_ELEMENTS local size interpretation, need their data to be measured in circular cross sections.	Re-measure the considered feature so that the data is arranged in circles. You may find a measurement strategy helpful, but you do not have to use one.

This local size tolerance requires the data to be measured in circular cross sections. To Fix this either measure the feature using circular cross sections or turn off local size in the Report tab	When reporting position, the orientation and runout of some local size tolerances (such as ASME CIRCULAR_ELEMENTS) require that you measure the data in circular cross sections.	Turn off local size (if not required) or re-measure the considered feature so the data is arranged in circular cross sections.
The tolerance needs at least one datum.	Several tolerances, such as perpendicularity, require at least one datum.	Add a primary datum to the tolerance.
A multi-feature datum cannot mix features with surface data and features without surface data.	Multi-feature datums include datum patterns and common datums. They must all either have surface data or all lack surface data. For information on feature command types that have and do not have surface data, see "Feature Types With and Without Surface Data".	Choose datums that all have surface data or choose datums that do not have any surface data.
Features' orientation must be compatible with their polar tolerance zones.	Polar tolerances zones include radial-arc tolerance zones and right-angle-to-radial tolerance zones. The considered features must be nominally parallel to the polar axis defined by the datum reference frame.	Ensure that all of your considered features are nominally parallel to the polar axis, or stop using polar tolerance zones.

Features with polar tolerance zones may not center on the polar origin.	Polar tolerances zones include radial-arc tolerance zones and right-angle-to-radial tolerance zones.	The considered features must not be coaxial with the polar axis. If you have a considered feature that is coaxial with the datum axis, you typically need to use a diametric tolerance zone instead of a polar tolerance zone.
The datum reference frame must define a clear polar origin.	Polar tolerances zones include radial-arc tolerance zones and right-angle-to-radial tolerance zones. They only make sense when the datum reference frame defines a clear polar axis.	Ensure the datum reference frame defines a clear polar axis.
Multi-feature datums at RMB must have surface data.	Multi-feature datums include datum patterns and common datums. If the multi-feature datum does not have a material modifier, then the datum features must have surface data.	Re-measure the datum features so they have surface data.
For perpendicularity tolerances, the nominal considered features must be perpendicular to the primary nominal datum.	The most common cause of this error is incorrect THEO values in the considered features and/or datum features.	Ensure that the considered features are nominally perpendicular to the primary datum.

For parallelism tolerances, the nominal considered features must be parallel to the primary nominal datum.	The most common cause of this error is incorrect THEO values in the considered features and/or datum features.	Ensure that the considered features are nominally parallel to the primary datum.
The orientation of a planar tolerance zone must be fully defined by the datum reference frame.	You get this error with planar tolerance zones on axial features such as cylinders, cones, and circles. Specifically, when the datum reference frame doesn't constrain the orientation of the tolerance zone, the actual value is not well defined.	You can fix the error in these ways: • Ensure that the datum reference frame fully constrains the orientation of the tolerance zone. • Use a diametric tolerance zone.
These simultaneous tolerances must use the DEFAULT tolerance zone math type.	You get this error when all these conditions are met: • You have multiple profile tolerances. • No datums belong to a simultaneous tolerance command. • The profile tolerances have different tolerance zone math types.	Set all the profile tolerance zone math types to **DEFAULT**.

Datums at RMB without surface data that follow datums at MMB or LMB are not supported.	The geometric tolerance command does allow datums without a material modifier to follow datums that have a material modifier. However, the no-material-modifier datum must have surface data.	Re-measure the lower-precedence datum so it has surface data.
When a tolerance specifies more than one considered feature, all the features must be pattern compatible. This means they must have the same type of shape, the same nominal size, and the same inner/outer.	When there are multiple considered features, they must be identical, except for having different locations and orientations. So, for example, cylinders must be all inner or all outer, and they must all have the same nominal size.	Use separate geometric tolerance commands for features that are not identical. If necessary, use a simultaneous tolerance command to consider all the features simultaneously.

Feature is used twice in fundamentally different ways, where the nominal shape (or lack thereof) must be considered differently in the two contexts.	This is a very unusual error. The most common way to get this error is with a simultaneous position and profile of a slot, notch, or ellipse: • Position treats the feature as a circle without surface data. • Profile treats the feature as a freeform feature with surface data. The error occurs because the simultaneous tolerance needs to treat the same feature two different ways at the same time.	You can fix the error in these ways: • Use only profile and not position to control the position of the slot. • Create a cast circle or generic circle from your slot, notch, or ellipse, and do simultaneous position of the cast circle with the profile of the slot.
Simultaneous tolerances must have identical datum reference frames.	This error occurs if geometric tolerances in your simultaneous tolerance have different reference frames; or the datums in those tolerances are not identical, are out of order, or use different modifiers.	Ensure that all geometric tolerances in your simultaneous tolerance command have identical datum reference frames. The datums must be the same, in the same order, with the same modifiers.
Simultaneous tolerances must be position or profile tolerances.	You get this error if you aren't using the correct geometric tolerances for the simultaneous tolerance.	Ensure all geometric tolerances in your simultaneous tolerance command are either position or profile tolerances.

Multi-feature datum error. This error may occur from incorrect nominals (X, Y, Z or I, J, K vectors) or an unsupported combination of features.	You get this error when you try to reference an unsupported combination of features as a common datum.	Review the table of supported feature combinations and general guidelines for common datums. Also, check the feature nominals for the multi-feature datum in question and make any corrections needed.
Specification requires a fully constrained tolerance zone.	Some tolerances, such as symmetry and concentricity, require that the tolerance zone is fully constrained by the datum reference frame.	Ensure that the datum reference frame fully constrains the tolerance zone.
Translation modifier is invalid because the datum does not have any translation degrees of freedom that could be unlocked.	There are many circumstances where the geometric tolerance command allows you to put a translation modifier on a datum where it does not make sense.	Remove the translation modifier from the datum.
Datum is invalid because it constrains no degrees of freedom.	If your secondary or tertiary datum constrains no degrees of freedom, this error occurs. Most likely, this either means (a) your print is wrong, or (b) there is a mistake in your measurement routine.	If this error happens with a new geometric tolerance, double-check your print and your feature types. Pay special attention to best fit lines, as discussed in Feature Types With and Without Surface Data.
Runout tolerances require features that are concentric with the datum reference frame.		Ensure that all considered features are nominally concentric with the datum reference frame.

Too many points for this type of calculation.	This error occurs when using a **DEFAULT** math option with too many points. It can happen due to the feature math option, the datum math option, or the tolerance zone math option. The cutoff for how many points is "too many" is several tens of thousands of points.	You can fix the error in these ways: • Use only a few thousand points. • Use the **LSQ** math option.
Concentricity tolerances require input features that are concentric with the datum reference frame.		Ensure all considered features are nominally concentric with the datum reference frame.
Symmetry tolerances require input features that are symmetric with the datum reference frame.		Ensure all considered features are nominally symmetric with the datum reference frame.
Invalid per-unit extent.		Ensure that the per-unit length and/or per-unit width is correct.
There is insufficient point density for the per-unit tolerance.		Re-measure the considered feature with increased point density.

The datum customization is invalid.	The geometric tolerance command allows customized ASME datum reference frames. This error occurs when the customization does not make sense mathematically. It is very common to make mistakes with customized DRFs, and that can result in this error.	You can fix the error in these ways: • Stop using datum customization. • For every datum in the datum reference frame, ensure that the customization results in a unique and fully-defined invariant.
Circles used to calculate the extracted median line must include at least 90 degrees of arc.	As discussed in "Deriving the Toleranced Feature", several ISO geometric tolerances use the extracted median line as the toleranced feature. All circular cross sections must have at least 90 degrees of arc. This error can also happen with straightness of an axis (either ASME or ISO). As discussed in the "Straightness" topic, straightness of an axis requires that you measure the surface data in circular cross sections. If any of the cross sections have less than 90 degrees of arc, PC-DMIS shows this error message.	Ensure that every circular cross section in your considered feature contains at least 90 degrees of arc.

When using a sample plane, it must be nominally orthogonal to the considered feature.		Ensure that the feature's sample plane is nominally perpendicular to the considered feature's axis. For information about how the sample plane is defined, see "Deriving the Toleranced Feature".
Feature <feature name> and its child features are out of synchronization. Please re-execute to re-synchronize them.	If you change a feature's measurement strategy, or if you change the number of rows in the measurement strategy, the feature can get out of synchronization with its child features, until the feature has been executed.	Execute the feature.
Segment <segment number> contains an invalid datum reference frame for the lower segment of a composite tolerance.	As discussed in "Position", "Profile of a Line", and "Profile of a Surface", lower segments of composite tolerances have strict rules to govern their datum reference frame.	Ensure that all lower segments' datum reference frames comply with the rules.

Datums referenced at MMB/LMB must have been previously toleranced at MMC/LMC to their higher precedence datums. Where no such tolerance is found, zero at MMC/LMC will be applied.	This is a warning to alert you that you should have already created the applicable Geometric Tolerance command(s) for any datum feature(s) with a material condition modifier. If you have not created the applicable Geometric Tolerances for the referenced datum with a material condition modifier, PC-DMIS uses a Geometric Tolerance of 0.0 MMC to determine the MMB/LMB (Maximum/Least Material Boundary) value.	You can continue to use geometric tolerances normally with this warning message. You should, however, check the drawing and ensure your routine already includes all Geometric Tolerances relating to the datum features. See the "Datum Reference Messages" section of the "How PC-DMIS Solves and Uses Datums" topic for more details.
Geometric tolerance calculation failed.	This indicates some internal problem in the calculation. Please file a request to open a support ticket with Hexagon Technical Support (support.hexagonmi.com).	With your ticket request, provide the following information: • Steps to duplicate the error message • Your measurement routine (.prg file) • Your CAD model (.cad file) • Supply a copy of the drawing and call-out you're trying to verify • Any probe files used (.prb files) • The debug file

The projected tolerance zone must have a positive projected distance.	The number that follows the P modifier defines the distance to project the zone beyond the feature, such as when evaluating a cylinder. The number must be a positive value.	Make sure that the projection zone's number is a positive value, not negative.

Scanning Your Part

Scanning Your Part: Introduction

PC-DMIS allows you to define a point measurement by scanning the surface of the part at specified increments. This provides you with a way to scan and digitize your part's surfaces.

PC-DMIS supports scanning in these products:

- PC-DMIS CMM - Using a touch trigger or analog probe on a CMM
- PC-DMIS Laser - Using a laser probe
- PC-DMIS Portable - Using a hard probe on a portable arm

For information on the above approaches to scanning, consult the appropriate documentation. The documentation for each product discusses the available scans and procedures to follow within those environments to create those scans.

Advanced Scans

Contact (PC-DMIS CMM)	Laser (PC-DMIS Laser)	Portable (PC-DMIS Portable)
Linear Open Advanced Scan	Linear Open Advanced Scan	
Linear Closed Advanced Scan		
Patch Advanced Scan	Patch Advanced Scan	
Perimeter Advanced Scan	Perimeter Advanced Scan	
Section Advanced Scan		
Rotary Advanced Scan		
Freeform Advanced Scan	Freeform Advanced Scan	
UV Advanced Scan		
Grid Advanced Scan		

Working with Section Cuts		

Basic Scans

Contact (PC-DMIS CMM)	Laser (PC-DMIS Laser)	Portable (PC-DMIS Portable)
Circle Basic Scan		
Cylinder Basic Scan		
Axis Basic Scan		
Center Basic Scan		
Line Basic Scan		

Manual Scans

Contact (PC-DMIS CMM)	Laser (PC-DMIS Laser)	Portable (PC-DMIS Portable)
Fixed Distance Manual Scan	Manual Laser Scan	Fixed Distance Manual Scan
Fixed Time / Distance Manual Scan		Fixed Time / Distance Manual Scan
Fixed Time Manual Scan		Fixed Time Manual Scan
Body Axis Manual Scan		Body Axis Manual Scan
Multisection Manual Scan		Multisection Manual Scan
Freeform Manual Scan		Freeform Manual Scan

The main topics in this chapter discuss information common to scanning in any of the supported applications as well as common functions of the scanning dialog boxes (**Insert | Scan**). They do not cover the specifics on how to create a scan, since the methods for doing so depend on your specific application.

The main topics discussed here include:

- Components of a Scan
- Working with CAD Surfaces
- Common Functions of the Scan Dialog Box

- Common Functions of the BASICSCAN Dialog Box
- Additional Manual Scan Options

Changing Animation Speeds: If you want to modify your offline animation speeds, see the **Execution** area on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). In addition, see "Executing and Debugging Measurement Routines Offline" in the "Working in Offline Mode" chapter.

Components of a Scan

Advanced scans in PC-DMIS are composed of "basic scans". For example, a Patch scan is actually made up of rows of data, where each row is a basic scan. The basic scans act as building blocks for the higher-level scans, such as Patch scans. The PC-DMIS CMM documentation discusses advanced scans and basic scans.

Working with CAD Surfaces

When you create scans, you often need to select one or more surfaces that you want PC-DMIS to scan. PC-DMIS shows selected surfaces in the defined highlight color:

Example showing a surface highlighted when selected.

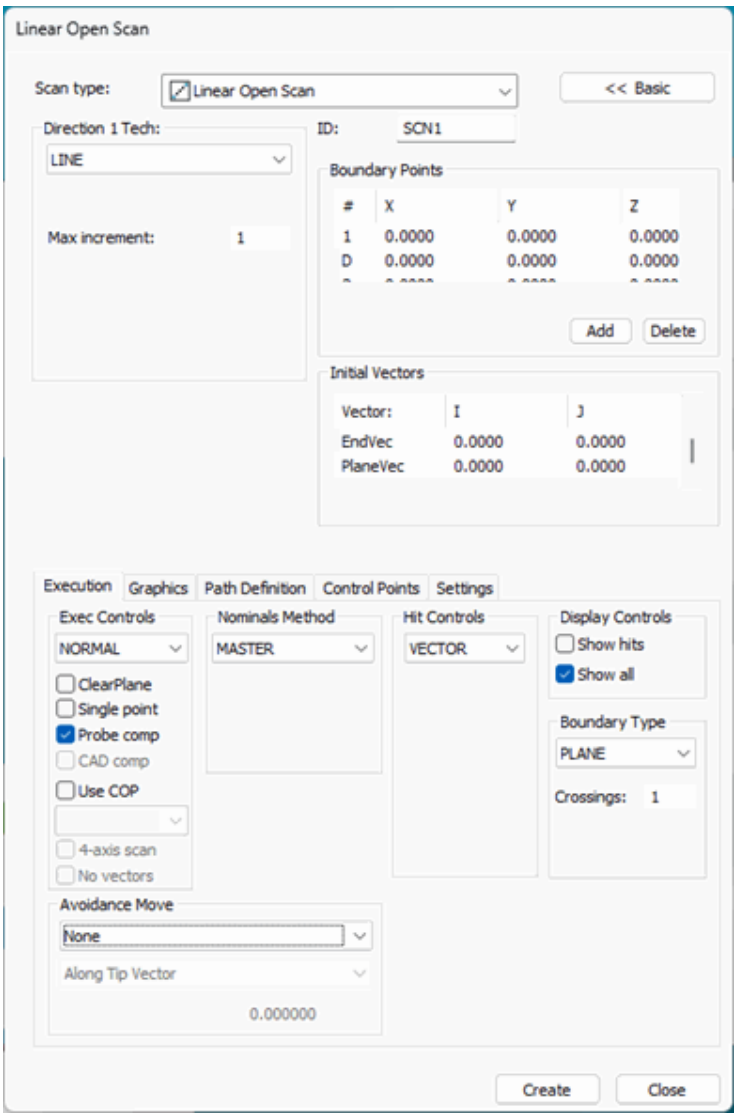
For more information on the highlight color, see the "Colors Tab" topic in the "Editing the CAD Display" chapter.

To deselect (or select) a desired surface:

1. Ensure that your part displays surface data.
    - Enable Surface Mode. To do this, from the **Graphic Modes** toolbar (**View | Toolbars | Graphic Modes**), select the **Surface Mode** icon ( ).
    - Show the part in solid view. To do this, from the **Graphic View** toolbar (**View | Toolbars | Graphic View**), select the **Show Graphic Window as Solid** icon ( ).
2. Access the appropriate scan dialog box (**Insert | Scan**), and start to define the scan.
3. As needed, click on a surface or surfaces to select them. If you make a mistake and need to deselect a surface, press Ctrl and click on the surface.

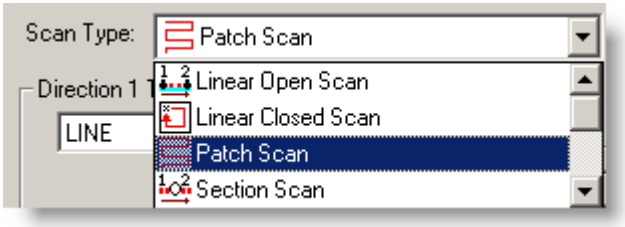
Common Functions of the Scan Dialog Box

Many of the functions described below are common to many of the scan dialog boxes (**Insert | Scan**) used in the supported applications. Options that relate specifically to one scan mode are appropriately indicated.



Scan Dialog Box

Scan Type

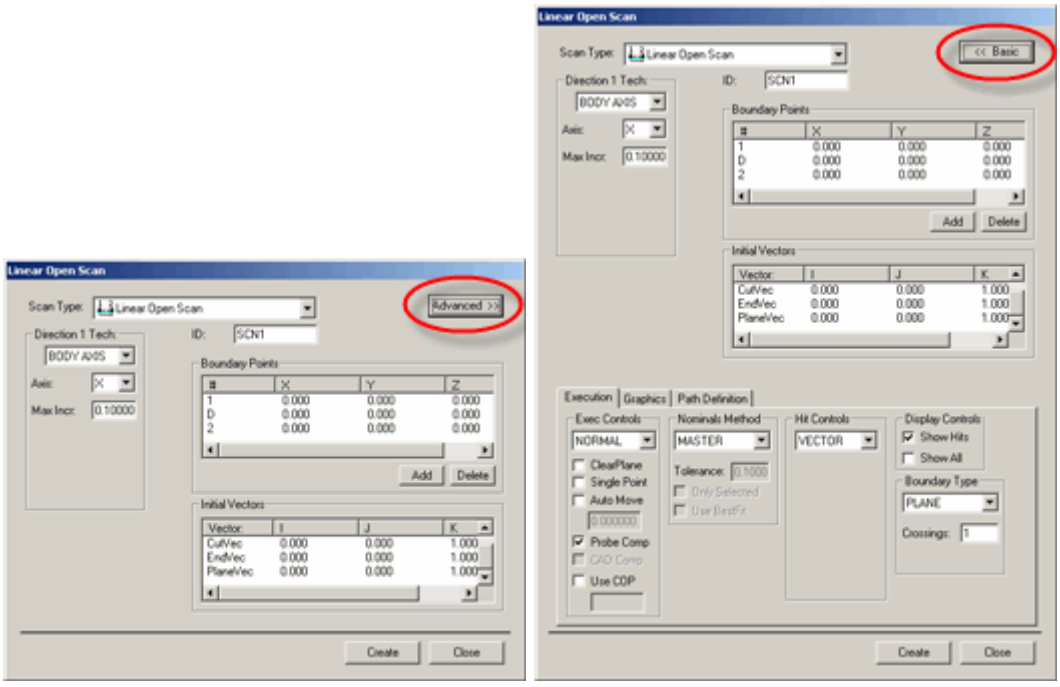


The **Scan Type** list in the scan dialog box (**Insert | Scan**) lets you switch between available scans. If you select a new scan, the dialog box changes to the selected scan type.

Basic and Advanced Buttons

The **<<Basic** and **Advanced>>** buttons in the scan dialog box (**Insert | Scan**) switch between displaying basic scanning options on the scan dialog box and more in-depth, advanced options.

- Click **Advanced>>** to expand the dialog box to include the **Execution**, **Graphics**, and **Path Definition** tabs in the lower section. Each tab contains additional options that you can use to define your scan.
- Click **<<Basic** to hide the more advanced items and display the basic information you need to create the scan.



Example showing basic options (left) and advanced options (right).

ID

The **ID** box in the scan dialog box (**Insert | Scan**) displays the ID of the scan to create.

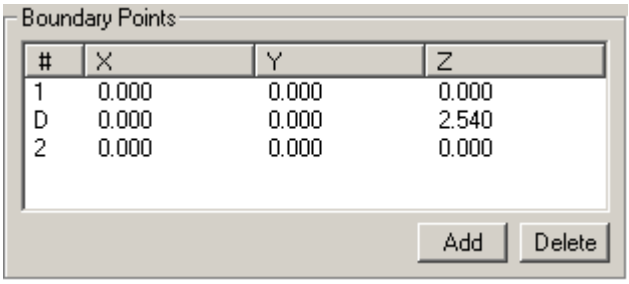
Measure

If you select the **Measure** check box in the scan dialog box (**Insert | Scan**) and click the **Create** button, PC-DMIS starts to measure the scan immediately. If you do not select the **Measure** check box when you click **Create**, PC-DMIS inserts a scan object into the Edit window that you can measure later. This enables you to set up a series of scans that you can insert into the Edit window and measure at a later time.

This check box is available *only* when PC-DMIS is Online.

Boundary Points Area

To define the boundary of a scan in the scan dialog box (**Insert | Scan**), PC-DMIS allows you to key in or measure points, or use CAD data.



Boundary Points Area

This function is only available for DCC scans.

LINEAROPEN - These scans can also be measured without an end point. Delete the end point, and PC-DMIS keeps measuring the scan until you manually stop it. You cannot delete the starting point and direction point.

LINEARCLOSE - These scans need to have the start and direction points. You cannot delete or add boundary points.

PATCH - These scans need to have at least three boundary points to create a triangular patch to work with. You can use the **Add** and **Delete** buttons in the **Boundary Points** area to add or delete extra points. It adds a **Closed Scan** check box to this area.

Closed Patch Scan Use the **Closed Patch Scan** check box to indicate that you would like to scan a closed feature like a cylinder, cone, slot, etc. If you select this check box, PC-DMIS reduces the number of boundary points needed to define the scan boundary. You would need to enter only the start and direction points and an end point. The end point indicates how far down and up the feature the scan needs to execute. The start and direction points together with the Initial Vector define the Cut Plane vector. Usually, the Cut Plane vector is parallel to the axis of the feature that you are measuring.

SECTION - Section scans use this area to set the boundary points for the SECTION scan, find holes defined in the CAD data, and toggle between displaying data for holes or for boundary points. Section scans add the **Cut CAD** and **Show Cut** buttons to this area for this purpose.

Cut CAD After you define a boundary and click **Cut CAD**, PC-DMIS automatically searches through the CAD data for any hole features along the scan's path. Any hole edges along the scans path are indicated with a integer for the point followed by a

"H" (i.e. 2H, 3H, etc.). The hole edge points are set at a default distance of 0.0787 inches from the theoretical hole edge.

You can cut CAD with specific, user-selected surfaces. To do this, select the boundary points, select the **Select** check box, select the desired surfaces, and then click **Cut CAD**. PC-DMIS then cuts only the selected surfaces to find the hole or holes.

You do not need to click Cut CAD if your CAD does not contain hole features. If you do not use this button, PC-DMIS scans the part using the given start and end boundary points.

When cutting surfaces, PC-DMIS uses only the surfaces that appear in the first View (the Blue View).

See "Setting up the Screen View" in the "Editing the CAD Display" chapter. When you have a complex CAD drawing with multiple surfaces, you can arrange groups of surfaces into CAD levels. (See the "Working with CAD Levels" topic in the "Editing the CAD Display" chapter.) Doing this helps you restrict Section CAD Cut operations to specific portions of the CAD model.

Use the **Show Cut** button to switch between displaying boundary or hole data. After you define a boundary and select the **Cut CAD** button, click **Show Cut** to switch to the appropriate display.

PERIMETER - These scans works the same as the LINEAROPEN scan.

ROTARY - These scans require you to have at least a start point and a direction point (shown as **1** and **D** respectively in the **Boundary Points** list and on the CAD display).

- If you don't have an end point (shown as **2**), PC-DMIS continues to measure the scan along the specified direction until it returns to the start point.
- If you have a start point and an end point, PC-DMIS scans along the specified direction until it reaches the end point.

  PC-DMIS defaults to providing you with a start, direction, and end point in the **Boundary Points** area. You can delete the end point, but you cannot delete the start or direction point.

  As you define each boundary point (by either clicking on the CAD or typing in the values), PC-DMIS automatically snaps each point to the radius distance from the center point unless you didn't define a radius. In this case, the first boundary point you define also defines the radius.

The **Boundary Type** area is only available in DCC Mode for Linear Open, Linear Close, Patch, Section, and Rotary scans.

Setting Boundary Points Using the Keyin Method

To use the keyin method to set the boundary of a scan:

1. Double-click the desired boundary point in the # column in the scan dialog box (**Insert | Scan**). This displays the **Edit Scan Item** dialog box.

Edit Scan Item Dialog Box

2. Manually edit the X, Y, or Z value.

3. Click the **OK** button to apply the changes.

The **Cancel** button disregards any changes that have been made and closes the dialog box.

The **Next** button accepts the changes and then brings up the next boundary point for you to edit.

Setting Boundary Points Using the Measured Point Method

To set the boundary of the scan using measured points, touch the probe on the part. This automatically updates the value of the boundary point that is currently selected in the **Boundary** list. The focus then moves to the next boundary point (if there are any in the list).

In the case of a PATCH scan, an extra boundary point is added automatically if the current point is the last point in the list. The PATCH scan displays the last point (this is the same as the previous point). PC-DMIS deletes this last point when you select the **OK** button in the scan dialog box (**Insert | Scan**).

Setting Boundary Points Using the CAD Data Method

PC-DMIS allows the selection of boundary points using both wireframe and surface data.

When you use CAD surface data:

1. Make sure that you have imported solid CAD data.

2. Make sure that the **Draw Surfaces** icon is selected from the **Graphic Modes** toolbar (**View | Toolbars | Graphic Modes**).

3. Access the scan dialog box that requires boundary points. Then, to select a boundary point, click on the desired location in the Graphic Display window.

The selected surface is highlighted. PC-DMIS automatically updates the value of the boundary point that is currently selected in the boundary list. The focus then moves to the next boundary point (if any are available). In the case of a PATCH scan, an extra boundary point is added automatically if the current point is the last point in the list.

When you use CAD wireframe data, there are two ways to select curve elements:

Mode 1 - Depth Curve

PC-DMIS uses a Depth curve during FindNoms operations to form a plane using two curves. Ideally, the Depth curve is normal to the other curves selected so that PC-DMIS can cross the two vectors (the vector of the Depth curve and the vector of any other selected curve) and form a plane from which it can find the nominals.

To indicate a Depth curve, select the **Depth** check box and then select a curve. Only one Depth curve is to be selected and this should be done after other curves have been selected.

1. Make sure that the **Curve Mode** icon is selected from the **Graphic Modes** toolbar.
2. Select the **Select** check box.
3. Select the **Depth** check box.
4. Select a curve.
5. Indicate the two CAD edges that are normal to each other.
6. Clear the check box.
7. Click on the part.

If a Depth curve is provided, PC-DMIS forms a plane by crossing the vector of each edge with the vector of the depth curve and piercing that plane to create a point.

Mode 2 - No Depth Curve

1. Select the **Select** check box.
2. Indicate the two CAD edges that are normal to each other.
3. Deselect the check box.
4. Click on the part.

If a depth curve is not selected, PC-DMIS simply drops the selected point onto the curve.

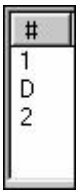
Only LINEAROPEN, LINEARCLOSE, and PATCH scans can use wire frame data.

Adding and Deleting Boundary Points

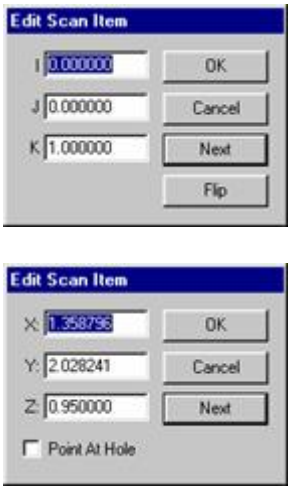
The **Add** and **Delete** buttons in the scan dialog box (**Insert | Scan**) enable you to add boundary points to the list of boundary points, or delete them. There are some restrictions about each type of scan. For example, a LINEARCLOSE scan will take only a Start and Direction point. It will not allow you to add more points or delete these two points. Please refer to each scan for specific restrictions.

Editing Boundary Points

You can edit boundary points by double-clicking the number of the desired point in the '#' column in the scan dialog box (**Insert | Scan**).



This displays the **Edit Scan Item** dialog box, which enables you to edit the X, Y, and Z values.



Example of the Edit Scan Item dialog boxes with the Flip button and the Point at Hole check box.

You can vary the column widths of the **Boundary Point** list by changing the width of the list display column header. To do this, select the right or left edge of a column header with the left mouse button and pull the edge to the desired size. You individually set and

determine the width of each list display. This information is saved in the INI file to be used each time you change the fields.

Flip:

The **Flip** button is only available when you edit a vector. Click this button to flip the selected vector.

Point At Hole:

The **Point At Hole** check box only becomes available when you work with Section scans. It allows you to change a non-hole point to a hole point.

A hole point defines where a linear section scan jumps over a hole that's found in its path. After you click the **Cut CAD** button, PC-DMIS places hole points on either side of any hole that interrupts the section scan.

Hole points are defined with the letter "H" after the point number (for example, 1H, 2H, 3H, etc.). These points, like any boundary point, are added to both the **Boundary Points** list and on the part model in the Graphic Display window.

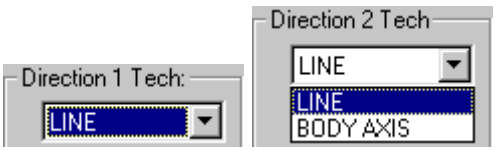
The **Point At Hole** check box is only available for non-hole points that you need to change to hole points. If you have a hole point that needs to be changed to a non-hole point, delete the hole point and create a new non-hole point.

Clearing Boundary Points

You can clear the **Boundary Points** list in the scan dialog box (**Insert | Scan**) of any of the scan types by right-clicking while the cursor is inside the **Boundary Points** list. A **Reset Boundary Points** button appears. Clicking this button resets all the boundary points to zero and the number of boundary points is set to the minimum for each scan type.

PC-DMIS doesn't allow you to clear the boundary points while you are using the **Cut CAD** button available on Section scans. In this case, you need to click the **Show Bnd** button to show the boundary points again before clearing the boundary points.

Direction Technique Areas



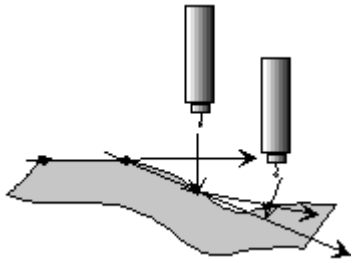
The **Direction 1 Tech** (left) and **Direction 2 Tech** (right) areas in the scan dialog box (**Insert | Scan**) determine how the scan takes hits. Most scans only scan in one row or line, so, they only have one set of direction techniques in the **Direction 1 Tech** list.

Patch scans are unique in that they scan over an area and so have additional rows of points and use a second set of direction techniques in the **Direction 2 Tech** list. In the **Direction 2 Tech** list, the selected technique determines the incremental technique that will be applied between rows.

Select the desired technique. PC-DMIS will automatically display the **Max / Min** or **Increment** boxes.

Line Technique

For Linear Open, Section and Patch scans - PC-DMIS determines each hit based on the set increment and the last two measured hits. The approach of the probe is perpendicular to the line between the last two measured hits. The probe will stay on the cut plane. PC-DMIS will start at the first boundary point and continue taking hits at the set increment, stopping when it reaches the end boundary point.



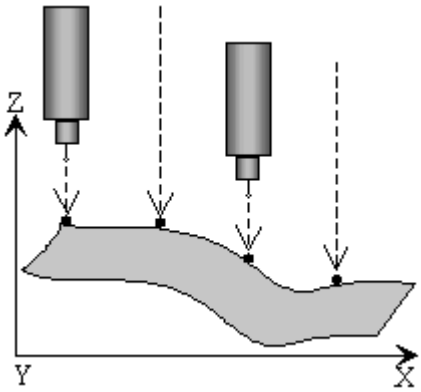
For Linear Closed scans - PC-DMIS determines each hit by the last two hits measured. The approach of the probe is perpendicular to the line between the last two measured hits. The probe will stay on the cut plane. PC-DMIS will not ask for the ending point when using this scanning technique. The scanning process terminates when the probe returns to the starting point.

For Rotary scans - PC-DMIS determines each hit based on the set increment and the last two measured hits. The approach of the probe is perpendicular to the line between the last two measured hits. The probe will always maintain the defined radial distance from the center point, perpendicular to the center point vector. PC-DMIS will start at the first boundary point and continue taking hits at the set increment, stopping when it reaches the end boundary point.

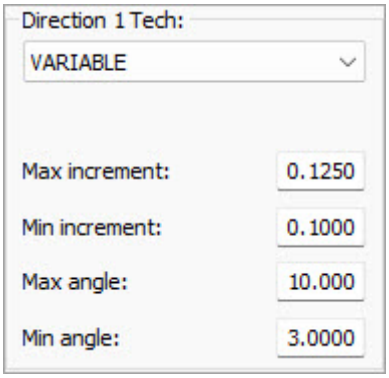
Body Axis Technique

The **BODY AXIS** technique in the scan dialog box (**Insert | Scan**) is only available for Patch and Linear Open scans when using a touch trigger probe.

PC-DMIS will take hits at the set increment along the current part's coordinate system. The approach of the probe is perpendicular to the indicated axis. The probe will stay on the cut plane. The approach vector will be normal to the selected axis and on the cut plane. The **BODY AXIS** technique uses the same approach for taking each hit (unlike the **LINE** technique which adjusts the approach to be perpendicular to the line between the previous two hits).



Variable Technique

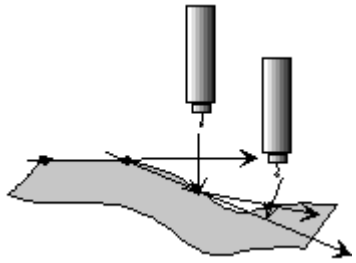


Available for Linear Open, Linear Closed, Patch, Section, and Rotary scans

The **VARIABLE** technique in the scan dialog box (**Insert | Scan**) enables you to set specific maximum and minimum angle and increment values that you can use to

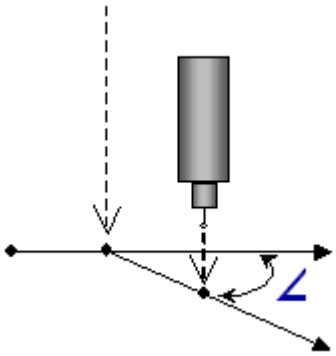
determine where PC-DMIS takes a hit. The probe's approach is perpendicular to the line between the last two measured hits.

To determine the increments between each hit, type the maximum and minimum values you want to use. You must also type the desired values for the **Max angle** and **Min angle** boxes. PC-DMIS takes three hits using the minimum increment. It then measures the angle between the first and second, and then the second and third hits.



- If the measured angle is between the **Max increment** and **Min increment** values, PC-DMIS continues to take hits at the current increment.
- If the angle is greater than the **Max angle** value, PC-DMIS erases the last hit and measures it again using one quarter of the current increment value.
- If the angle is less than the **Min angle** value, PC-DMIS takes the hit at the minimum increment value.

PC-DMIS again measures the angle between the newest hit and the two previous hits. It continues to erase the last hit and drop the increment value to one quarter of the increment until the measured angle is within the range defined, or the minimum value of the increment is reached.



- If the measured angle is less than the minimum angle, PC-DMIS doubles the increment for the next hit.
- If the measured angle is greater than the maximum increment value, PC-DMIS takes the hit at the maximum increment.

PC-DMIS again measures the angle between the newest hit and the two previous hits. It continues to double the increment value until the measured angle is within the range defined or the maximum increment is reached as shown below.

If ANGLE > MAX ANG then INC = INC / 4 until MIN INC

If ANGLE < MIN ANG then INC = INC * 2 until MAX INC

By default, Patch scans always start each new scan line with the minimum increment. If you would prefer that each new line starts with the increment from the previously scanned line, you can select the **Patch Scans Maintain Last Increment** check box on the **General** tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**). For information, see "Patch Scans Maintain Last Increment" in the "Setting Your Preferences" chapter.

Nullfilter Technique

Available to Linear Open, Linear Close, Patch, Section, and Rotary scans**.**

The **NULLFILTER** technique in the scan dialog box (**Insert | Scan**) does not filter data at all. Whatever data PC-DMIS receives from the machine controller is the data given you. Although Probe Compensation and FindNoms are still applied there is no data reduction. This technique lets you control the increment of the hits by using the `OPTIONPROBE` command which sets the point increment during a scan. See the "Parameter Settings: Optional Probe tab" topic of the "Setting Your Preferences" chapter for more information.

PC-DMIS will start at the first boundary point and continue taking hits at the set increment, stopping when it reaches the end boundary point.

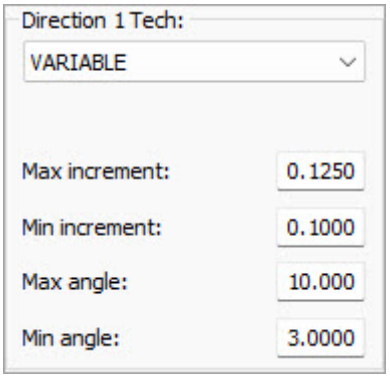
The **NULLFILTER** technique only appears in the **Direction 1 Tech.** list if you're using an analog probe head, such as the SP600.

For Rotary scans, the probe always maintains the defined radial distance from the center point, perpendicular to the center point vector.

Max and Min Boxes

These boxes are available for Linear Open, Linear Close, Patch, and Section scans if you select the **VARIABLE** scanning technique.

The **Max increment**, **Min increment**, **Max angle**, and **Min angle** boxes in the scan dialog box (**Insert | Scan**) are available when you use the **VARIABLE** scanning technique described in the "Variable Technique" topic. Only the **Max increment** box is available for *all* of the scanning techniques.



Example of the Direction 1 Tech area with options for the VARIABLE scanning technique.

Max increment

The **Max increment** box enables to set the maximum increment distance. Even though increments may increase while using the Variable option, the increments will never be greater than this distance.

Min increment

The **Min increment** box enables you to set the minimum increment. Even though increments may decrease using the Variable option, the increments will never be smaller than this distance.

Max angle

The **Max angle** box enables you to set the maximum angle. Even though angles measured may increase while using the Variable option, the angles will never be greater than this value.

Min angle

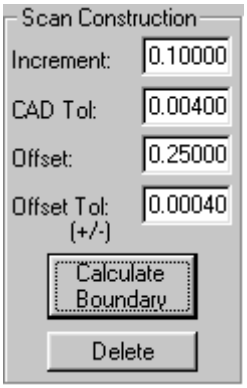
The **Min angle** box enables you to set the minimum angle. Even though angles measured may decrease while using the Variable option, the angles will never be less than this value.

Increment box

Available only for Patch scans.

Used with **Patch** scans, the **Increment** box in the scan dialog box (**Insert | Scan**) enables you to set the incremental distance between rows on the patch scan. For example, if you enter 0.5, then the scan sets the rows at increments of 0.5.

Scan Construction Area (for Perimeter Scan)



Used with the Perimeter Scan.

The **Scan Construction** area in the scan dialog box (**Insert | Scan**) has various options for constructing a Perimeter Scan. These include:

- Increment
- CAD Tol
- Offset
- Offset Tol (+/-)
- Calculate Boundary
- Delete

Increment Box for a Perimeter Scan

The **Increment** box in the scan dialog box (**Insert | Scan**) indicates the distance between each of the hit points on the scan.

CAD Tol

The **CAD Tol** box in the scan dialog box (**Insert | Scan**) is useful to detect neighboring surfaces. The larger the tolerance, the farther apart the CAD surfaces can be and still be recognized as a neighboring surface.

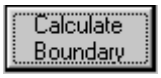
Offset

The **Offset** box in the scan dialog box (**Insert | Scan**) indicates the distance in from the perimeter where the scan will be created and executed.

Offset + / -

The **Offset Tol (+/-)** box in the scan dialog box (**Insert | Scan**) indicates the amount of allowable deviation from the offset value. It is a user supplied value.

Calculate Boundary

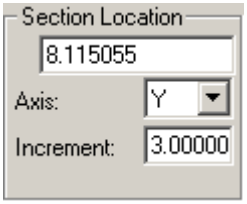


The **Calculate Boundary** button in the scan dialog box (**Insert | Scan**) determines the composite boundary of the input surfaces. The calculated boundary appears as red dots in the Graphic Display window.

Delete

The **Delete** button in the scan dialog box (**Insert | Scan**) deletes the previously created boundary.

Section Location area (for Section Scan)



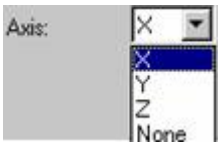
The box in the **Section Location** area in the scan dialog box (**Insert | Scan**) specifies the initial section at which you would want the scan to start.

If you want multiple sections at X = 5, X=5.5, X = 6, etc., you must first specify 5.0 as the initial section. After each scan, PC-DMIS will automatically jump to the next section at 5.5 and so on.

This value can be edited directly, or it can be set using the first boundary point. The first boundary point's coordinate value corresponding to the cut axis is the section location. When the first boundary point is defined by either taking a hit, selecting from CAD data, or keying in a value, the coordinate value for the cut axis is used.

If the first boundary point is set to 45, 37, 100 and the cut axis is Y, the section location is at 37. If the cut axis is X, the section location is 45.

Axis List



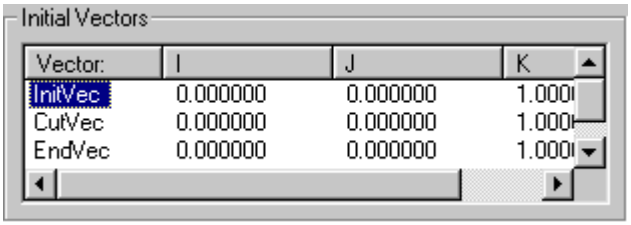
The **Axis** list in the scan dialog box (**Insert | Scan**) enables you to select the axis (**X**, **Y**, or **Z**) where you want multiple sections. The **None** option is also available. It allows you to select a "section line" on screen.

Scans do not usually allow you to work with curve data. However, if you select **None**, a section scan lets you select a graphically-displayed section line that is then used to define the cut plane and scanning path.

Increment

The **Increment** box in the scan dialog box (**Insert | Scan**) specifies the jump distance along the section axis after each scan is done.

Initial Vectors Area



The **Initial Vectors** area in the scan dialog box (**Insert | Scan**) displays a list of vectors that are used to start and stop a scan. Some scans do not use any initial vectors. These are UV, Grid, Perimeter, and FreeForm scans. The following lists the available initial vectors, when they are used, and their descriptions:

InitVec (Initial Touch Vector)

Used in Linear Open, Linear Closed, Section, Patch, Rotary, and Basic scan types.

The values that are displayed in the **Initial Touch Vector** row indicate the surface vector of the first point in the scanning process.

CutVec (Cut Plane Vector)

Used in Linear Open, Linear Closed, Patch, Section, and several Basic scan types.

A cut plane is used internally for DCC scanning calculations. This cut plane is derived differently for each supported scan type.

- *For Linear Open*, the Cut Plane vector (CutVec) is the cross product of the Initial Touch vector (InitVec) and the line between the start and end point. If there is no end point, the line between the start point and direction point is used.
- *For Linear Closed*, the Cut Plane vector (CutVec) is the cross product of the Initial Touch vector (InitVec) and the line between the start and end point.
- *For Patch Scan*, the cut plane vector (CutVec) is derived by crossing the Initial Touch vector (InitVect) and the line between the first and second point. The cut plane vector is then set to the correct direction by using the line between the second and third points. The End Touch vector (EndVec) is the vector used to take the second boundary points and is used to jump to the second row after completing the first row.
- *For Section Scan*, the Cut Plane and Initial Touch vectors are used to measure the scan. The Cut Plane vector is the cross product of the Initial Touch vector and the line between the start and end points. If there is no end point, then PC-DMIS uses the line between the start and direction points.

EndVec (End Touch Vector)

Used in Linear Open, Patch, Section, Rotary, and Line Basic scan types.

The End Touch vector is the approach vector of the scan at the end of the row. This is used only to stop the scan or to move to the next row (in the case of a Patch scan).

PlaneVec (Boundary Plane Vector)

Used in Linear Open, Linear Closed, Patch, Section, and Rotary scan types.

The Boundary Plane vector and the End Touch vector are used with the given Boundary Condition to stop the scan. The Boundary Plane Vector has different uses when applied to different Boundary Conditions:

- *For a Plane*, when used with a Plane Boundary Condition, it represents the normal vector of the Plane.
- *For a Sphere*, it is not used with the Sphere Boundary Condition.
- *For a Cylinder*, when used with a Cylinder Boundary Condition, it represents the Axis of the Cylinder.
- *For a Cone*, when used with a Cone Boundary Condition, it represents the Axis of the Cone.

DirVec (Initial Direction Vector)

Used in Rotary, Manual, and Line Basic scan types.

It represents the direction in which the scan will begin and is used with the Initial Touch vector to derive the Cut Plane vector.

SurfVec (Top Surface Vector)

Used in Linear Open and Linear Closed scan types.

This appears when you use the **Edge** hit type. It is the vector of the initial top surface of the edge and is used to start the scan.

vector1

Used in Linear Open and Linear Closed scan types.

This row is the surface normal vector of the first surface selected for angle hits. It appears when you use the **Angle** hit type. This corresponds to the **Surf 1 Vec** values in the **Angle Points** tab when creating Auto features. See "Creating an Auto Angle Point" in the "Measuring Features" chapter of the PC-DMIS CMM documentation.

vector2

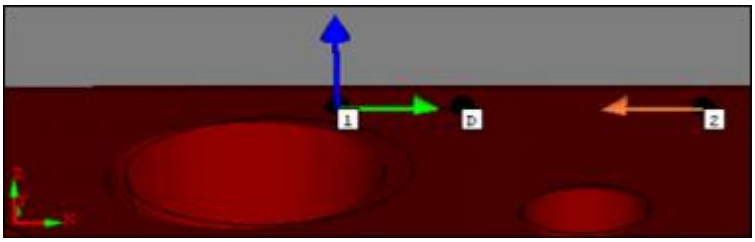
Used in Linear Open and Linear Closed scan types.

This row is the surface normal vector of the second surface selected for angle hits. It appears when you use the **Angle** hit type. This corresponds to the **Surf 2 Vec** values in the **Angle Points** tab when creating Auto features. See "Creating an Auto Angle Point" in the "Measuring Features" chapter of the PC-DMIS CMM documentation.

Graphical Representation of Scan Vectors

When setting up the start, direction, and end points of the scan, PC-DMIS allows you to see a graphical representation of the initial touch vector, the direction vector, and the vector that is normal to the boundary plane where the scan will stop.

These vectors are shown as blue, green, and orange colored arrows in the Graphic Display area of your part.



Example showing vectors with colored arrows.

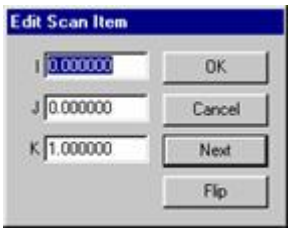
Vectors and their graphical representations are:

- Initial Touch: Blue arrow
- Direction: Green arrow
- Boundary Plane: Orange arrow

Editing Vectors

To edit each vector, double-click on the vector to edit in the vector column in the scan dialog box (**Insert | Scan**).

This displays the **Edit Scan Item** dialog box:



Edit Scan Item Dialog Box

Use the different fields to edit the **I**, **J**, and **K** values.

- Click the **OK** button in the **Edit Scan Item** dialog box to apply any changes that you made.

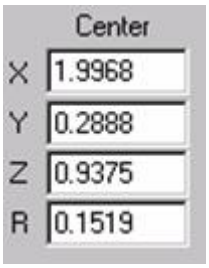
- Click the **Cancel** button to close the **Edit Scan Item** dialog box without applying any changes.
- Click the **Next** button to cycle through the available vectors in the **Initial Vectors** list. Some of the initial vectors can be flipped. If so, the **Flip** button becomes available in the **Edit Scan Item** dialog box.
- Click the **Flip** button to reverse the direction of the selected vector.

Select Center (for Rotary Scan)

Choosing the **Select Center** check box in the scan dialog box (**Insert | Scan**) enables you to click on the CAD to indicate the center point. You can select a surface point or wireframe point. PC-DMIS fills in the Center Point boxes with the XYZ information for the selected point.

When you select this check box, be aware that the boundary points of the scan will not be updated. PC-DMIS will only update the boundary points when you clear this check box.

Center Point and Radius (for Rotary Scan)

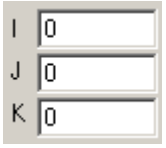


The X, Y, and Z values of Center in the scan dialog box (**Insert | Scan**) define the center point of the ROTARY scan.

You can directly type the center point **X**, **Y**, and **Z** values, or you can choose the **Select Center** check box and click on the CAD drawing to take the center point directly from the CAD model.

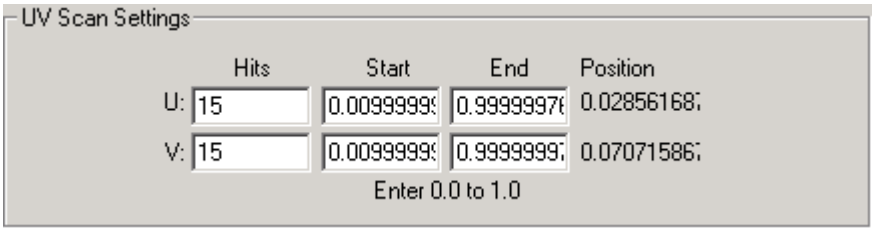
R defines the radius. When PC-DMIS executes the scan it will rotate around the center point maintaining this distance as the scan moves from the Start to the End point.

IJK (for Rotary Scan)



The **I**, **J**, and **K** values in the scan dialog box (**Insert | Scan**) make up a vector normal to the plane in which the **Radius** is maintained from the center point. PC-DMIS will follow this vector when performing the scan.

UV Scan Settings Area



UV Scan Settings Area

The **UV Scan Settings** area in the scan dialog box (**Insert | Scan**) enables you to define your UV scan. It contains the U and V rows and allows you to define the following controls.

- The **Hits** values allow you to specify how many hits the scan will take on the surface in the U or V direction.
- The **Start** and **End** values allow positioning of the hit matrix on the surface that is being scanned. These values can be set for both the **U** and **V** rows and apply to the scan along the U and V axes. Note that UV space uses numbers between 0.0 and 1.0 to represent the entire surface. So 0.0, 0.0 will be on the opposite diagonal corner from 1.0, 1.0.
- The non-editable **Position** fields indicate the probe 's current position along the U and V axes.

Grid Scan Settings Area

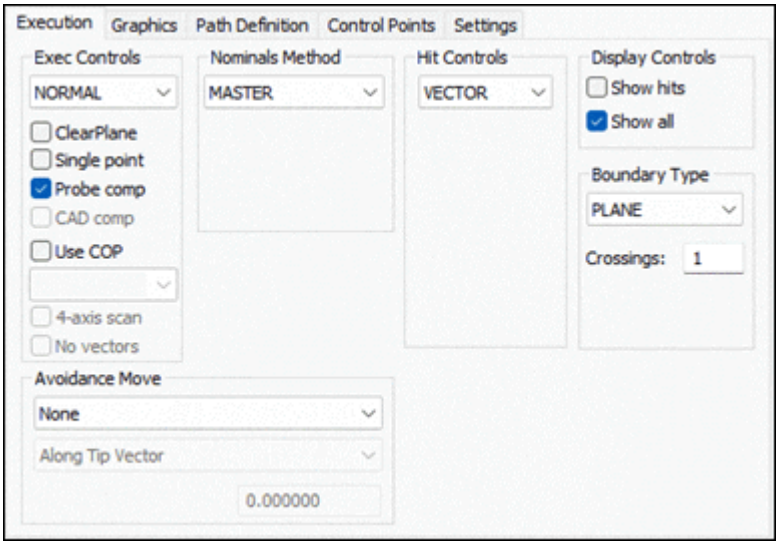
The **Grid Scan Settings** area in the scan dialog box (**Insert | Scan**) lets you define the number of hits to equally space in the A and B directions of a Grid scan. The A direction is horizontal, B is vertical.

If you typed 20 in the A direction and 20 in the B direction, PC-DMIS would attempt to space 20 rows and 20 columns of points on the combined selected surfaces within the rectangular area.

In the image below, only the top surface on the Hexagon block is selected. PC-DMIS only drops points onto that surface, not the others.

Example of a grid scan with 20 points in the A and B directions.

Execution Tab

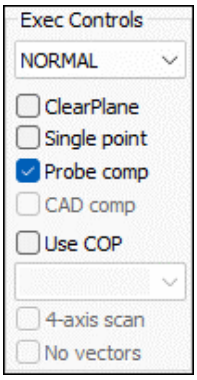


Scan Dialog Box - Execution Tab

The options on the **Execution** tab in the scan dialog box (**Insert | Scan**) let you determine what happens when you execute the scan being created. It contains these areas:

- Exec Controls Area
- Avoidance Move Area
- Nominals Method Area
- Hit Controls Area
- Display Controls Area
- Boundary Type Area

Exec Controls Area



The options in this area on the **Execution** tab in the scan dialog box (**Insert | Scan**) aren't used for all scan types. For example, manual scans only use a few of these options.

Manual scans use the following Exec Controls options:

- 3D check box
- CAD Comp check box (if FindNoms is selected)
- Use COP check box when there is a laser probe
  Execute list - This list lets you determine how PC-DMIS executes a scan after the scan has been learned.

  Normal - PC-DMIS executes the scan in its 'normal' manner; it triggers a hit when the probe touches the part.

  If a DCC scan is executed, PC-DMIS takes hits at each of the learned locations in stitch scanning mode, storing the newly measured data. The displayed nominals are the same as when the scan was learned and *cannot* be re-calculated using a different Nominals mode.

  Re-learn - PC-DMIS executes the scan as though it is learning it. All learned measured data replaces the new measured data. The nominal is re-calculated depending on the Nominals Mode (see "Nominals Mode"). Re-learn completely ignores the settings in the **Path Definition** tab and relearns the path as it goes.

  If a DCC scan is being re-learned, PC-DMIS relearns the scan from the beginning, instead of taking hits at the learned locations (as it would do in the case of the NORMAL mode).

  Defined - PC-DMIS allows the controller to ‘define’ a scan. PC-DMIS gathers all hit locations from the editor and passes them onto the controller for scanning. The controller will then adjust the path allowing the probe to pass through all the points. The data is then reduced according to the increment provided and the new data will replace any old measured data.

When this option is used after Generating the scan Offline, the nominal locations obtained from CAD are used every time to drive the CMM.

This mode is available only when you use analog probe heads that can do continuous contact scanning.

> The **Defined** mode with Perimeter scans does not support hole avoidance. Ensure that there aren't any holes in your scan's pathway with this execution mode; if there are, either adjust your Perimeter scan's path or switch to the **Normal** execution mode.

ClearPlane check box - The **ClearPlane** check box inserts a CLEARP MOVE a predetermined distance relative to the current coordinate system and part origin before taking the first hit.

After the last point in the scan is measured, the probe stays at probe depth until called to the next feature. Using clearance planes reduces programming time because the need to define intermediate moves is reduced. (See "Parameter Settings: Clearplane Tab" in the "Setting Your Preferences" chapter for additional information on clearance planes.) This option is only available for DCC scans.

Single Point check box - The **Single Point** check box considers each hit as a single measured point.

If this option is check marked, PC-DMIS will make each hit a measured point and insert it into the measurement routine. This sequence will happen after the manual scan has been reduced. If the scan is in DCC Mode, it takes place after the scan has been learned.

Probe Comp check box - The **Probe Comp** check box allows you to determine whether or not PC-DMIS turns on probe compensation for this specific scan.

In most cases you probably don't need to perform this compensation since a scan's FINDNOMS operation automatically does it. However, if you don't have CAD and want to reverse-engineer a part, you should select this check box.

CAD Comp check box - This check box determines whether or not PC-DMIS compensates for each point using the 3D surface vector from the CAD file. If not selected, PC-DMIS uses a 2D cut plane as usual.

This check box becomes available for selection if you select FINDNOMS from the list in the **Nominals Method** area or if you click on the CAD model in the Graphic Display window.

Inner Bound check box (not shown in image) - This check box allows you to determine whether PC-DMIS will perform an interior or an exterior perimeter scan.

- If selected, PC-DMIS will perform an interior perimeter scan.

- If deselected, PC-DMIS performs an exterior perimeter scan.

For a description of interior or exterior scans see the "Performing a Perimeter Advanced Scan" topic in the PC-DMIS Laser documentation.

Use COP check box - This check box determines whether or not the scanned points also get added to an existing Pointcloud (COP) command. If you select this check box, you can type the ID for the COP command into which you want to add the newly scanned points. If the COP command doesn't yet exist, PC-DMIS prompts if it should generate it for you.

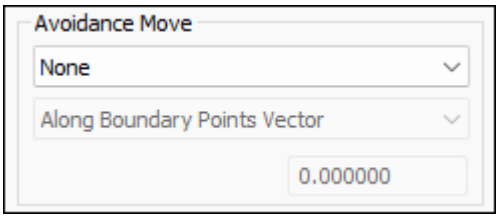
For information on COP commands, access the PC-DMIS Laser documentation where this command is discussed.

4-axis scan check box - Select this check box to allow scanning on a rotary table while the table is moving.

No vectors check box - This option is a special function of the Leitz B5 controller used for Technology Server 1.5 with PC-DMIS I++ DME machine interface. The purpose of this function is to find the highest point in the measured scanning trajectory.

For example, let us say you select this option and then run a rotary table scan around the edge of a blade. PC-DMIS will perform the scan without collision when only the points of the trajectory are sent to the controller, and not the surface point vectors of the defined scan.

Avoidance Move Area



Avoidance moves are special moves that you add to your scans to help PC-DMIS avoid driving the probe through the part during surface scans.

You can define the type of avoidance move, the approach path for the probe tip, and the distance to move before and/or after PC-DMIS creates the scan.

Type list - You can use this list to define the type of avoidance move:

None - If you select this option, PC-DMIS does not perform an avoidance move.

Before - If you select this option, PC-DMIS only performs the avoidance move before the scan.

After - If you select this option, PC-DMIS only performs the avoidance move after the scan.

Both - If you select this option, PC-DMIS performs the avoidance move before and after the scan.

Approach Path list - You can use this list to select the direction along which PC-DMIS performs the avoidance move:

Along Boundary Points Vector - If you select this option, PC-DMIS applies the avoidance move along the vector of the first scan path point and/or along the vector of the last scan point.

Along Tip Vector - If you select this option, PC-DMIS applies the avoidance move along the probe tip's vector.

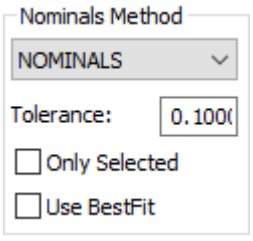
Along Cut Vector - If you select this option, PC-DMIS applies the avoidance move along the cut vector of the scan.

The **Along Cut Vector** option is available only if the selected scan has a defined cut vector.

Distance box - You can use this box to specify the distance that the probe moves during the avoidance move.

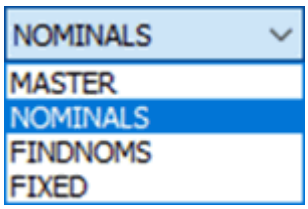
PC-DMIS applies the avoidance move to the entire scan, not to each basic scan defined within the scan.

Nominals Method Area



This area on the **Execution** tab of the **Scan** dialog box (**Insert | Scan**) contains these items:

- **Nominals** list

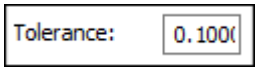


The **Nominals** list lets you determine how PC-DMIS gathers nominals for the measured data.

- **MASTER** - When you select this option, PC-DMIS treats the measured data from the first scan as nominal data. Subsequent scans are then compared to this data. When you select **MASTER** with a DCC scan and **Normal** from the **Execute** list, PC-DMIS executes a Stitch-type scan using the measured data.
- **FINDNOMS** - When you select this option, PC-DMIS pierces the CAD model to locate the closest location on a CAD surface to the measured point. It then sets the nominals to the location found on the CAD surface.

  With the Circle, Cylinder, and Axis types of Basic Scans, there is no need to have CAD data to find the nominals. To execute the Basic Scan, PC-DMIS obtains the nominals from the nominal data you supply. For more information, see "Nominals Mode Tab" in PC-DMIS Core documentation.

  If proper nominals are not found, PC-DMIS prompts you to provide a new Find Nominal tolerance.

  You can enter a new value in the **Tolerance** box and apply it to either the current scan or to the entire measurement routine.

  - If you select **Yes**, PC-DMIS uses the new tolerance to find nominals.
  - If you select **No,** PC-DMIS proceeds as follows:

  If proper nominals still cannot be found for any hit after you provide the new tolerance, PC-DMIS prompts you to confirm whether the hits can be deleted.

  - If you select **Yes**, PC-DMIS deletes the hits.
  - If you select **No**, the hits remain in the scan.
- **NOMINALS** - When you use this option, PC-DMIS uses any measured data from the first scan as nominal data. This option allows you to relearn the scan without relearning the nominals. The main difference between **NOMINALS** and **MASTER** is that the **NOMINALS** option constructs a nominal curve from the

nominal data. PC-DMIS then compares subsequent scan executions to this nominal curve, using the value specified in the **Find Noms Tol** box.

You can use the Nominals mode with the **RELEARN** option from the **Execute** list in the **Exec Controls** area. For more information, see "Exec Controls area" in the PC-DMIS Core documentation.

If you toggle to `NOMINALS` in the Edit window from a different mode, PC-DMIS automatically creates the nominal curve from the current theoretical data. This may take some time, depending on the amount of data used. Also, if the Edit window is set to `NOMINALS` and you toggle to a different mode, PC-DMIS deletes the nominal curve.

For descriptions of **LEAST_SQR**, **VECTOR_LST_SQR**, **MIN_MAX**, and **VECTOR_MIN_MAX**, see "The Best Fit Alignment Methods" in the "Creating and Using Alignments" chapter.

- **FIXED** - When you select this option, PC-DMIS keeps nominals the same in every execution of the measurement routine.

  The FIXED nominal method is available only when you select **DEFINED** from the **Execute** list in the **Exec Controls** area.

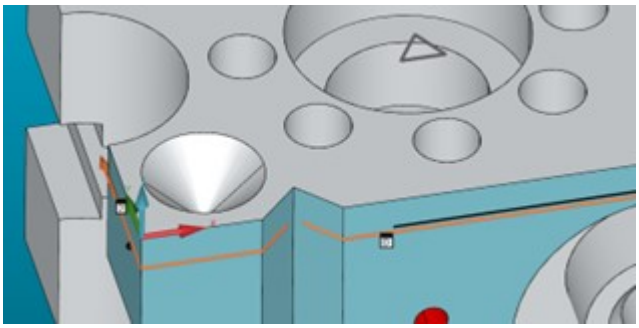
  Important Points:

  - You must use the **FIXED** nominals method only for 2D curves. PC-DMIS disables the **CAD Comp** check box in the **Exec Controls area** for this method.
  - This method only supports the Vector type of Hit Controls.
  - When you use this method, PC-DMIS selects and greys out the **Probe comp** check box in the **Exec Controls** area of the **Execution** tab.
  - The **Direction 1 Tech** list only offers the **NULLFILTER** technique. In this technique:
    - The **Point density** setting under the **Settings** tab controls the number of points that PC-DMIS generates per mm.
    - It also controls the points sent back by the controller.
  - You can adjust the point density to control the spacing between points in the **Path Definition** tab. To do this:
    - Define the point density in the **Settings** tab.
    - Generate the path in the **Path Definition** tab.

- It is important to have dense points (a high number of points) returned by the controller for this method to ensure accurate measurements. During execution, if the point density is set to less than 10 points/mm, PC-DMIS requests the controller to send back points at a minimum density of 10 points/mm. If the point density is set higher than 10 points/mm, PC-DMIS sends the point density request to the controller based on your specified value.

- This method removes any unreachable points from the scan path. Unreachable points are locations where the probe tip cannot physically reach due to the shape of the part.

  For example, as shown in this image, PC-DMIS removes points from the scan path where the probe tip cannot reach the surface due to the part's V shape:

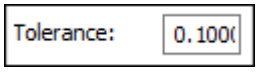


- This method considers the current probe tip radius. If the probe tip is replaced with one that has a different radius, regenerate the path to ensure that PC-DMIS removes the correct number of points from internal sharp corners.

- This method also removes nominal points at external corners of parts. At these points, the controller changes force from one direction to another. PC-DMIS controls the radius within which the points are removed by the **RemoveExternalCornerRadiusForFixed** settings entry, with a default value of 0.2 mm. You can increase this value or set it to 0 to remove points at external corners.

- You must generate the path only after you select the **FIXED** option from the **Nominals** list. This ensures that PC-DMIS removes points near corners and unreachable areas of the scan path while it generates the path.

- **Tolerance** box



The **Tolerance** box allows you to set a new nominal tolerance if proper nominals are not found when you use the **FINDNOMS** option from the **Nominals** list.

Ensure that the tolerance value is sufficient to compensate for the probe's radius. If the value is too low, PC-DMIS may display an error message stating that it cannot find the part when it attempts to generate the scan.

- **Use BestFit** check box

When you select the **Use BestFit** check box, PC-DMIS performs a temporary BestFit alignment on the scan to find better nominals for the measured data. For more information, see the **FINDNOMS** option in the **Nominals** list.

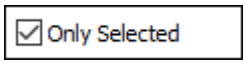
PC-DMIS follows this sequence for your scan:

- Performs a FindNoms operation.
- Uses the found nominal points and the measured data of the scan to create an internal Best Fit alignment. If it is a Patch scan, the Best Fit is 3D.
- Performs a FindNoms operation.
- Uses the found nominal points and the measured data of the scan to create another internal Best Fit alignment. If it is a Patch scan, the Best Fit is 3D.
- Performs a FindNoms operation.
- Restores the original alignment.

This check box is useful for parts with large deviations from nominal data, as it helps PC-DMIS find nominal data that more accurately represents the part.

For more information on Best Fit alignments, see "Creating a Best Fit Alignment" in the "Creating and Using Alignments" chapter of the PC-DMIS Core documentation.

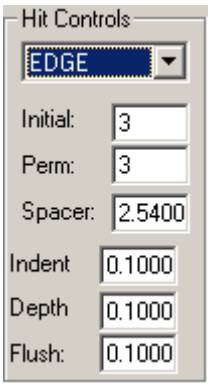
- **Only Selected** check box



When you execute a scan in your measurement routine and search for nominal values for the scan's measured points, you can use the **Only Selected** check box to have PC-DMIS only look for the nominals on the currently selected set of surfaces.

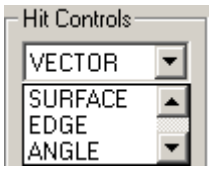
If any of your selected surfaces are already defined as priority surfaces in the **Edit CAD Elements** dialog box, PC-DMIS maintains their priority over other selected surfaces in the set. For more information, see "Editing CAD" in the "Editing the CAD Display" chapter of the PC-DMIS Core documentation.

Hit Controls Area



The **Hit Controls** area on the **Execution** tab in the scan dialog box (**Insert | Scan**) contains a list that controls where hits are taken. Other boxes in this area get hidden or displayed depending on the type of hit selected from the hit type list. This area contains these items:

Hit Type list



The **Hit Type** list contains these hit types:

- VECTOR - The scan uses vector hits.
- SURFACE - The scan is taken along a surface and uses surface hit data.
- EDGE - The scan is taken along an edge. When **Edge** hits are used and CAD data is available, PC-DMIS will allow you to enter a flush thickness for the nominals. This thickness is applied normal to the edge approach vector when finding the nominals for the scan. (This is contrary to the regular thickness that is applied along the surface normal.)
- ANGLE - This scan uses ANGLE hit data.

The scan will always do a stitch scan regardless of the probe head type.

Initial box

The **Initial** box allows you to set how many sample hits are taken before each actual point is measured. These sample hits are taken only the *first* time a scan is run.

Perm box

The **Perm** box allows you to set how many sample hits are permanently taken before each actual point is measured. These sample hits are taken *each* time the scan is run.

Spacer box

The **Spacer** box allows you to set the distance between sample hits.

Depth box

The **Depth** box allows you to set the depth from the edge where the sample hits will be taken. This option is only available if **EDGE** is selected from the **Hit Type** list.

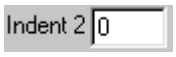
Indent box

The **Indent** box allows you to set the indent on the top surface from the edge where the sample hits will be taken. This option is only available if **EDGE** is selected from the **Hit Type** list.

Indent 1 box

The **Indent 1** box allows you to set the indent on the top surface from the edge where the sample hits will be taken. This option is only available if **ANGLE** is selected from the **Hit Type** list.

Indent 2 box

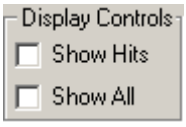


The **Indent 2** box allows you to set the indent on the top surface from the edge where the sample hits will be taken. This option is only available if **ANGLE** is selected from the **Hit Type** list.

Flush box

The **Flush** box allows you to enter a flush thickness for the nominals if an **Edge** or **Angle** hit type is used and CAD data is available. This thickness is applied normal to the edge approach vector when finding the nominals for the scan. (This differs from the regular thickness which is applied along the surface normal.)

Display Controls Area



The **Display Controls** area on the **Execution** tab in the scan dialog box (**Insert | Scan**) lets you determine what kind of information the scan displays in the Edit window. It contains these items:

Show Hits check box

If you select this check box, PC-DMIS displays each scan in the Edit window as a set of the measured hits surrounded by a scan object and an end measured object. If this option is chosen, all of the hits will be displayed in the Edit window. If you don't select it, the hits will not be shown.

Show All check box

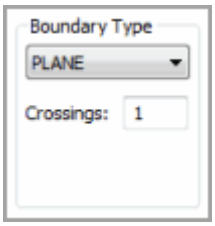
If you select this check box, PC-DMIS will display in the Edit window all of the scan parameters, such as:

- Cut planes
- Boundary points
- Direction vector
- Initial approach vector

If you don't select it, PC-DMIS only displays these data types in the Edit window:

- Increments
- Techniques
- Hit Types

Boundary Type Area

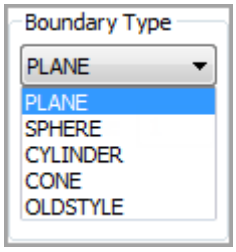


The **Boundary Type** area on the **Execution** tab in the scan dialog box (**Insert | Scan**) creates an imaginary feature that acts as a boundary around a scan's end point. By default, this feature is a plane at the end point that when crossed one time stops the scan. However, you can define the end boundary type to be a different type of feature.

Boundary types are available only for **Linear Open**, **Linear Close**, **Patch**, **Section**, and **Rotary** scans in DCC Mode.

The area contains these items:

Boundary Type list



The boundary type list lets you choose how a scan ends. Each boundary type uses a value from the **Crossings** box.

Plane - The scan stops after the probe crosses the plane at the end point the given number of times.

Sphere - The scan stops after the probe crosses (breaks) the sphere at the end point the given number of times.

Cylinder - The scan stops after the probe crosses (breaks) the cylinder at the end point the given number of times. The cylinder is unbounded (i.e. is considered infinite in length).

Cone - The scan stops after the probe crosses (breaks) the cone at the end point the given number of times. The cone is unbounded (i.e. is considered infinite in length).

OldStyle - (Retained for backwards compatibility)

Previous versions of PC-DMIS used a combination of boundary crossings and scan increments to stop the scan. OldStyle is not actually an option

that you choose, rather it is an internal setting for scans that were created in earlier versions of PC-DMIS.

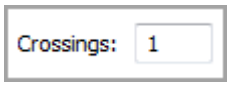
When scans from PC-DMIS version 2.3 are read into version 3.0 and higher, they are converted and their respective boundary conditions are tagged as OldType.

The Edit Window command line for the OldStyle boundary type reads:

```
BOUNDARY/OLDSTYLE, x,y,z,PlaneVec=i,j,k,
EndVec=i,j,k
```

- **PlaneVec**: This vector defines the plane normal vector at the end point.
- **EndVec**: The approach vector at the end point.

Crossings box



The **Crossings** box determines how many times a scan crosses the selected boundary type before stopping the scan. For example, if you specify that the number of crossings is two, the scan will stop once the probe's ball center crosses the given condition’s surface (planar, spherical, cylindrical, conical etc.) two times.

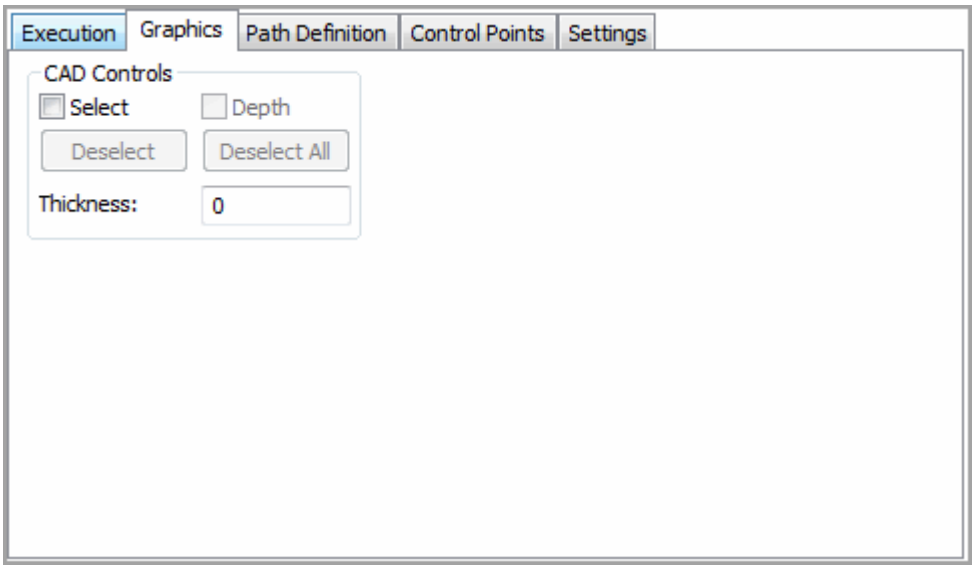
Linear Closed scans always require at least two boundary crossings even if the **Crossings** box specifies a lower number.

Radius box
The **Radius** box appears when you select either **Sphere** or **Cylinder** as your boundary type. It lets you define the radius of that boundary type feature.

Angle box
The **Angle** box appears when you select **Cone** as your boundary type. It lets you define the cone's total 'included' angle.

You can change the boundary condition at any time for a scan. If you choose a new condition for a DCC scan, PC-DMIS applies it to all basic scans making up the DCC scan. If, however, you chose to change a specific value in a condition, for example, maybe the radius of the sphere boundary type, PC-DMIS does not propagate that change to the basic scans. You would have to change that value in each basic scan yourself.

Graphics Tab



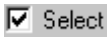
Scan Dialog Box - Graphics Tab

The **Graphics** tab in the scan dialog box (**Insert | Scan**) lets you use the on-screen CAD model to aid in creating a scan. This tab contains the **CAD Controls** area. This area lets you specify the CAD surface/wireframe elements that will be used to find nominals as well as the part's thickness.

In some cases, a scan might start over a certain surface and travel over many other surfaces before completion. In such cases, PC-DMIS does not know which CAD elements are to be used to find nominals. It must therefore search through every surface in the CAD model. If the CAD model has many surfaces, it might take a long time before the FINDNOMS operation is successful.

The **Graphics** tab has these options:

Select check box



Selecting this check box and clicking on surfaces lets you define the exact surfaces to scan. Manually determining the surfaces to scan helps speed up the FINDNOMS process. Once a CAD surface is selected, it is highlighted in the Graphic Display window. The Status bar displays the number of surfaces that have been selected. If the **Select** check box isn't selected, PC-DMIS assumes any clicks on the surface to be Boundary Points.

Two edges that are normal to each other must be selected for every surface the scan is going to travel over. If the scan is going over three surfaces, six edges (representing the three surfaces) must be selected in the proper order. The first two edges indicate surface one. The third and fourth edges indicates surface two. Edges' five and six make up the third surface, and so on.

Deselect button

If you select an incorrect surface, click on that surface a second time. This will deselect the surface. Clicking the **Deselect** button will deselect one surface at a time with each click of the button from a group of highlighted surfaces until all are deselected. Clicking the **Deselect All** button will deselect all highlighted surfaces at once.

The **Deselect** button removes one highlighted CAD element at a time from a group of CAD elements that are created using the **Select** check box.

Deselect All button

The **Deselect All** button removes *all* the selected CAD elements that are created using the **Select** check box.

Depth check box

This check box is used only when selecting Curve elements. You can indicate a particular CAD curve element as a Depth element.

To use the **Depth** check box:

1. Select all other CAD elements first.
2. Select the **Depth** check box.
3. Select a CAD element.

The Depth curve is used during FINDNOMS operations. Whenever PC-DMIS has to find nominals from curve elements, it will take the vector of the Depth CAD element, cross it with the vector from the other selected CAD elements to get a plane. It will then pierce the plane to get the proper nominal. If many CAD elements are selected, the closest pierce point is used as the nominal point. When CAD wireframe data is being used, PC-DMIS will look for the wireframe data in pairs.

vector1 check box

The **vector1** check box only appears if you select **Angle** from the **Hit Type** list and you are using surface data. It allows you to select CAD surfaces that PC-DMIS will use to find the nominal. You can indicate the group of surfaces that PC-DMIS uses to find the angle hit Surf 1 Vec by selecting this check box and then selecting CAD surfaces from the Graphic Display window.

vector2 check box

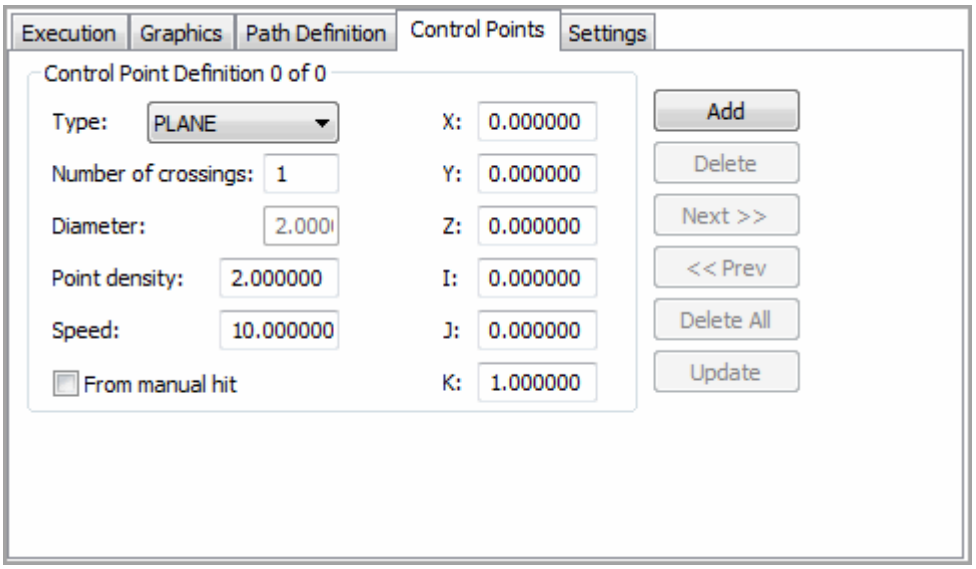
The **vector2** check box only appears if you select **Angle** from the **Hit Type** list and you are using surface data. It allows you to select CAD surfaces that PC-DMIS will use to find the nominal. You can indicate the group of surfaces that PC-DMIS uses to find the angle hit Surf 2 Vec by selecting this check box and then selecting CAD surfaces from the Graphic Display window.

Thickness box

The **Thickness** box allows you to input the part thickness. Positive or negative values can be used. This amount is primarily used for thin parts (plastic or sheet metal) where the CAD data only describes one side. Often with thin parts, the CAD engineer will only draw one side of the part, and then specify the material thickness. PC-DMIS will apply this material thickness automatically when using the CAD surface data.

This thickness is applied along the surface normal vector when FINDNOMS mode is selected and PC-DMIS pierces the CAD surfaces to get the nominals, even when using Edge hits.

Control Points Tab



Scan Dialog Box - Control Points Tab

> The **Control Points** tab only appears if you use an analog probe head that allows continuous contact scanning.

The **Control Points** tab in the scan dialog box (**Insert | Scan**) enables you to add control points to your scan. Control points interrupt the normal scan functions and allow you to define specific locations on the scan which alter the scan speed, point density, or both. Good continuous scans often require different scan speeds and/or point densities for different parts of the scan.

The ability to add control points has been added to these types of scans:

- Linear Open
- Linear Closed
- Patch
- Section
- Line (Basic Scan)

The **Control Points** tab contains the **Control Points Definition** area. This area lets you define each control point. It contains the following items.

Type list
This list defines the type of control point. The available types include:

1. Plane
   `PLANE, X, Y, Z, I, J, K, Num Crossings, Scan Speed, Point Density`
2. Sphere
   `SPHERE, X, Y, Z, I, J, K, Num Crossings, Scan Speed, Point Density, Diameter`
3. Cone
   `CONE, X, Y, Z, I, J, K, Num Crossings, Scan Speed, Point Density, Angle`
4. Cylinder
   `CYLINDER, X, Y, Z, I, J, K, Num Crossings, Scan Speed, Point Density, Diameter`

These types are similar to the boundary points.

The **Control Points** tab has these options:

Number of crossings box
This box defines the number of times the probe must cross the interrupt boundary before adjusting the specified parameters.

Diameter box
This box defines the diameter of the **Cylinder** or **Sphere** control point.

Angle box
This box defines the half angle of the **Cone** control point.

Point density box
This box defines the density of the points that are read into the scan data during a continuous contact scan. This is listed as the number of points per mm.

Speed box
This box defines the speed of the machine while it traverses the part surface performing scans.

From manual hit check box
If you select this check box, you can define the control point's XYZ and IJK location by manually taking a hit with the probe or by clicking on the part model.

XYZ boxes
These boxes define the XYZ location of the control point.

IJK boxes
These boxes define the IJK vector of the control point.

Add button
Adds a new, undefined control point.

Delete button
Deletes the current control point.

Next button
Goes to the next control point. The boxes in the area change to show the new point's data.

Prev button
Goes to the previous control point. The boxes in the area change to show the new point's data.

Delete All button
Deletes all the control points in the scan.

Update button
Updates all the control points in the scan with the setting for the current control point.

Adding and Using Control Points

1. Select the menu option **Insert | Scan** to open the **Scan** dialog box.
2. Define your scan. Based on your type of scan, PC-DMIS automatically defines the minimum number of boundary points for these scan types:
    - Plane
    - Sphere
    - Cone
    - Cylinder
3. Click the **Control Points** tab.
4. Add interrupt point locations by one of these three available methods:
    - **Method 1** - Key-in point locations. Click the **Add** button and then type the required values for each point.
    - **Method 2** - Select the **From manual hit** check box and then use the machine to touch the locations on the part where you want to have the control points. PC-DMIS adds the information to the **Control Points** dialog box.
    - **Method 3** - Select the **From manual hit** check box and then in the Graphic Display window, click on the CAD model where you want to have the control points. PC-DMIS adds the information to the **Control Points** dialog box.

    For the last two methods, PC-DMIS automatically snaps the chosen points to the cut plane of the scan for all scan types except Patch scans.

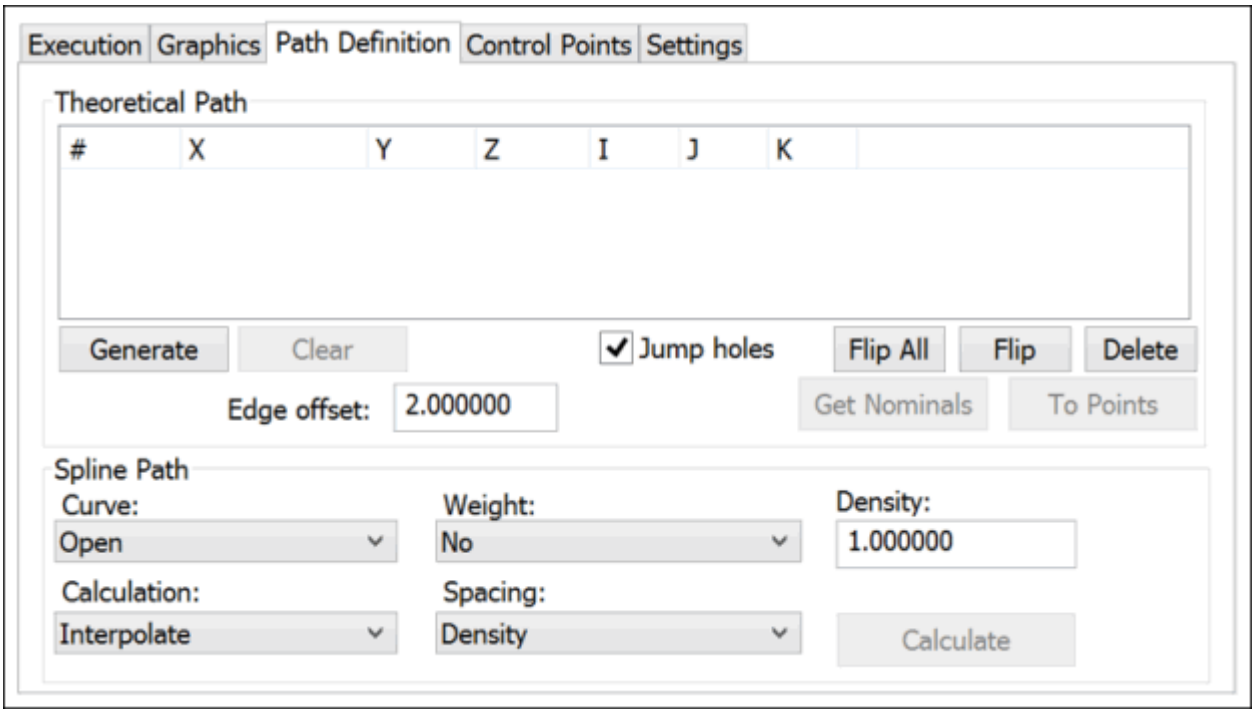
5. Continue to use the methods in step 4 to define all the desired control points.
6. Once you define all your control points, click the **Create** button. The **Scan** dialog box closes and creates your scan.
7. If you want to edit, delete, or add additional control points, click the **Control Points** tab at any time and make the modifications as needed.

Scanning after Adding Control Points

Once you've defined all the control points and created your scan, during execution, PC-DMIS does the following.

1. The scan will initially use the globally set **Scan Speed** and **Point Density** values.
2. As it scans along the path and passes within the parameters defined by the control points, the scan behavior changes as directed by the control points.
3. Once the scan is completed, the **Scan Speed** and **Point Density** will return to the globally set values.

Path Definition Tab



Path Definition Tab - Spline Path Area

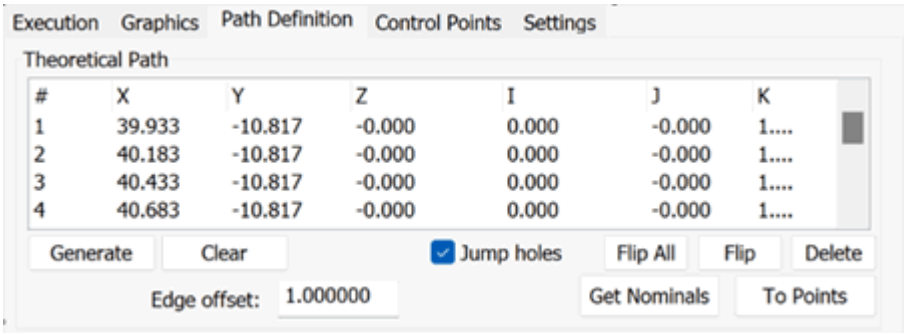
Only some scans support the **Spline Path** area. These include: Linear Open, Linear Closed, Patch, Section, and Freeform. On other scans, the **Spline Path** area does not appear.

The **Path Definition** tab in the scan dialog box (**Insert | Scan**) has, at most, two areas:

Theoretical Path and **Spline Path**

You can use these areas to generate a theoretical pathway for your scan and, on supported scans, you can fit the theoretical data points to a secondary path called a spline path. This essentially filters the number of theoretical points.

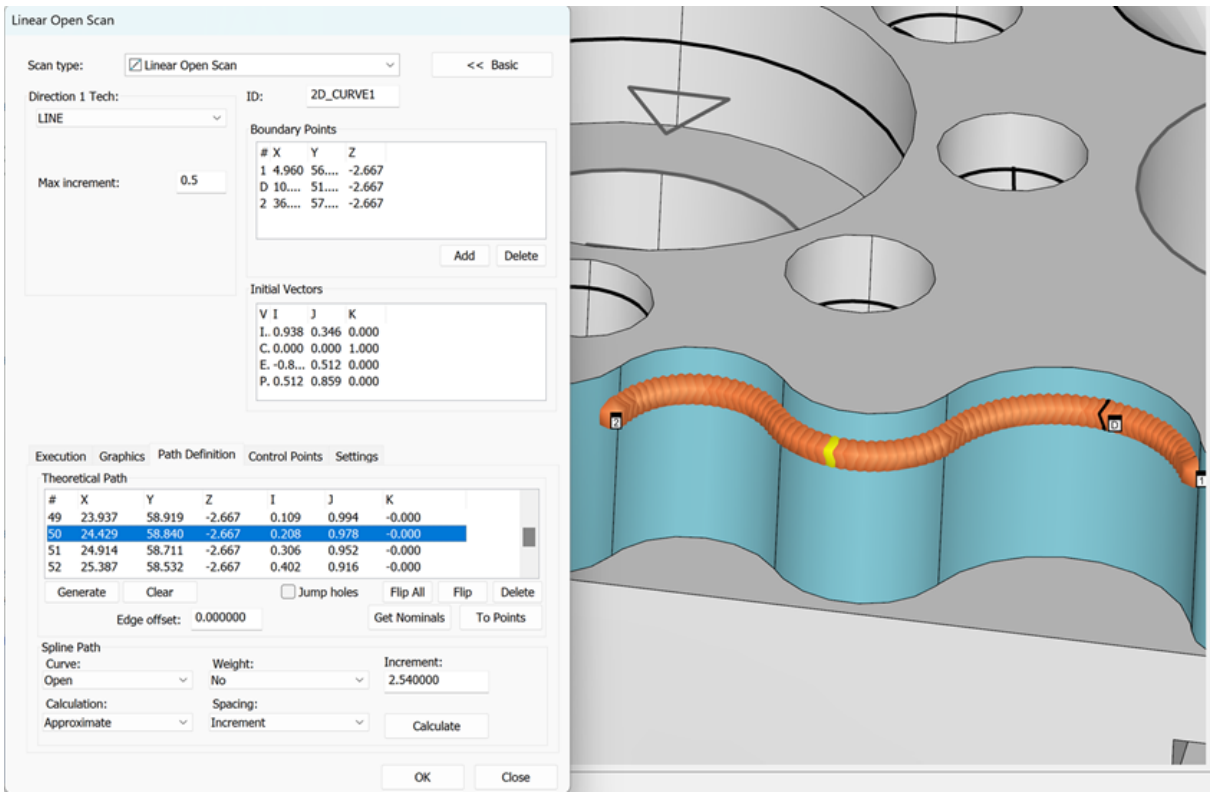
Theoretical Path Area



The **Theoretical Path** area on the **Path Definition** tab in the scan dialog box (**Insert | Scan**) contains some or all of these items depending on the type of scan selected:

- **Theoretical Path** list

  This list displays the theoretical pathway that your scan will take when executed. It displays the XYZ and IJK data for each point. To import points from a text file, click the **Read File** button (if using a Freeform scan). You can also click the **Generate** button to have PC-DMIS automatically generate the theoretical points. When you click on a point in the # column, PC-DMIS highlights the point in yellow in the Graphic Display window. In the case of a Freeform scan, you can click the points on the CAD model to create them.



Example showing the selected point in yellow.

To edit the value of any theoretical point:

1. Double-click the appropriate point number in the # column. The **Edit Theo Data** dialog box appears.
2. Change the values in the desired **X**, **Y**, **Z** and **I**, **J**, **K** boxes.
3. Click **OK**. The change appears in the **Theoretical Path** list.

To cycle through the point in the **Theoretical Path** list, click the **Next** button in the **Edit Theo Data** dialog box. This enables you to edit them one by one.

- To delete all of the points in the list, right-click on the list and select **Delete All Theoretical Points**.
- To delete a single point, select the point and then click **Delete**.

- **Generate** button

  First, select the required boundary point. The **Generate** button takes information from the CAD model and automatically generates the theoretical pathway within the scan's boundary for these supported scans: Linear Open, Linear Closed, Patch, and Section.

  PC-DMIS always uses the **Find Nominal Tolerance** value on the **Execution** tab when it generates nominal data for the scans and finds the nominals for the learned scan.

- **Clear** button

  Use this button to remove all of the points in the **Theoretical Path** list.

- **Read File** button

  This button appears when you use a Freeform scan. It lets you import points from a text file with a .txt filename extension. The text file should be in a comma-delimited format with one point on each line, like this: X,Y,Z,I,J,K

- **Jump holes** check box

  The **Jump holes** check box determines whether or not the theoretical path should jump over holes and other hole-like features in the surface.

  - If you select the check box, point data that does not lie within the selected surface (such as hole features) does not appear in the **Theoretical Path** list and will be skipped during scan execution.
  - If you clear the check box, the routine scans into hole-like features.

  When you work with Surface Mode and a tactile probe, the **Jump holes** check box is always selected when the scan spans through multiple surfaces. When you work with Curve Mode and a tactile probe, the **Jump holes** check box is not available for selection. The scan will always jump when multiple curves are selected.

  When you work with a spot laser probe in Surface Mode or Curve Mode, PC-DMIS records the last setting of the **Jump holes** check box and sets it accordingly.

  This appears on all scans but Perimeter and Freeform. It is not supported in these scans.

- **Edge offset** box

  The **Edge offset** option defines the minimum distance away from a boundary/edge where scan points are considered valid. Scan points that are closer to the boundary that the specified offset distance are not allowed. For example, if you set your offset distance to .5 mm, any scan point located within .5 mm of the boundary/edge is not allowed. This is used with the **Jump holes** option.

  PC-DMIS records the value that you enter for the **Edge offset** option and converts it into the appropriate unit of the next measurement routine.

- **Get Nominals** button

  This button lets you find the nominal after a scan has been learned or executed. PC-DMIS finds the nominals for the scan from the CAD data that is available.

- **To Points** button

This button runs a process where the individual points that make up an existing scan can be converted into single points after execution. PC-DMIS creates single points and places them in a group. The Group ID is the same as the scan ID.

For example:

1. Place your cursor on a scan using hit data in the Edit window.
2. Press the F9 key to display the **Scan** dialog box.
3. Select the **To Points** button.

PC-DMIS converts all hit data into single points and creates a group in the Edit window.

- **Flip** button

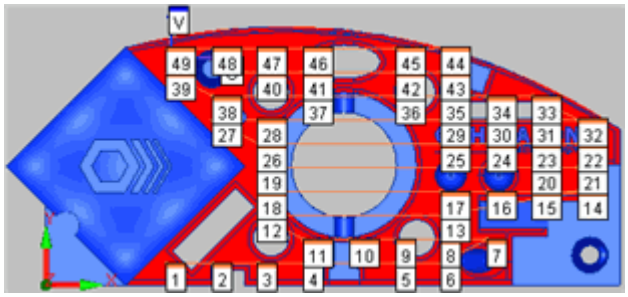
  This button flips the approach vectors of the theoretical scan point that is selected in the **Theoretical Path** list.

- **Flip All**

  This button reverses all of the vectors for all points in the **Theoretical Path** list.

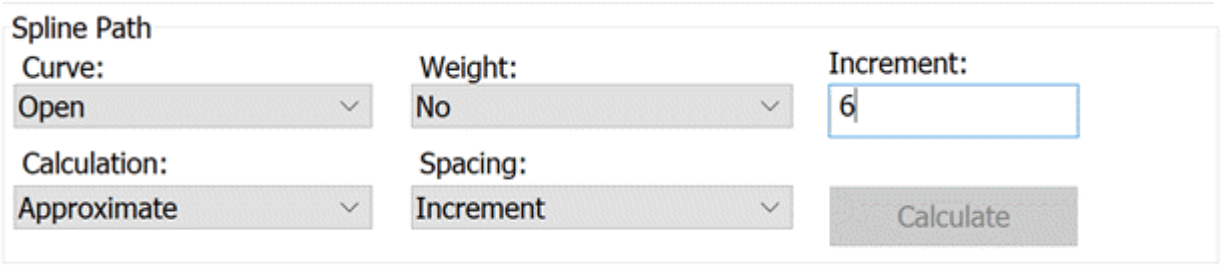
- **Label Hits** check box

  This check box determines whether or not PC-DMIS should label each hit in the scan. Labeled hits look something like this:



Example of a UV scan with labeled hits.

Spline Path Area



Spline Path Area

The **Spline Path** area on the **Path Definition** tab in the scan dialog box (**Insert | Scan**) contains several options that you can use to create a spline path to which your scan's theoretical path will be fitted.

This option is not available for Perimeter, Rotary, UV, and Grid scans.

To create a theoretical path and fit it to a spline path, do the following:

1. Select a scan type that supports a spline path (Linear Open, Linear Closed, Patch, Section, or Freeform).
2. Select the **Path Definition** tab.
3. Define the theoretical path. In the case of the Freeform scan, you can use the **Read File** button to import a pre-existing set of theoretical data points, or click **Generate**. The **Generate** button automatically generates a set of theoretical data points between your starting point and ending point from the existing CAD data.

   In the case of the Freeform scan, the **Generate** button does not exist. Instead, click on the CAD itself to select the theoretical points.

4. The points appear in the **Theoretical Path** list and show their XYZ and IJK data. As long as you have more than five points in the **Theoretical Path** list, you can fit it to a spline path by using the **Calculate** button in the **Spline Path** area. You don't have to fit the theoretical path to a spline path, but if the theoretical path list contains too many points, then you want to use the spline function to reduce or spread the points equally.
5. Define the parameters as described below for the spline path to which you will fit the theoretical path.
6. When the **Theoretical Path** and **Spline Path** areas are filled out, click **Calculate** in the **Spline Path** area. PC-DMIS changes the theoretical data to fit the parameters that you specified in the **Spline Path** area.
7. When you have the points as you like them, click **Create** to generate the scan.

The **Spline Path** area offers the following choices:

Curve list
This list contains two options: **Open** and **Closed**. An open curve passes through the start, control, and end points and then stops. A closed curve does the same, but when the curve passes through the end point, it also returns to the start point.

Calculation list
This list contains two options: **Approximate** and **Interpolated**. It determines whether the spline path passes through points in the spline (interpolated) or simply approaches the points (approximate). **Interpolated** results in sharper curves. **Approximate** results in smoother curves.

Weight list
This list contains two options: **Yes** and **No**. What you select determines whether or not PC-DMIS should give the points a weight when it constructs the spline path. If you select **Yes**, PC-DMIS gives them a weight according to how close they are to other points on the curve. The further they are from other points, the greater weight they have in the fitting process.

Spacing list
This list contains two options: **Increment** and **Number of Hits**.

- If you select **Increment**, PC-DMIS creates the spline path points at a given increment defined by the value in the **Increment** box.
- If you select **Number of Hits**, PC-DMIS creates the spline path points using and spacing the number of hits you specify in the **Number of Hits** box.

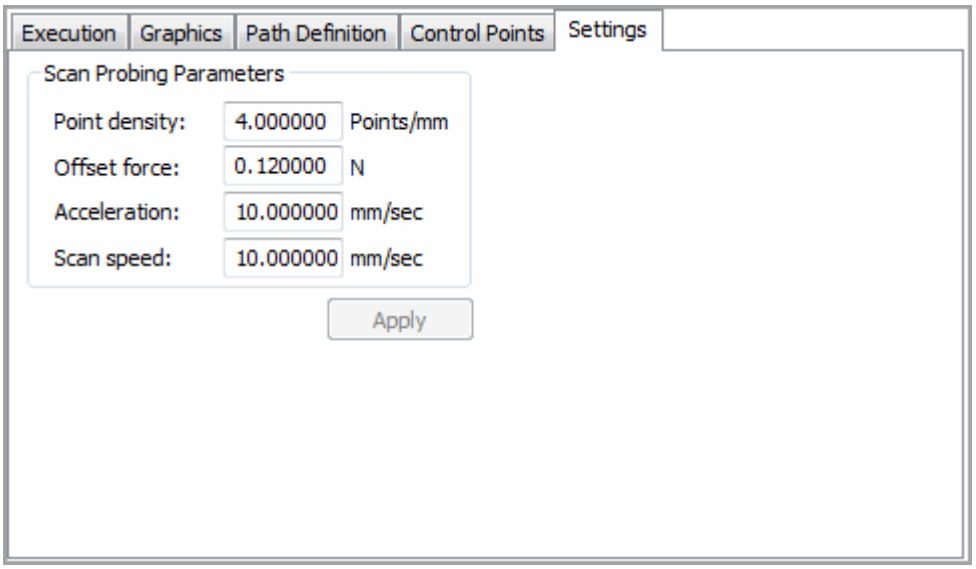
Increment or Number of Hits box
This box controls the number of hits along the scan's path.

- If you selected **Increment** in the **Spacing** list, this box holds the increment value. The distance between two path points is equal to this increment. The smaller the increment, the greater the number of path points.
- If you selected **Number of Hits** in the **Spacing** list, this box holds the number of hits to space along the theoretical pathway.

If you typed **50**, PC-DMIS would attempt to generate fifty points along the scan path.

Calculate button
This button calculates the spline and fits the theoretical data points to the spline, usually filtering out several hits. This button only becomes enabled if you have at least five hits in the **Theoretical Path** list.

Settings Tab



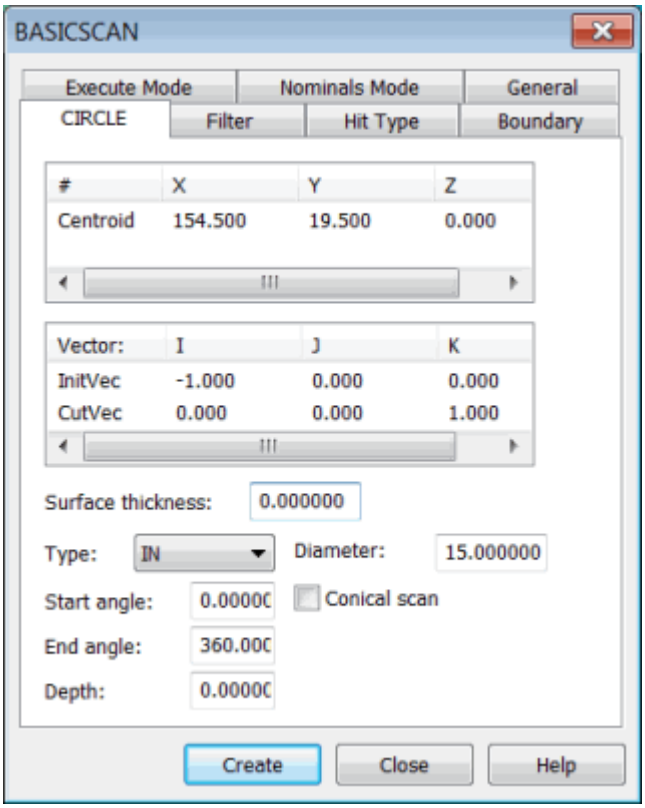
Scan Dialog Box - Settings Tab

The **Settings** tab only appears if you use an analog probe head that allows continuous contact scanning.

This tab in the scan dialog box (**Insert | Scan**) contains the **Scan Probing Parameters** area. Using the items in this area, you can modify these frequently used scan parameters, without having to leave the scan dialog.

These options are also included on the **Motion** tab or the **Opt. Probe** tab in the **Parameter Settings** dialog box (Edit | Preferences | Parameters). See the "Parameter Settings: Optional Probe tab" and "Parameter Settings: Motion tab" topics in the "Setting Your Preferences" topic for information.

Common Functions of the BASICSCAN Dialog Box



BASICSCAN Dialog Box

Many of the functions in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) are common to basic scans. Some of these options are the same as the options in the advanced scans and have already been discussed. (For additional information, see "Common Functions of the Scan Dialog Box" in this chapter.)

The tabs in the dialog box help you define the scan.

[Basic Scan] Tab

The [Basic Scan] tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) is the main tab for each basic scan feature. It is one of these tabs:

- **CIRCLE**
- **CYLINDER**
- **AXIS**
- **CENTER**
- **LINE**

The options on this tab include the following.

#, X, Y, and Z Columns

- Displays the points used to generate the scan. These include the following:

- **Start Point**: Used with axis, center, and line scans. This point is the start point from which the execution starts.
- **End Point**: Used with axis, center, and line scans. This point is the end point at which the execution ends.
- **D**: Used with a line scan. The Direction Point starts the scan and calculates the cut plane. The probe always remains within the cut plane while doing the scan.
- **Centroid**: Used with circle and cylinder scans. This point (found in the first list in the # column) is the center of the circle or cylinder. You can type the center directly or get it from the machine or CAD. For a cylinder, this is the center from which the execution starts.

X, **Y**, and **Z** - These columns display the coordinates for the item in the # column.

Vector, I, J, and K Columns

Vector - Displays the type of vector. These include the following:

- **InitVec**: Used with all of the basic scans. The **InitVec** is the surface normal vector of the point the scan defines as 0 degrees for the circle and cylinder scan types. It is the surface vector of the starting point for the other scan types.
- **CutVec**: Used with circle, cylinder, center, and line scans. The **CutVec** defines the plane in which the feature lies.
- **EndVec**: Used with the line scan. The end vector is the approach vector at the end point of the scan.
- **DirVec**: Used with the line scan. The direction vector is the vector from the start point to the direction point.

I, **J**, and **K** - These columns display the IJK vector information for the vector.

Surface Thickness Box

This is used with all basic scans. Use the **Surface thickness** box to enter the part thickness. PC-DMIS applies this material thickness automatically when it uses CAD surface data. This thickness is applied along the surface normal vector when **FINDNOMS** mode is selected. PC-DMIS pierces the CAD surfaces to get the nominals.

Control Points Button

This is used with the line scan. This button displays the **Control Points** dialog box. This dialog box contains controls that are similar to the **Control Points** tab in the scan dialog box. For more information, see "Control Points Tab".

This button appears only if you are using a continuous contact probe (analog probe), such as the SP600. You can select it once you define the Start, Direction, and End Points of the line scan.

Type List

The **Type** list defines the type of circle, cylinder, or center scan to perform. When you use it with circle and cylinder scans, this switches between:

- **IN**: Defines the scan as a hole.
- **OUT**: Defines the scan as a stud.
- **PLANE**: A plane circle is executed on the plane the circle is lying on.

When you use this option with center scans, it switches between the available centering methods:

- **Axis:** The Start Point **(S)** is projected on the defined Axis **(A)**. The resulting point is **(SP)**. The **InitVec** is projected in the plane defined by the Projected point **(SP)** and the axial direction **(A)**. The direction **(N)** thus defined is vertical to the axial direction. Thereafter, as centering is performed, the probe's center point remains in the plane defined by the axial direction and **(SP)**. Centering takes with/against the direction **(N)** as an input, and the probe tip is free in the direction defined by the axial direction **(A)** crossing the direction (**N**).
  - **S** = Start Point
  - **A** = Defined Axis / Axial direction
  - **SP** = Projected Start Point
  - **N** = Direction vertical to the axial direction
- **Plane:** After probing the point defined by the *Start Point*, the CMM centers with/against the probe direction while remaining free in the plane defined by the *CutVec*.

Diameter Box

This is the diameter value for Circle and Cylinder feature scans.

Conical Check Box

This check box enables you to perform circle scans more quickly when not perpendicular to the part surface. PC-DMIS continues to monitor the probe force as needed.

Start Angle Box

This box defines the start angle (in degrees to scan) from the start point for circle scans. You can use both positive and negative angles.

- PC-DMIS considers positive angles as counterclockwise.

- PC-DMIS considers negative angles as clockwise.
- PC-DMIS considers the **CutVec** as the axis about which the angle rotates.

End Angle Box

This value is the same as the **Start angle** box except that it defines the end angle for circle scans. With start and end angles, you can define a specific portion of a hole or stud to scan.

Angle Box

When you perform cylinder scans, this value defines how far around the cylinder to scan. For example, if you type 360, it scans one complete revolution. If you type 720, it scans two revolutions, and so forth.

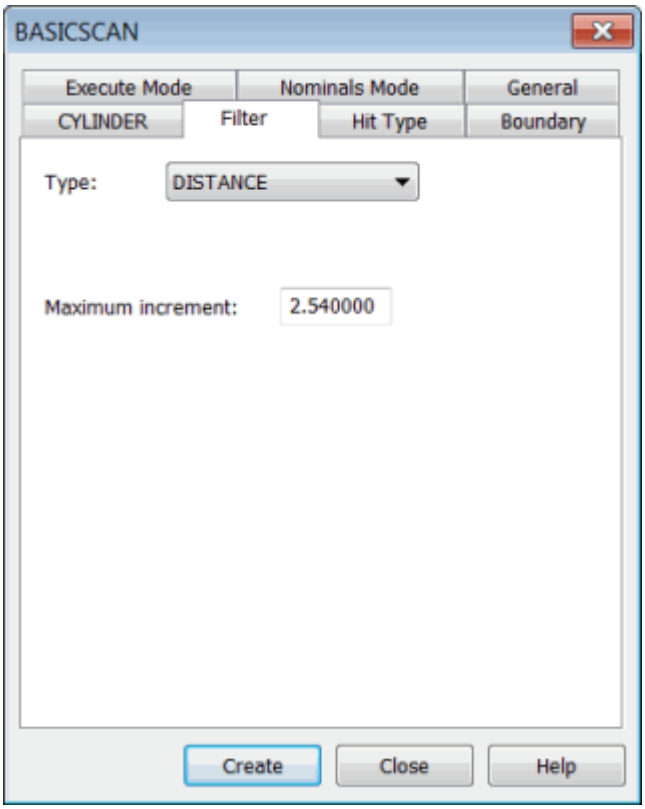
Depth Box

This value is the depth applied against the **CutVec** direction for circle and cylinder scans. You can use both positive and negative values.

Pitch Box

This box defines the distance between threads along the axis of the feature when you perform cylinder scans. This enables your scan to move in a spiral-like fashion along the cylinder.

Filter Tab

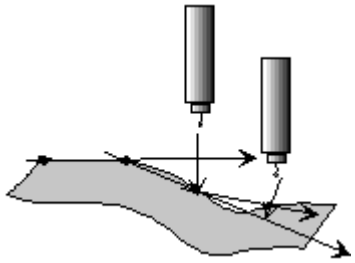


BASICSCAN Dialog Box - Filter Tab

The following options are available on the **Filter** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**). For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

Distance Type

For Line Basic and Axis Basic scans - PC-DMIS determines each hit based on the set increment and the last two measured hits. The approach of the probe is perpendicular to the line between the last two measured hits. The probe will stay on the cut plane. PC-DMIS will start at the first boundary point and continue taking hits at the set increment, and stop when it reaches the end boundary point.



For Circle Basic and Cylinder Basic scans - PC-DMIS determines each hit based on the set increment and the last two measured hits. The approach of the probe is perpendicular to the line between the last two measured hits. The probe will always maintain the defined radial distance from the center point, perpendicular to the center point vector. PC-DMIS will start at the first boundary point and continue taking hits at the set increment, and stop when it reaches the end boundary point.

Variable Type

For details, see "Variable Technique".

NULLFILTER Type

The **NULLFILTER** type is available for these scans: Line Basic, Circle Basic, Cylinder Basic, Axis Basic, and Center Basic.

The **NULLFILTER** technique does not filter data at all. Whatever data PC-DMIS receives from the machine controller is the data given you. Except for the center basic scan, probe compensation and the finding of nominals are still performed.

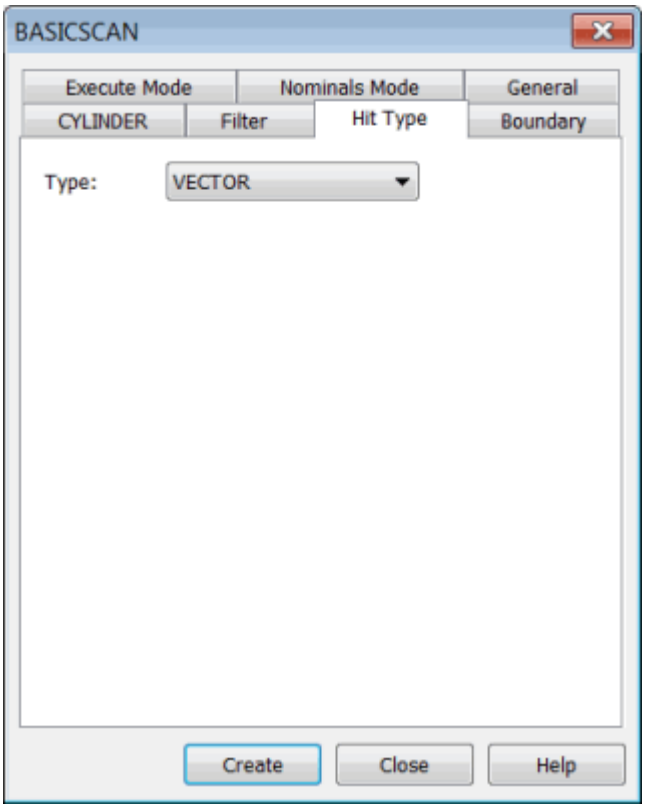
You control the increment of the hits by using the OPTIONPROBE command, which sets the point increment sent to the controller during a scan. For more information, see "Parameter Settings: Optional Probe tab" in the "Setting Your Preferences" chapter.

PC-DMIS will start at the first boundary point and not filter any data points, and stop when it reaches the end boundary point.

Maximum Increment

Use the **Maximum increment** box to set the incremental distance between hits on a scan. For example, if you enter .5, then the scan touches the part at increments of .5.

Hit Type Tab



BASICSCAN Dialog Box - Hit Type Tab

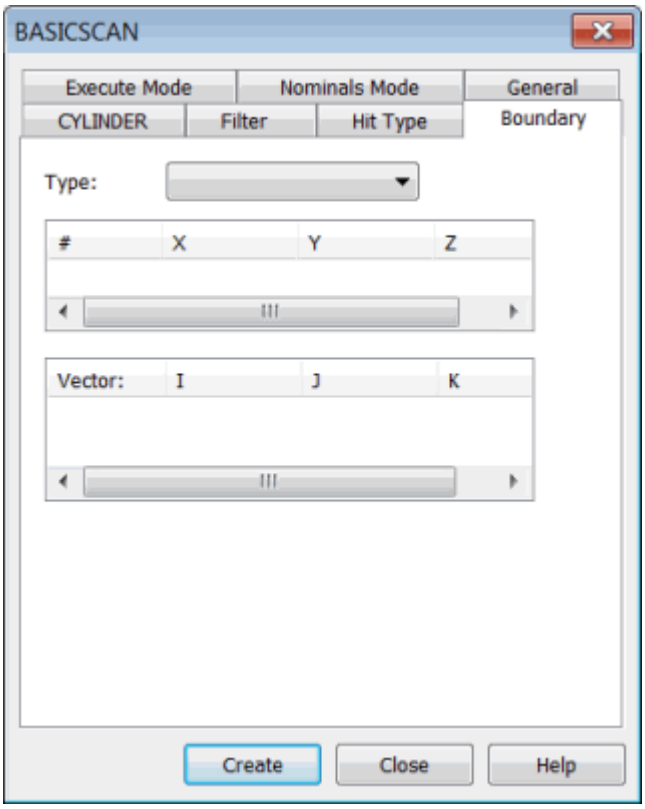
The options in the **Type** list on the **Hit Type** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) determine the type of hits used for the scan. The options are:

- **VECTOR**
- **SURFACE** (available only for line scans)

The type of hit corresponds to the type of hit in scans previously defined. For more information, see **Hit Type** in "Hit Controls Area" in "Common Functions of the Scan Dialog Box".

For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

Boundary Tab



BASICSCAN Dialog Box - Boundary Tab

The options in the **Type** list on the **Boundary** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) are:

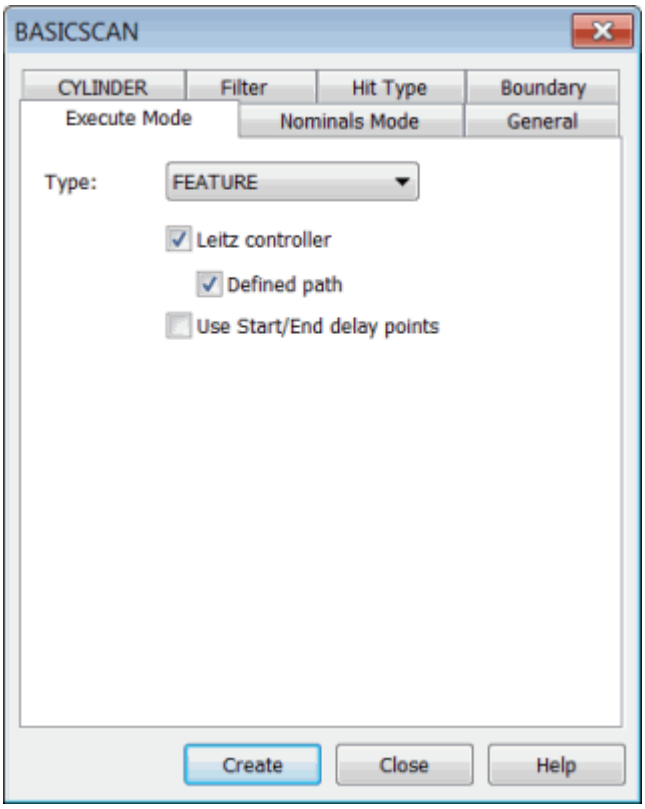
- Plane
- Sphere
- Cylinder
- Cone
- Oldstyle

Some basic scans, such as circle, cylinder, axis, and center, do not need a boundary condition since they are executed by the controller.

For descriptions, see "Boundary Type Area".

For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

Execute Mode Tab



BASICSCAN Dialog Box - Execute Mode Tab

The modes in the **Type** list on the **Execute Mode** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) are:

- Normal
- Re-learn
- Defined

> For more information on the Normal, Re-learn, and Defined modes, see "Exec Controls Area".

- **FEATURE** - This mode is available only for analog probe heads. If you select this mode, PC-DMIS uses the built-in high speed scanning capability of the controller to execute a scan.

> If you selected a circle scan, PC-DMIS uses a corresponding circle scanning command in the controller and pass on the parameters to the controller to execute. In this case, PC-DMIS does not control the execution of the scans.

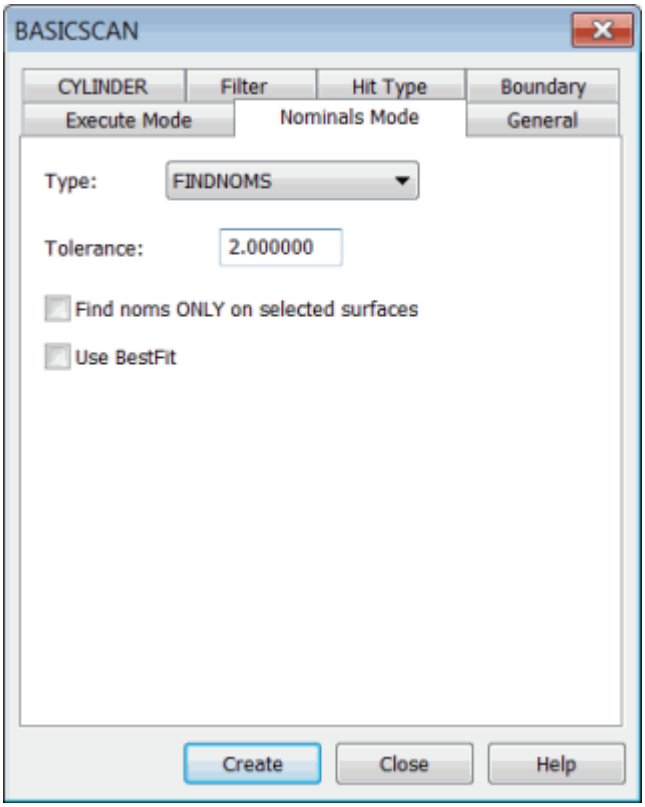
The **FEATURE** mode also enables you to use the following check boxes to further customize how PC-DMIS basic scans are done using this mode. Note that these check boxes only work with controllers that use Leitz interfaces.

- **Leitz controller** - Select this check box if running on a Leitz controller.
- **Defined path** - If you select this check box, PC-DMIS sends a distinct set of points defining the feature measurement path to the controller. If you clear this check box, PC-DMIS sends the feature-specific parameters to the controller.
- **Use Start/End delay points** - If you select this check box, PC-DMIS takes the points sent to the controller and discards those points taken during acceleration and deceleration of the probe during the scan, thereby filtering out fluctuation during these machine speed changes and improving the overall accuracy of the scan.

> ⚠ Be aware that when you use start/end delay points, the machine will over travel the defined start angle and end angle by the distance it takes to arrive at full speed when starting the scan. If you are not careful, this could cause a probe crash if there is an obstruction on the circle feature.

For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

Nominals Mode Tab



BASICSCAN Dialog Box - Nominals Mode Tab

The options in the **Type** list on the **Nominals Mode** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**) are:

- **MASTER** - Corresponds to Master mode, which is described in the "Nominals Mode" section.
- **FINDNOMS** - Corresponds to FindNoms mode, which is described in the "Nominals Mode" section. You can enter a FindNoms tolerance that can be applied when finding nominals for a basic scan. For more information, see **Nominals** list in "Nominals Method Area".

Find noms ONLY on selected surfaces - This check box causes PC-DMIS to look for nominal values for a scan's measured points on only the selected set of surfaces.

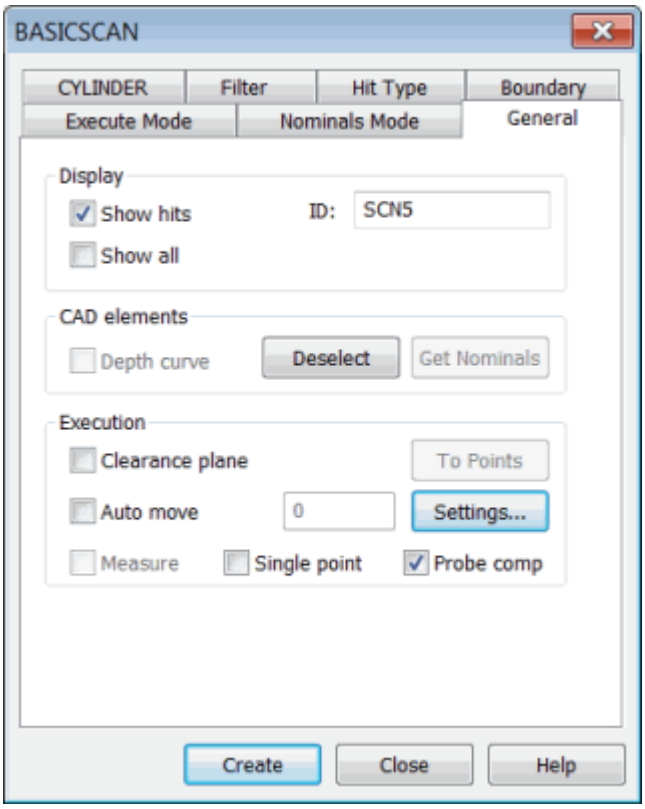
For the circle, cylinder, and axis basic scans, there is no need to have CAD data to find the nominals. To execute the basic scan, PC-DMIS obtains the nominals from the nominal data that you supply.

For example, if you entered nominal data such as center and diameter, to execute the circle, that data is used to find the nominals for the circle scan.

Use BestFit - This check box causes PC-DMIS to use best fit algorithms on the data found from the original FINDNOMS operation and then repeat the FINDNOMS operation. This finds the nominal data with greater accuracy when the actual part has a lot of error with respect to the CAD data.

For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

General Tab



BASICSCAN Dialog Box - General Tab

The following options are available on the **General** tab in the **BASICSCAN** dialog box (**Insert | Scan | Circle**, **Cylinder**, **Axis**, **Center**, or **Line**). For more information about the **BASICSCAN** dialog box, see "Common Functions of the BASICSCAN Dialog Box".

Display Area

- **Show hits** check box: If you select this check box, PC-DMIS displays each scan in the Edit window as a set of the measured hits surrounded by a scan object and an end measured object.
    - If you choose this option, all of the hits appear in the Edit window.
    - If you don't select it, the hits are not shown.
- **ID** box: Displays the current ID for the scan being created. You can change the ID.
- **Show all** check box: If you select this check box, PC-DMIS displays all of the scan parameters in the Edit window, such as:
    - Cut planes
    - Boundary points
    - Direction vector (**DirVec**)
    - Initial approach vector (**InitVec**)

If you don't select the check box, PC-DMIS displays only these data types in the Edit window:

- Increments
- Techniques
- Hit Types

CAD Elements Area

- **Depth curve** check box: The depth curve is used during FINDNOMS operations. Whenever PC-DMIS has to find nominals from curve elements, it takes the vector of the Depth CAD element and crosses it with the vector from the other selected CAD elements to get a plane. It then pierces the plane to get the proper nominal. If many CAD elements are selected, the closest pierce point is used as the nominal point. When CAD wireframe data is being used, PC-DMIS looks for the wire frame data in pairs.
- **Deselect** button: If you select an incorrect surface, you can click **Deselect** to deselect the selected surface (or you can press Ctrl and click on the surface to clear it). You can also click **Deselect** multiple times to deselect one surface at a time from a group of highlighted surfaces until all are deselected.
- **Get Nominals** button: Use this button to find the nominal after a scan has been learned or executed. PC-DMIS finds the nominals for the scan from the CAD data that is available.

Execution Area

- **Clearance plane** check box: To insert a CLEARP MOVE a predetermined distance relative to the current coordinate system and part origin, before taking the first hit, select this check box.

  After the last point in the scan is measured, the probe stays at probe depth until called to the next feature. The use of clearance planes reduces programming time because the need to define intermediate moves is reduced. (For additional information on clearance planes, see "Parameter Settings: Clearplane Tab" in the "Setting Your Preferences" chapter.) This option is only available for DCC scans.
- **Auto move** check box: To turn on Auto Moves for each scan, select this check box. Once it is selected, you can type the distance for the move in the box.
- **Measure** check box: If you select this check box, the measurement process for this auto feature begins immediately after you click the **Create** button.
- **Single point** check box: To consider each hit as a single measured point, select this check box. If you select this check box, PC-DMIS makes each hit a measured point and inserts it into the measurement routine. This sequence will happen after the manual scan has been reduced. If the scan is in DCC Mode, it takes place after the scan has been learned.

- **Probe comp** check box: Use this check box to specify whether or not PC-DMIS turns on probe compensation for this specific scan. In most cases, you probably don't need to perform this compensation, since a scan's FINDNOMS operation automatically does it. However, if you don't have CAD and want to reverse-engineer a part, you should select this check box.
- **To Points** button: To run a process where the individual points that make up an existing scan can be converted into single points after execution, select this button. The scanned set is then deleted.

  For example:

  1. Place your cursor on a scan using hit data in the Edit window.
  2. Press the F9 key to display the **BASICSCAN** dialog box.
  3. Select the **To Points** button. PC-DMIS converts all hit data into single points and deletes the scan from the Edit window.

- **Settings** button: Enables you to change the probe setting that is relevant to scanning: scan speed and scan point density.

Additional Manual Scan Options

In addition to the scanning options detailed in "Common Functions of the Scan Dialog Box", when executing or learning a manual scan, the **Execution** dialog box (**File | Execute** or **File | Partial Execution**) shows the **New Row** and **Scan Done** buttons. For more information on these buttons, see the "Using the Execution Dialog Box" topic in the "Using Advanced File Options: Introduction" chapter.

Inserting Move Commands

Inserting Move Commands: Introduction

This chapter discusses the various move commands that you can place in your measurement routine. Move commands provide you with the ability to alter the movement of the probe between hits.

The main topics covered in this chapter reflect the move commands that you can insert from the **Insert | Move** submenu and any other move commands. These topics include:

- Inserting a Move Point Command
- Using a Moveset Command
- Inserting a Move Increment Command
- Inserting a Move Circular Command
- Inserting Clearance Moves with Collision Detection
- Inserting a Move Sync Command
- Inserting a Move Sweep Command
- Inserting a Move All Command
- Inserting a Move Exclusion Zone Command
- Inserting a Move Rotary Table Command
- Inserting a Clear Point Move
- Inserting a Clearance Plane
- Inserting a Move Clear Plane Command
- Using the ClearanceCube
- Inserting Clearance Moves Automatically

Inserting a Move Point Command

The difference between the Move Point and Move Increment commands is that the Move Point command moves the probe to a specific location in your volume. The Move Increment command moves your probe from its current position to a specified amount in the X, Y, and Z directions. You must be very careful with the Move Increment command since it could cause a crash if you do not account for obstacles when you define this command.

For example, suppose there's a feature on your part that you need to avoid crashing into and you perform two incremental moves of X=0, Y=0, Z=50 and X=100, Y=0, Z=0. This defines probe movements of 50 units in the Z direction and 100 units in the X direction. If you get a false trigger halfway through the second move increment command and you click **Continue**, your probe, which has already moved 50 units, will proceed to make the same 100 unit move from where the false trigger occurred. The net result is that your probe will move a total of 150 units and may end up crashing into another feature.

You can use the **Move Point** command to insert a move point into your measurement routine. When PC-DMIS encounters this command, the probe moves to that XYZ location.

To use this option, select **Insert | Move | Move Point** or **Operation | Move To**.

- On the **Insert | Move** menu, there are two **Move Point** commands. The first **Move Point** automatically inserts a move point command at the current probe location. You need to modify the XYZ values in the Edit window.
- If you selected **Operation | Move To**, or if you selected the second **Insert | Move | Move Point** menu, PC-DMIS displays the **Move Point** dialog box. With this dialog box, you can fill out the information before PC-DMIS inserts the command.

If you attempt to insert the move point in a location that PC-DMIS does not allow, the software asks you if you want to insert the move point at the next valid location.

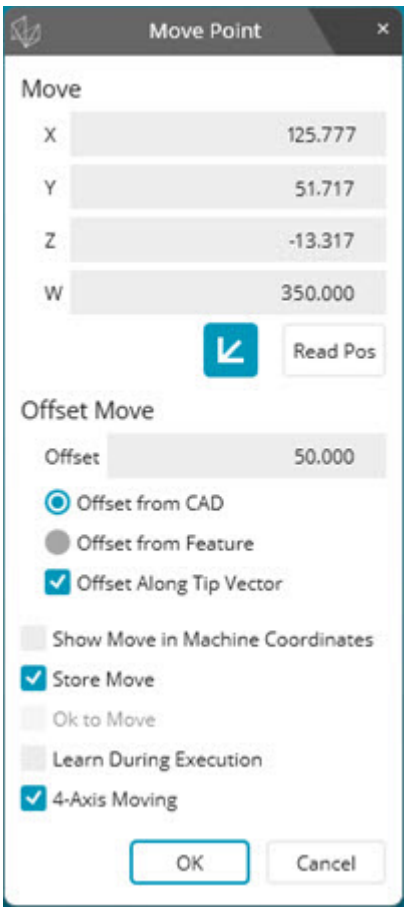
The Edit window command line for a sample Move Point reads:

```
MOVE/POINT tog1,<x,y,z>
```

tog1 - This field displays either NORMAL or RELEARN.

x,y,z,w - These fields display the X, Y, Z, and if a rotary table is defined in your measurement routine, the W component which determines the rotary table's rotational angle move point location. The location is always displayed using the current part alignment's coordinates.

To change the values that appear in the Edit window, select the desired value and type a new value, or press F9 on the command to open the **Move Point** dialog box.



Move Point Dialog Box

This dialog box provides you with a more user-friendly way of creating or editing the `MOVE/POINT` command as well as additional options for the move point. The options in the **Move Point** dialog box are discussed in the topics listed below.

If you want to access the dialog box before inserting a MOVE/POINT command, select **Operation | Move To**.

When you insert a `MOVE/POINT` command, PC-DMIS automatically marks it for execution.

Using Move Points While Animating

If you select this menu option during probe path animation with the CMM, PC-DMIS always asks whether to insert the move before or after the current position. The current position appears in the status bar.

Inserting Move Points from the Status Bar

You can also insert a move point while the system is in Program mode. To insert it, place the cursor on the Status bar over the readouts and click the left mouse button.

Inserting Move Points from the Jog Box

You can press the **PRINT** button on the jog box (on Sharpe32 systems) while in Learn mode to automatically insert a move into the measurement routine. The `PrintButton` entry must be set to **True** for this to work.

If the **4-Axis Moving** option is selected when you press the **PRINT** button on the jog box, PC-DMIS adds the W angle after the X, Y, and Z position in the Edit window's command line. Using the **Move Point** dialog box example image above, you can see that the W angle is set to 350 which creates this command line in the Edit window:

```
MOVE/POINT,NORMAL,<125.777,51.717,-13.317>,350
```

When you execute the above command, the X, Y, and Z movement and the rotary table rotation movement are performed in a synchronized motion.

Moving In Only 1 or 2 Axes

In the Edit window in Command mode, you can set one of the axis values to blank (which means that you completely delete the value) to give the axis a relative move of zero during measurement routine execution. This essentially lets you have a hybrid relative and absolute move point behavior using the `MOVE/POINT` command.

This feature only works fully in Command mode. If you use Summary mode to do this, the field disappears from the Edit window. If you want to then set it back to a number value, you need to set it in Command mode.

For example, suppose that you want to move in only two axes. Create an Edge Point feature followed by a `MOVE/POINT` command. Execute the measurement routine. Now, delete the X value of the command and execute the measurement routine again. There won't be any X component in the move point motion, regardless of where you take the edge point.

Move

Move Area

The **Move** area contains **XYZ** boxes that display the X, Y, and Z nominal for the position of the `MOVE/POINT` or the increment distance of the `MOVE/INCREMENT` command.

To change the X, Y, or Z value, follow these steps:

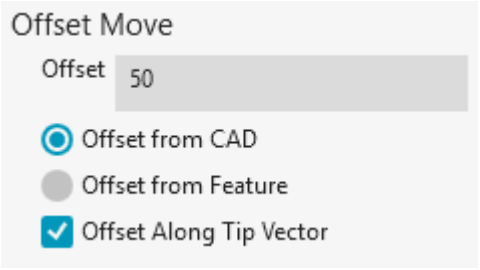
1. Select the value that you want to change.
2. Type a new value.

After you enter a new value, PC-DMIS draws the animated probe at the new location. This location indicates the location of the stored move. When you click the **Create** button, PC-DMIS inserts the Move command and positions the CMM to the location that you specified. If you created an offset move, PC-DMIS updates the values in the **XYZ** boxes to match the computed offset location.

- The **Cartesian/Polar** icon switches the display of the X, Y, and Z values between the Cartesian and Polar coordinate systems. The **XYZ** labels display **RAH** when you switch to Polar.

The **Read Pos** button reads the current position of the probe and places those values in the **X**, **Y**, and **Z** boxes. This is helpful if you have manually moved your probe to a new location and you want to use its current position as your move point.

Offset Move



Offset Move Area

The **Offset Move** area lets you create move points offset from a selected CAD point or from a selected feature's centroid.

Offset - This box determines the offset distance from the CAD point or feature centroid. The unit of measurement is the same as your measurement routine. The default is 50 mm or 1.96 inches.

Offset from CAD - If you select this option and click on the CAD, PC-DMIS computes the move point at the selected point on the surface and offsets the move point by the specified distance.

- If you clear the **Offset Along Tip Vector** check box, PC-DMIS creates the move point offset away from the surface along the surface's normal vector.
- If you select the **Offset Along Tip Vector** check box, PC-DMIS creates the move point in the direction of the probe tip's vector.

Offset from Feature - If you select this option and click on a feature, PC-DMIS computes the move point by the offset distance at the feature's centroid.

- If you clear the **Offset Along Tip Vector** check box, PC-DMIS creates the move point offset in the direction of the feature's vector.

- If you select the **Offset Along Tip Vector** check box, PC-DMIS creates the move point in the direction of the probe tip's vector.

Offset Along Tip Vector - You can select this check box to offset the move point along the active tip's vector. If you clear the check box, PC-DMIS offsets the move point along either the CAD's surface vector or along the feature's vector.

Different feature types have different vectors. When the software offsets along a vector of the feature, PC-DMIS uses the vector that gives the best representation of being "outward" from the surface of the part.

Show Move in Machine Coordinates

- If you mark the **Show Move In Machine Coordinates** check box, PC-DMIS displays the move point's location in the dialog box in machine coordinates.
- If you clear the **Show Move In Machine Coordinates** check box, PC-DMIS displays the move point's location in the dialog box in the current part alignment's coordinates.

This does not affect how the command appears; the command always appears with the current part alignment's coordinates.

Store Move

If you mark the **Store Move** check box, PC-DMIS inserts the appropriate move command (`MOVE/POINT` or `MOVE/INCREMENT`) for the given X, Y, Z values into the measurement routine.

OK to Move

If you select the **OK to Move** check box, PC-DMIS immediately moves the probe to the X, Y, Z position (or by the X,Y,Z increment amount) when you click **OK** or **Create**. This allows you to position the CMM without storing the move.

This option fills in the X, Y, Z values with the current position of the probe.

Learn During Execution

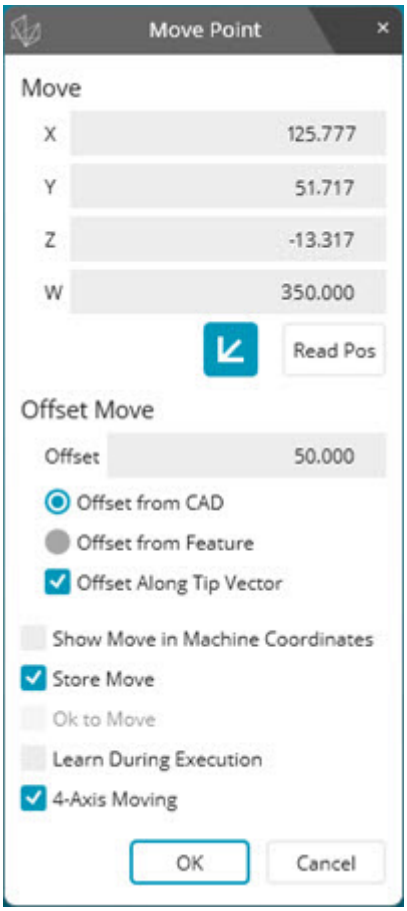
If you mark the **Learn During Execution** check box, PC-DMIS moves to and relearns the XYZ point location that appears in the **Move Point** dialog box during execution. You must also mark the **Store Move** check box for this command to work. For information, see "Store Move".

The Edit window command line for a move point that PC-DMIS relearns during measurement routine execution would read:

```
MOVE/POINT, RELEARN
```

4-Axis Moving

The 4-Axis Moving option provides you with the option to move the rotary table to a specific location and then align the probe to that location.



Move Point Dialog Box

An example for a typical command line in the Edit window for the above settings in the **Move Point** dialog box would look like this:

```
MOVE/POINT,NORMAL,<125.777,51.717,-13.317>,350
```

When this command is executed, the `MOVE/POINT` command moves the rotary table to the W angle position defined in the command as 350 degrees.

Using a Moveset Command

About the Moveset Command

A Moveset command is a set of move points that you can store in a moveset. You can use the moveset to use with a ClearanceCube.

- A Moveset command has a feature ID and a ClearanceCube face that you can use to set for that feature ID.
- A Moveset command provides you with the ability to use ClearanceCube on parts with complex geometries.

Creating a Moveset Command

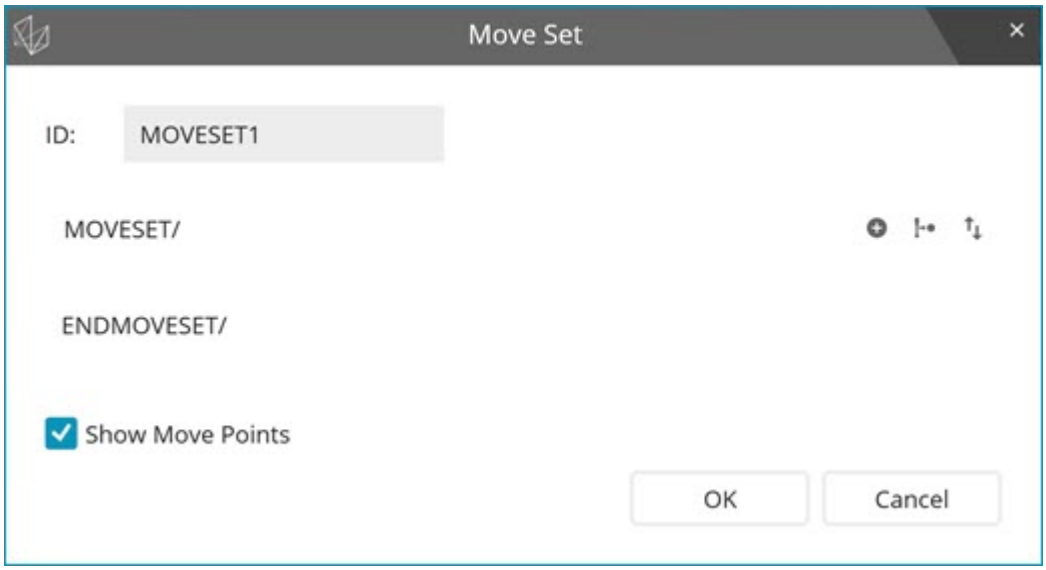
Use one of these methods to create a `MOVESET` command in the Edit window:

Method 1

1. Insert Move Points in the Edit window. To insert a Move Point, see "Inserting a Move Point Command".
2. Select the Move Points that you want to add to the `MOVESET` command.
3. Right-click the selected Move Points and click the **Convert to MoveSet** option.

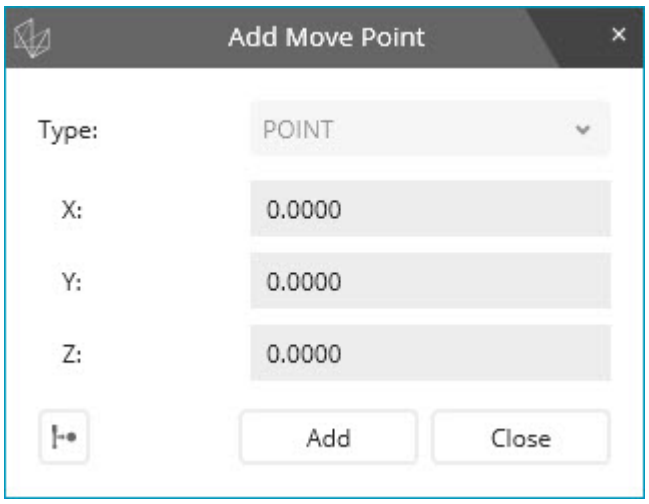
Method 2

1. Open the **Move Set** dialog box (**Insert | Move | Moveset**).



Move Set Dialog Box

2. From the **Move Set** dialog box, click the **Add** button to open the **Add Move Point** dialog box.



Add Move Point Dialog Box

3. From the **Add Move Point** dialog box, type the X, Y, and Z values in the **X**, **Y**, and **Z** boxes and then click **Add**. You can also click the **Read Machine Position** button or press the **Print** button on the jog box to add Move Points at the current tip position.

4. You can add multiple Move Points and then click **Close** to return to the **Move Set** dialog box.

5. From the **Move Set** dialog box, click the **Read Machine Position** button to add a Move Point at the current machine position.

6. To arrange Move Points in a specific order, click and drag a Move Point to a desired location. To reverse the order of an entire set of Move Points, click the **Reverse** button.

7. From the **Move Set** dialog box, click **OK**.

> Click the **Cancel** button before you click the **OK** button to close the dialog box without inserting a `MOVESET` command.

Editing a Moveset Command

To edit a `MOVESET` command, follow these steps:

1. Place your cursor on the command, right-click on it, and select **Edit** to open the **Move Set** dialog box.

2. Select a `MOVESET` command from the **MOVESET/ ENDMOVESET/** area.

3. To open the **Edit Move Point** dialog box, click **Edit** or double-click on the `MOVESET` command.

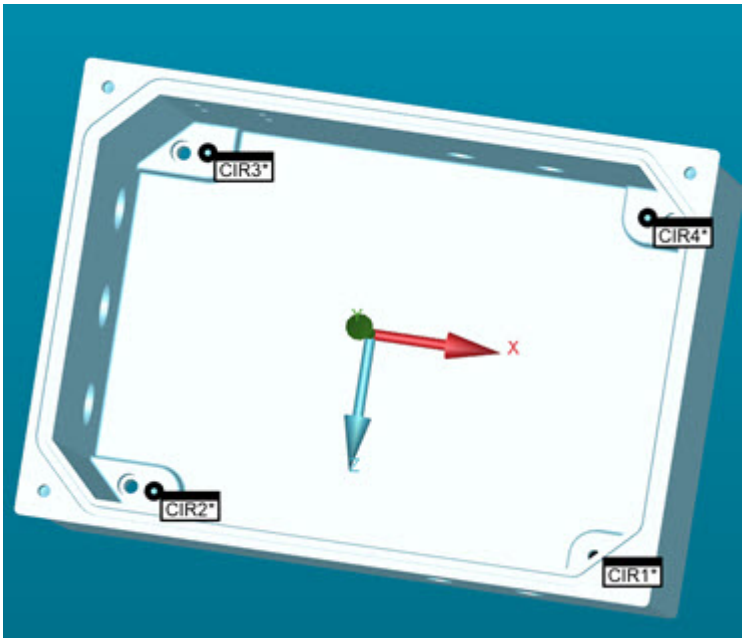
4. To remove a MOVESET command, click .

Using a Moveset Command with ClearanceCube

You can use a MOVESET command to implement ClearanceCube for complex geometric parts.

A MOVESET command enables you to measure selected dimensions using mini-routines with a ClearanceCube.

Suppose you want to measure the four circles shown in the image.



Example of using Moveset with a NIST part.

Since the holes are too deep inside the part, if you define a ClearanceCube for each circle, the probe tip moves too far before and after measuring the circles.

You may also want to use a mini-routine and perform a measure dimension that is associated with just one circle.

To do this, you can use the MOVESET command.

```
GRP1          =GROUP/SHOWALLPARAMS=YES
               TIP/TIP1,
               SHANKIJK=0, 0, 1, ANGLE=0
MOVESET1      =MOVESET/
CIR1          =FEAT/VISION/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
               THEO/
<4.291,1.2208,-0.7874>,<0,0,1>,0.6457,0,360
               ACTL/
<4.291,1.2208,-0.7874>,<0,0,1>,0.6457,0,360
               TARG/<4.291,1.2208,-0.7874>,<0,0,1>
               ANGLE
VEC=<1,0,0>
               SHOW
FEATURE PARAMETERS=NO
               SHOW_VISION_PARAMETERS=NO
CIR2          =FEAT/VISION/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
               THEO/<4.7638,2.3622,0>,<0,0,1>,0.6457,0,360
               ACTL/<4.7638,2.3622,0>,<0,0,1>,0.6457,0,360
               TARG/<4.7638,2.3622,0>,<0,0,1>
               ANGLE
VEC=<1,0,0>
               SHOW
FEATURE PARAMETERS=NO
               SHOW_VISION_PARAMETERS=NO
CIR3          =FEAT/VISION/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
               THEO/<4.291,3.5036,0>,<0,0,1>,0.6457,0,360
               ACTL/<4.291,3.5036,0>,<0,0,1>,0.6457,0,360
               TARG/<4.291,3.5036,0>,<0,0,1>
               ANGLE
VEC=<1,0,0>
               SHOW
FEATURE PARAMETERS=NO
               SHOW_VISION_PARAMETERS=NO
CIR4          =FEAT/VISION/CIRCLE/
DEFAULT,CARTESIAN,IN,LEAST_SQR
               THEO/<3.1496,3.9764,0>,<0,0,1>,0.6457,0,360
               ACTL/<3.1496,3.9764,0>,<0,0,1>,0.6457,0,360
               TARG/<3.1496,3.9764,0>,<0,0,1>
               ANGLE
VEC=<1,0,0>
               SHOW
FEATURE PARAMETERS=NO
               SHOW_VISION_PARAMETERS=NO
MOVESET2      =MOVESET/
ENDGROUP/ID=GRP1
```

This example shows you how to create a group to measure circles deep within a part. You must insert the `MOVESET` commands before the first measured circle and after the last measured circle.

You must set the ClearanceCube to active for MOVESET commands and then set it to inactive for the circles in a group.

- The first MOVESET command has ClearanceCube at the start.
- The last MOVESET command has ClearanceCube at the end.
- The ClearanceCube for all circles between the two MOVESET commands is set as off.
- The avoidance moves define the motion of the probe between the measurements of the circles.

Important Notes

This important information may help you to work on a MOVESET command:

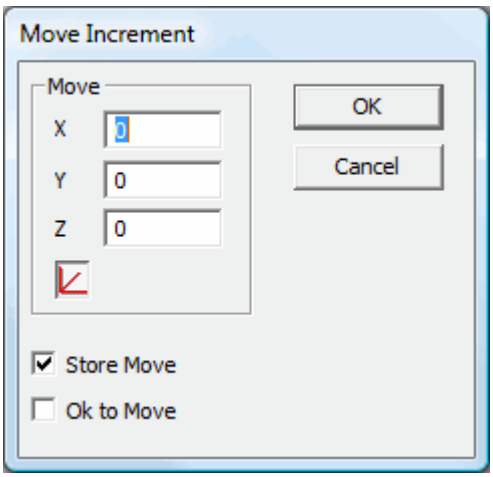
- The MOVESET command supports move points only of the Point type. You cannot use a MOVESET command for incremental moves or any other type of moves.
- You can use the MOVESET command for measurement routines that do not use ClearanceCube. This helps to place multiple move points in a single MOVESET command.

Inserting a Move Increment Command

The difference between the Move Point and Move Increment commands is that the Move Point command moves the probe to a specific location in your volume. The Move Increment command moves your probe from its current position to a specified amount in the X, Y, and Z directions. You must be very careful with the Move Increment command since it could cause a crash if you do not account for obstacles when you define this command.

For example, suppose there's a feature on your part that you need to avoid crashing into and you perform two incremental moves of X=0, Y=0, Z=50 and X=100, Y=0, Z=0. This defines probe movements of 50 units in the Z direction and 100 units in the X direction. If you get a false trigger halfway through the second move increment command and you click **Continue**, your probe, which has already moved 50 units, will proceed to make the same 100 unit move from where the false trigger occurred. The net result is that your probe will move a total of 150 units and may end up crashing into another feature.

The **Insert | Move | Move Increment** menu option displays the **Move Increment** dialog box. You can use this dialog box to define the move increment command before you insert the command into your measurement routine.



Move Increment Dialog Box

When you click **OK**, PC-DMIS inserts the Move Increment command.

The Edit window command line for a Move Increment would read: `MOVE/INCREMENT, x, y, z`

When PC-DMIS encounters this command, it moves the probe the indicated XYZ *distance* (not an XYZ *location* as in a `MOVE/POINT` command). You must select the **Store Move** check box for this command to appear in the Edit window.

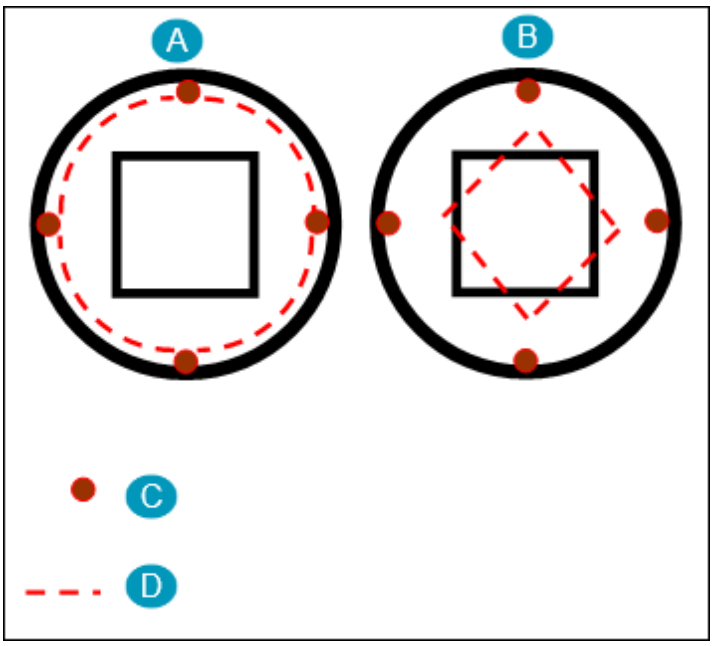
For descriptions of the **Move** area and the **Store Move** and **Ok to Move** check boxes, see the same topics under the **Move Point** dialog box above.

Inserting a Move Circular Command

The **Insert | Move | Move Circular** option allows you to insert a `MOVE/CIRCULAR` command at the cursor's location in the Edit window.

When PC-DMIS encounters this command, the probe follows a circular path when it takes the next hit. It moves from one point to another in a circular fashion based on the radius of the feature you are measuring.

By taking a circular path, the probe may avoid colliding with certain features that would otherwise be in the way of the standard straight line path. For example, if you are measuring a hole that has a square pin inside, that pin may block your probe path unless you use Move Circular.



A. *Circular path avoids inside feature*

B. *Normal path collides with inside feature*

C. *Probe Hits*

D. *Probe Path*

Overall, this command has very specific uses, although using it to measure a pin speeds up measurement somewhat since PC-DMIS doesn't have to move as far away from its last hit.

Inserting Clearance Moves with Collision Detection

When marked, the **Operations | Graphic Display window | Clearance Moves | With Collision Detection** menu item adjusts the Auto feature path line preview from the previous feature to the current feature to avoid possible collisions between the two features. When you create the Auto feature, PC-DMIS automatically inserts one or more `MOVE/POINT` commands into your measurement routine to avoid the collision. See "Show Hit Targets Toggle" in the "Creating Auto Features" chapter where the Auto feature path line previews are discussed.

Inserting a Move Sync Command

Multiple arm measurement routines have commands that control multiple arms. Each machine executes these commands separately, keeping both arms moving.

The **Insert | Move | Move Sync** menu option allows you to insert a `MOVE/SYNC` command into the Edit window.

Using this command on dual-arm machines allows you to synchronize the motion of the arms. Whichever arm reaches the `MOVE/SYNC` command in the measurement routine first will stop and wait until the other arm also arrives before they proceed together with the rest of the measurement routine.

Inserting a Move Sweep Command

The `Move/All` command replaces the `Move/Sweep` command. Please use the `Move/All` command instead of the `Move/Sweep` command. For details on the `Move/All` command, see the "Inserting a Move All Command" topic in the PC-DMIS Core documentation.

The **Insert | Move | Move Sweep** menu item inserts the following sweep block into the Edit window.

```
SWEEPSTART/

MOVE/SWEEP

SWEEPEND/
```

Movement commands that are inside of `SWEEPSTART` and `SWEEPEND` command blocks cause the probe to move in all five axes simultaneously. This occurs when you use certain controllers, such as the PHS wrist with a Sharpe32Z controller.

Three of these axes are the X, Y, and Z axes. These are used when moving the machine itself. The other two axes refer to the probe's AB angle movement.

This command only functions with a continuous type wrist (CW43, CW43Light, or PHS) and a Renishaw SP600 analog probe.

Movement in these five axes takes place simultaneously. This differs from a PH9 wrist, which must first move the machine in the X, Y, and Z axes. Then, when the machine stops, the probe orientation (AB angles) moves.

Suppose your current CMM position is 10, 10, 10 (in X, Y, and Z), and your wrist orientation (AB position) is 0, 0. You want to move the CMM to 20, 20, 20 and the probe orientation to 0, 180. A PHS wrist using the `Move/Sweep` commands can perform both movements simultaneously. A PH9 wrist first moves the CMM and then moves the wrist orientation.

For comparison:

- A PH9 wrist can move between 0º and 110º in the A axis and between +180º to -180º in the B axis.
- A PHS wrist can move between -180º and +180º in both the A and B axes at 0.1º increments.

Advantages of using `MOVE/SWEEP` commands include:

- A much more fluid movement. This makes it easier to measure hard to reach features.
- Saved time. You don't need to wait for the CMM's ram and arm to move to the X, Y, and Z location before changing the probe's orientation.
- A slight improvement in accuracy from reduced vibration.

PC-DMIS requires at least three `MOVE/SWEEP` commands between the `SWEEPSTART/` and `SWEEPEND/` command block. These move commands are essential to generate the elliptical movement path.

Inserting a Move All Command

The `MOVE/ALL` command will function with any motorized probe head - continuous or indexing - and any probe combination.

The **Insert | Move | Move All** menu option inserts a `MOVE/ALL` command into the Edit window.

A `MOVE/ALL` command is the same as a `MOVE/POINT` command combined with a TIP / command. This command has the benefit of rotating the probe head together with machine movement.

When using a wrist device (PHS, CW43L, or CW43) this movement is synchronized such that the wrist's AB movement will start and end its motion at the same time that the machine starts and ends the X,Y, and Z movement.

When using this command with an indexable wrist such as a PH10M, this movement cannot be synchronized and will occur at an unspecified time during the machine movement (depending on time delays in the signal processing of the electronic hardware). Because of this, the AB wrist movement may not always complete by the time the machine performs the X, Y, and Z move.

The Edit window command looks like this:

```
MOVE/ALL,X,Y,Z,TIP=T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
```

- `X,Y,Z` allow you to specify the X, Y, and Z coordinates to which to move the probe.
- `TIP= T1A0B0` shows the AB wrist movement to perform.
- `SHANKIJK = IJK` allows you to specify the shank's vector.
- `ANGLE=0` specifies the theoretical angle.

Note that the probe head angle is usually be a predefined TIP position.

Maestro Users

For Maestro CMM users, PC-DMIS provides a `DIRECTION` option that you can use with the `MOVE/ALL` command:

- `DIRECTION=CCW` - This option allows you to define the swivel direction of the wrist as either `CW` , `CCW` , or `LEGACY` . The settings for this option are:

  `CW` - This setting rotates the wrist in the clockwise direction.

  `CCW` - This setting rotates the wrist in the counter-clockwise direction.

  `LEGACY` - This setting allows for the Maestro system to determine the direction of the wrist rotation.

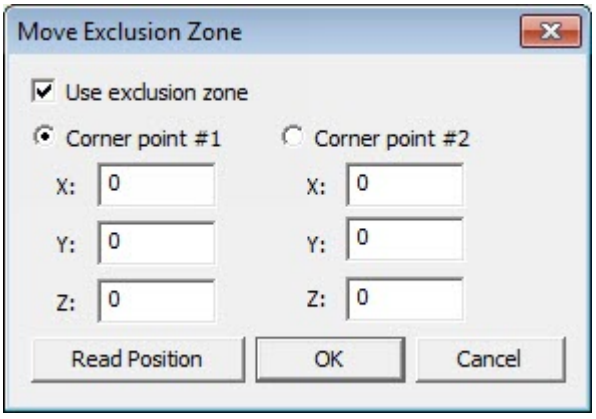
  The command in the Edit window for each of these settings might look something like:

```
MOVE/
ALL,<10,10,0>,TIP=T1A45B45,SHANKIJK=-0.5,0.707,0.5,ANGLE
=180,DIRECTION=CW

MOVE/
ALL,<20,200,0>,TIP=T1A45B45,SHANKIJK=-0.5,0.707,0.5,ANGL
E=180,DIRECTION=CCW

MOVE/
ALL,<0,0,0>,TIP=T1A0B0,SHANKIJK=0,1,0,ANGLE=-90,DIRECTIO
N=LEGACY
```

Inserting a Move Exclusion Zone Command

The **Insert | Move | Exclusion Zone** menu option allows you to insert a move exclusion zone so that the two arms on a dual arm system don't collide with each other.

Selecting this option opens the **Move Exclusion Zone** dialog box.



Move Exclusion Zone Dialog Box

You can use this dialog box to insert a `MOVE/EXCLUSIVE_ZONE` command into the Edit window.

The `MOVE/EXCLUSIVE_ZONE` command applies to either the Arm1 or the Arm2.

Before PC-DMIS executes this command, it makes sure that the arm assigned to the command has not already requested an exclusive move zone that overlaps with the new request.

If there is a conflict of movement, the `MOVE/EXCLUSIVE_ZONE` command waits until the arm assigned to the command has released the commanded volume, and then it proceeds to execute the motion commands listed below the `MOVE/EXCLUSIVE_ZONE` command.

Defining a Move Exclusion Zone

There are some things to keep in mind when you define an exclusion zone (**Insert | Move | Exclusion Zone** menu option):

- In the measurement routine, you need to have a `MOVE/EXCLUSIVE_ZONE` command before each section of commands that move either arm into the overlapping region of the dual arm volume. The `MOVE/EXCLUSIVE_ZONE` command should define a 3D box around all the motion that is about to be commanded in the overlapped area of the dual-arm CMM.
- There must be a `MOVE/EXCLUSIVE_ZONE=OFF` command after the probe has been withdrawn from the overlapped part of the dual-arm CMM.

For example, if you want each arm to check a common sphere using automatic sphere commands, you should set up the measurement routine like this:

`MOVE/EXCLUSIVE_ZONE=ON` (for Arm 1)
`AUTO/SPHERE` (for Arm 1)
`MOVE/EXCLUSIVE_ZONE=OFF` (for Arm 1)
`MOVE/EXCLUSIVE_ZONE=ON` (for Arm 2)
`AUTO/SPHERE` (for Arm 2)
`MOVE/EXCLUSIVE_ZONE=OFF` (for Arm 2)

The following steps describe how to define the exclusion zone.

Step 1 Enable the Exclusion Zone

Select the **Use exclusion zone** check box. This allows you to edit the X, Y, and Z values under the **Corner point #1** and **Corner point #2** options.

Alternately, if you click on an already inserted `MOVE/EXCLUSIVE_ZONE` command in the Edit window and press the F9 key, the **Move Exclusion Zone** dialog box opens, and you are able to select or deselect this check box.

- If this box *is selected* when you click the **OK** button, PC-DMIS will display the following command in the Edit window:
  `MOVE/EXCLUSIVE_ZONE=ON, CORNER1=x,y,z,CORNER2=x,y,z`
- If this box is *not selected* when you click the **OK** button, PC-DMIS will display the following command in the Edit window:
  `MOVE/EXCLUSIVE_ZONE=OFF`

Step 2 Enter Corner Point Values

Type the X, Y, and Z values for **Corner point #1** and **Corner point #2**. You can read in the probe's current into the selected corner point by clicking the **Read Position** button.

Defining two corners sets up the exclusion zone; be sure to select the corner points that will correctly define the exclusion zone. The two points (corner 1 and corner 2) represent two corners that are *diagonal* from each other.

A rectangular zone can be created in 3D space from two points if you use the current coordinate system to create the sides. A combination of different parts of the two points can construct the eight points needed to form the rectangular area.

This is shown below:

First corner = X1, Y1, Z1

Second corner = X2, Y2, Z2

Third corner = X1, Y1, Z2

Forth corner = X1, Y2, Z1

Fifth corner = X1, Y2, Z2

Sixth corner = X2, Y1, Z1

Seventh corner = X2, Y1, Z2

Eight corner = X2, Y2, Z1

Step 3 Click OK

Finish defining the Exclusion Zone by clicking the **OK** button. PC-DMIS then places the defined information into the Edit window, which reads:

```
MOVE/EXCLUSIVE_ZONE=TOG1,CORNER1=X,Y,Z,CORNER2=X,Y,Z
```

TOG1
This defines whether the exclusion zone is in effect or not. This toggle field switches between ON and OFF.

X, Y, Z
These coordinates define the corner points used to form the exclusion zone.

To close the dialog box without making any changes to your measurement routine, click the **Cancel** button.

Move Exclusion Zones with Iterative Alignments

If you use exclusion zones (**Insert | Move | Exclusion Zone**) with iterative alignments, PC-DMIS automatically computes the starting and ending lines of the measurement routine that are to be re-executed. This means that PC-DMIS ends execution at the last feature used as part of the iterative alignment. This could prevent an arm from releasing the exclusion zone to the other arm, which thereby stops the measurement routine.

To resolve this, insert a start label for the iterative alignment. PC-DMIS executes from the label to the command just in front of the iterative alignment, which causes the arms to release the exclusion zone.

For information on how to use a start label with iterative alignments, see "Start Label" under "Description of the Iterative Alignment Dialog Box".

Inserting a Move Rotary Table Command

The **Insert | Move | Move Rotary Table** menu item allows you insert a `MOVE/ROTAB` command at the cursor's current location.

Command line in the Edit window:

```
MOVE/ROTAB,angle,DIRECTION,feature
```

Command line with a stacked configuration:

```
MOVE/ROTAB, angle, DIRECTION, angle2, DIRECTION2,feature
```

`Direction` = determines the direction of the table rotation. The available options are:

Clockwise
Rotates the table in a clockwise direction until it reaches the angle entered in the **Rotate Table Angle** box.

Counterclockwise
Rotates the table in a counterclockwise direction until it reaches the angle entered in the **Rotate Table Angle** box.

Shortest
Rotates taking the shortest route (either clockwise or counterclockwise) until it reaches the angle entered in the **Rotate Table Angle** box.

`angle` and `angle2` = the relative offset angle used to rotate the table.

`feature` = the feature or alignment to which that the table will rotate to.

When PC-DMIS encounters this command, it rotates the table to a position determined by whichever `MOVE/ROTAB` command is active. If both an angle and a feature are defined, PC-DMIS rotates to an angle which is the sum of the angle to rotate the defined feature to +Z and the offset angle.

Consider a measurement routine that has multiple different `MOVE/ROTAB` commands. If you click on a particular portion of the measurement routine and select the **Move Rotary Table** menu option, PC-DMIS rotates the table to the position determined by whichever `MOVE/ROTAB` command is active at that point in the measurement routine.

This menu option is only available if your PC-DMIS license is configured for rotary tables.

Inserting a Clear Point Move

The **Insert | Move | Clear Point** menu option inserts a `MOVE/CLEARPOINT` command into the Edit window. This command, when executed, moves to the clear point position that is defined in the **Clear Point** tab of the **Parameter Settings** dialog box. In addition, it uses the move sequence defined in the **Clear Point** tab to control the form of the move. See the "Parameter Settings: Clear Point tab" topic in the "Setting Your Preferences" chapter.

Inserting a Clearance Plane

The `CLEARP` command (**Insert | Parameter Change | Clearance Planes**) allows you to define a clearance plane value for the current workplane. Clearance planes, in essence, create an envelope around a part where the probe will always travel to when it moves from one feature to another.

The **Clearance Plane** tab of the **Parameters** dialog box allows you to turn clearance planes on globally. The `CLEARP` command only defines the clearance plane for the first hit of a feature. To use the defined clearance plane, open a new line, at any place beneath the `CLEARP` command location in the Edit window, and type the command: `MOVE/CLEARPLANE` , or select **Insert | Move | Move Clearplane** from the menu.

To define a new clearance plane, select **Edit | Preferences | Parameters**, and then use the **Clearance Plane** tab to define the new plane's values.

For an in-depth discussion on how to define and work with clearance planes, see "Parameter Settings: Clearance Plane tab" in the "Setting Your Preferences" chapter.

Inserting a Move Clear Plane Command

The **Insert | Move | Move Clearplane** command allows you to insert the `MOVE/CLEARPLANE` command between blocks and within features.

A `MOVE/CLEARPLANE` command works in conjunction with a preceding `CLEARP/` command and a following motion command to move to a clearance plane. The `MOVE/`

`CLEARPLANE` command gives permission for the next move, measure, tip selection, or Auto feature command to move to the predefined clearance plane prior to execution.

The probe remains at the clearance plane until a command is encountered that moves the probe away from the clearance plane, such as a `MEAS HIT` , or `MOVE/POINT` command. If you want the probe to move to the clearance plane before each hit of a measured feature, then a `MOVE/CLEARPLANE` command must be inserted directly before each `HIT/` command.

PC-DMIS searches for the clearance plane definition. For this command to work, the parameters must have already been set.

To set the parameters for a Clearplane command:

1. Access the **Parameter Settings** dialog box (**Edit | Preferences | Parameters**).
2. Select the **Clearance Plane** tab.
3. Type the distance for the **Value** box in the **Active Plane** area.
4. Type the distance for the clearance plane in the **Value** box of the **Pass Through** area.
5. Select the **Clearance Planes Active** check box.
6. Click the **OK** button.

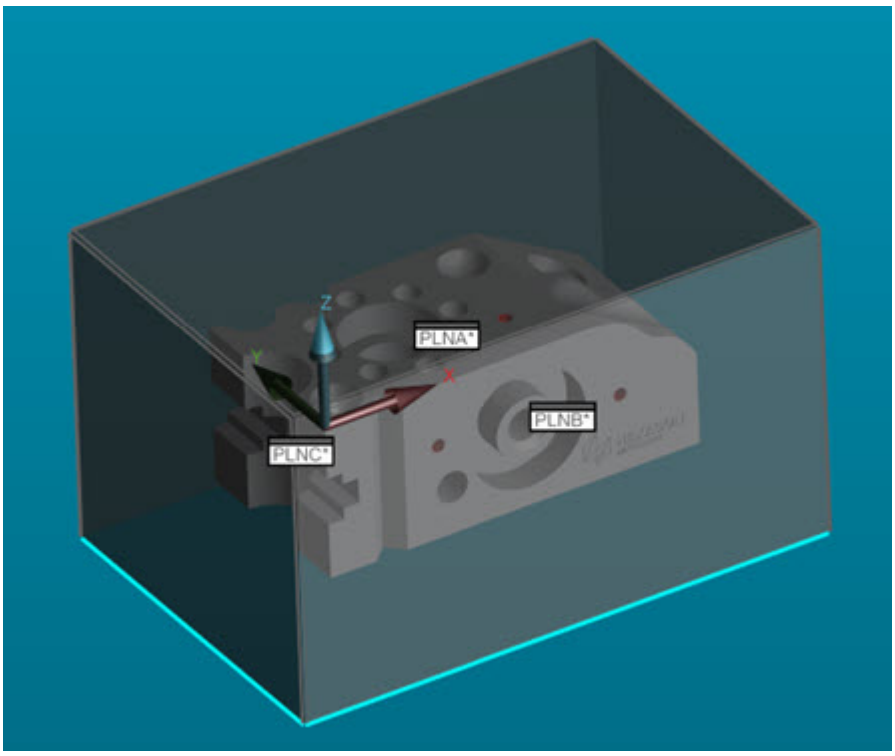
If the parameters have not been set, PC-DMIS displays an error message.

The Edit window command line for a sample Move Clearplane would read:
`MOVE/CLEARPLANE`

For an in-depth discussion on how to define and work with clearance planes, see "Parameter Settings: Clearance Plane tab" in the "Setting Your Preferences" chapter.

Using the ClearanceCube

A ClearanceCube is a three-dimensional virtual box that you define in PC-DMIS. It's purpose is to act as an invisible protective envelope that surrounds your part and its fixture. During measurement routine execution, PC-DMIS uses the faces of the ClearanceCube as safe pathways for your probe to travel to avoid collisions with your part.



A sample ClearanceCube surrounding the CAD model.

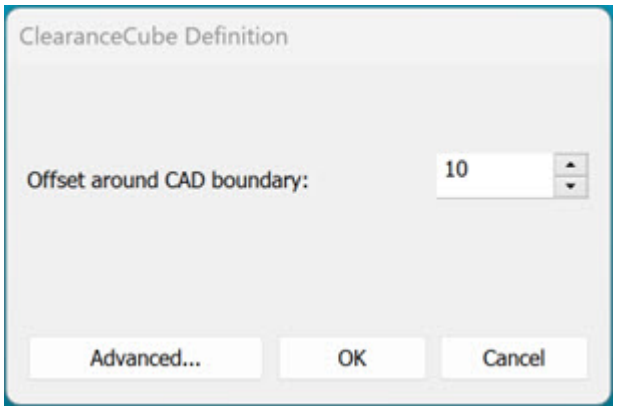
Defining The ClearanceCube

You can define the ClearanceCube in a measurement routine with or without a CAD model.

Measurement Routine with CAD

To define a ClearanceCube in a measurement routine with a CAD model, follow these steps:

1. Select **Operation | Graphic Display Window | ClearanceCube**, or from the **ClearanceCube** toolbar, click the **Clearance Cube Definition** icon . This opens the **ClearanceCube Definition** dialog box.



ClearanceCube Definition Dialog Box

Offset around CAD boundary - Type an offset value in this box or use the up and down arrows to increase or decrease the current value by one. Based on this value, PC-DMIS draws the ClearanceCube around the CAD model the current distance away from the part on all six sides. PC-DMIS uses the same units of measurement as the measurement routine.

Advanced - This button expands the dialog box from a single-tab to a multi-tabbed dialog box. The tabs on the advanced dialog box include: **Size** and **Constraints**. This button then changes to read **Simple**. Click the button again to display the simple dialog box.

You can use the advanced area to:

- Customize the ClearanceCube offset along each axis.
- Define a ClearanceCube for a measurement routine that does not use a CAD model.
- Set the constraints for the ClearanceCube.

If you use the **Advanced** tab to customize the size, the offset box in the simple definition dialog box displays "N/A".

OK - This button defines and activates the ClearanceCube.

Cancel - This closes the dialog box without saving the changes.

2. Type an offset value in the **Offset around CAD boundary** box.
3. Click the **Advanced** button to customize the size and constraints for the ClearanceCube.
4. Click **OK** to close the dialog box.

Measurement Routine without CAD

To define a ClearanceCube in a measurement routine without a CAD model, follow these steps:

Prerequisites

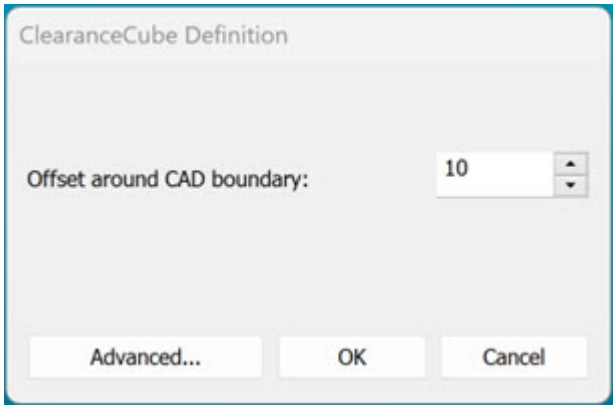
1. Create an alignment that controls all six degrees of freedom of the part.
2. Set **CAD = Part** (**Operation | Graphic Display Window | CAD Equals Part**).

You must create an alignment and then set CAD Equals Part even though you do not have a CAD model in your measurement routine.

If you don't complete the prerequisite steps before you define the ClearanceCube, PC-DMIS displays an error that informs you to complete the required steps.

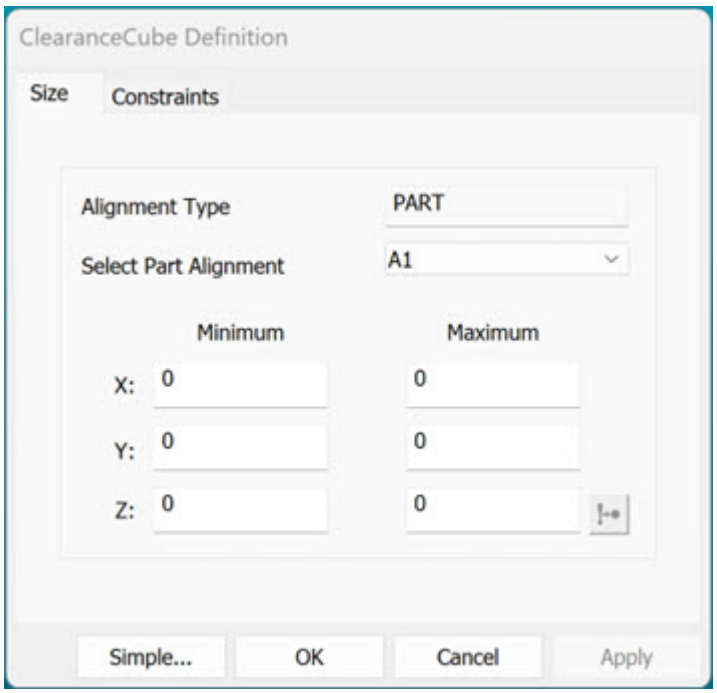
Define ClearanceCube

1. Select **Operation | Graphic Display Window | ClearanceCube**, or from the **ClearanceCube** toolbar, click the **Clearance Cube Definition** icon . This opens the **ClearanceCube Definition** dialog box.



ClearanceCube Definition Dialog Box

2. Click **Advanced** to open the **Size** and **Constraints** tabs on the **ClearanceCube Definition** dialog box.



ClearanceCube Definition Dialog Box - Advanced Option

3. In the **Size** tab, from the **Select Part Alignment** list, select the part alignment in which you want to define ClearanceCube.

4. In the **Minimum** and **Maximum** area, type the location values for the ClearanceCube in the **Minimum XYZ** and **Maximum XYZ** boxes relative to the selected alignment.

You must make sure that the minimum value of each axis is smaller than the maximum value of the corresponding axis.

5. Click **OK** to close the dialog box. If you want to define a ClearanceCube and continue to modify its definition, click **Apply**.

You cannot define a ClearanceCube from the STARTUP alignment. PC-DMIS does not display this alignment in the STARTUP alignment list.

Re-import a CAD in a Measurement Routine with ClearanceCube

When you re-import a CAD model into a measurement routine with a pre-defined ClearanceCube, you must consider these factors:

- **ClearanceCube is defined using CAD**

  When you re-import a CAD model into a measurement routine, PC-DMIS redefines the ClearanceCube automatically. PC-DMIS uses the same offset boundary for each side as defined in the original CAD model.

- **ClearanceCube is defined using Part Alignment**

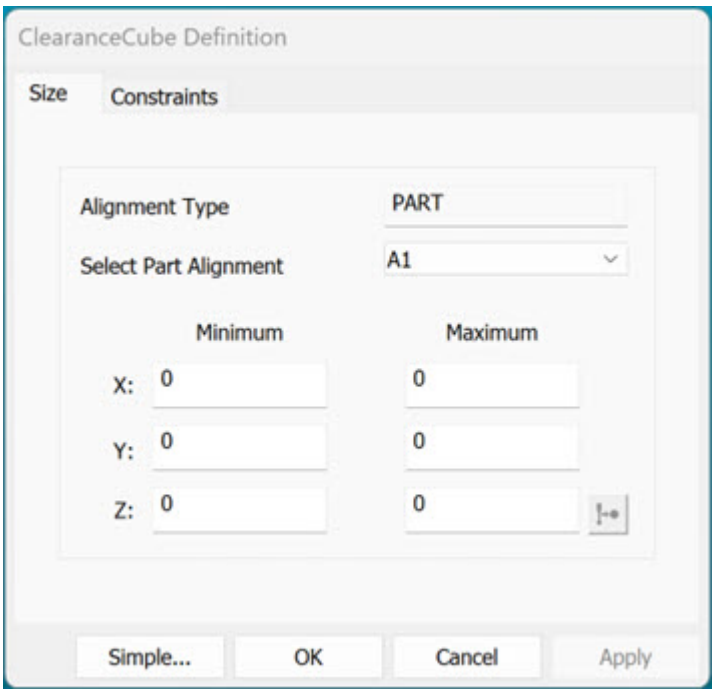
  PC-DMIS redefines the ClearanceCube by default with a 10 mm offset to the CAD model's boundary.

- **ClearanceCube was created using Part Alignment in previous versions**

  PC-DMIS 2023.2 and later versions do not allow you to define a ClearanceCube with part alignment if a CAD model is available. PC-DMIS redefines the ClearanceCube and displays a message to inform you of the new ClearanceCube settings.

ClearanceCube Definition - Size tab

To open the **Size** tab in the **ClearanceCube Definition** dialog box, click the **Advanced** button.



ClearanceCube Definition Dialog Box - Size Tab

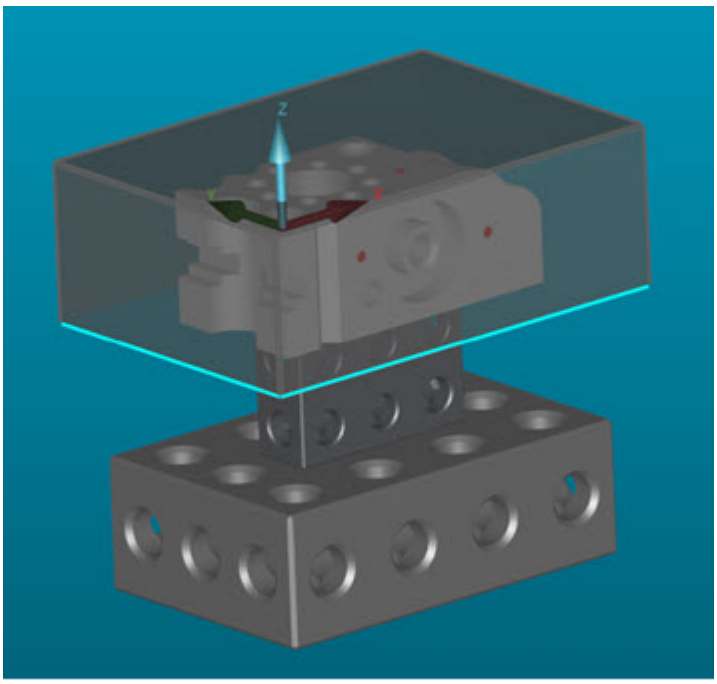
For a measurement routine with a CAD model, PC-DMIS defines the minimum and maximum boundary of the ClearanceCube in the CAD alignment.

For a measurement routine without a CAD model, PC-DMIS defines the minimum and maximum boundary of the ClearanceCube in one of the alignments created in the measurement routine.

You can modify the boundary in each axis as desired.

Defining the ClearanceCube to be Smaller than the CAD Boundary

You can define the ClearanceCube to be smaller than the CAD boundary when you have a CAD model that includes a large fixture.



Example showig a ClearanceCube smaller than the CAD boundary with a large fixture.

PC-DMIS displays a warning message when the ClearanceCube is smaller than the CAD boundary to confirm or cancel the ClearanceCube definition process.

Resize ClearanceCube using CAD

When you are in the **ClearanceCube Definition** dialog box:

- To resize ClearanceCube, drag any side of the ClearanceCube box.
- To rotate the CAD model, press Ctrl and right-click.

Teach Size of ClearanceCube

You can teach to define the size of the ClearanceCube:

Ensure that the probe tip you want to use is the active tip. If required, position your mouse pointer in the Edit window at the desired tip command.

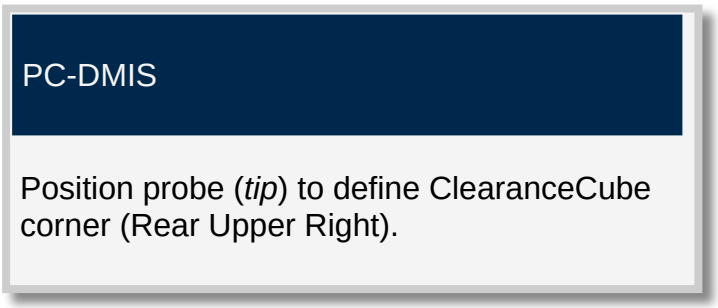
1. In the **ClearanceCube Definition** dialog box, from the **Maximum** area of the **Size** tab, click the **Teach** button. PC-DMIS displays this message:

   PC-DMIS

   Position probe (*tip*) to define ClearanceCube corner (Front Lower Left).

   where *tip* is the active probe tip at the cursor's current location in the Edit window.

2. Position the probe tip at the front lower left corner of the ClearanceCube and then click **OK** (or press the **Done** button on the jogbox). This message appears:

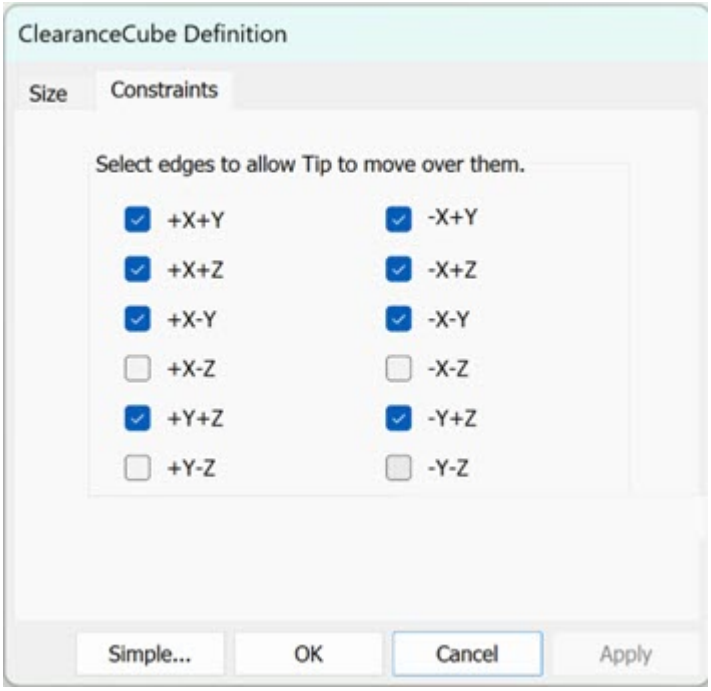


3. Position the probe tip at the rear upper right corner of the ClearanceCube and then click **OK** or press the **Done** button on the jogbox.

4. PC-DMIS automatically fills in the values in the **Minimum XYZ** and **Maximum XYZ** boxes and draws the ClearanceCube.

ClearanceCube Definition - Constraints tab

Use the **Constraints** tab to specify which of the 12 edges you want to allow the tip to move over. If you do not select an edge, PC-DMIS does not move the tip.



ClearanceCube Definition Dialog Box - Constraints Tab

If an edge is cleared, PC-DMIS does not move the tip over that edge. You can use constraints to restrict tip movement to avoid collision with the clamps.

The Graphic Display window displays those edges in blue on the ClearanceCube. If none of the four edges of a face are selected, PC-DMIS displays that face in bright yellow.

Clear Edges Using the CAD Model

You can select or deselect an edge that you want or not want to pass over using the CAD in the Graphic Display window. To do this, with the **ClearanceCube Definition** dialog box displayed, click an edge on the CAD in the Graphic Display window. You can see as you select or deselect the edge on the CAD model, PC-DMIS selects or deselects that same edge.

You can also rotate the CAD model with this dialog box open. To do this, press the Ctrl button and right-click and then move your mouse cursor to the desired rotation position.

ClearanceCube Settings for Feature or Moveset Command

You must set ClearanceCube motion for every feature in a measurement routine.

- PC-DMIS moves the tip to the starting face set for a feature before the measurement of the feature.
- PC-DMIS moves the tip to the ending face set for a feature after the measurement of the feature.

PC-DMIS does not execute ClearanceCube moves if it is set off for a feature.

ClearanceCube Settings for Tip Command

You can activate ClearanceCube motion for tip commands.

- If the tip change completes safely in front of the part face, set the ClearanceCube motion for the TIP command to the Off condition. You must ensure that the last position of the tip is safe for rotation or change.

If ClearanceCube motion is activated for the TIP command, PC-DMIS needs to move the tip to a safe location. The tip must be located on the face of or outside of the ClearanceCube.

This can be accomplished by either:

- Setting ClearanceCube motion to the ending face of the last feature

  Or,

- Adding a Moveset command and setting the ClearanceCube motion to the ending face

PC-DMIS displays the message during execution of a measurement routine if the ClearanceCube is activated for the TIP command and located inside the ClearanceCube.

ClearanceCube Settings for Scan Command

The Advance Scans are containers for BASICSCAN commands. You can set ClearanceCube motion for a scan and each BASICSCAN in a scan.

```
SCN1        =FEAT/SCAN,LINEAROPEN,NUMBER OF HITS=185,SHOW HITS=NO,SHOWALLPARAMS=NO
             MEAS/SCAN
               BASICSCAN/LINE,NUMBER OF HITS=50,SHOW HITS=NO,SHOWALLPARAMS=NO
               ENDSCAN
               BASICSCAN/LINE,NUMBER OF HITS=22,SHOW HITS=NO,SHOWALLPARAMS=NO
               ENDSCAN
               BASICSCAN/LINE,NUMBER OF HITS=23,SHOW HITS=NO,SHOWALLPARAMS=NO
               ENDSCAN
               BASICSCAN/LINE,NUMBER OF HITS=90,SHOW HITS=NO,SHOWALLPARAMS=NO
               ENDSCAN
             ENDMEAS/
```

Example of a BASICSCAN command in a scan.

PC-DMIS executes the scan and the Basic scan in this way:

PC-DMIS moves the ClearanceCube based on the starting face of the scan (SCN1).
PC-DMIS executes the Auto Move before scanning as defined in the scan command.

> PC-DMIS moves the ClearanceCube based on the starting face of the Basic scan (SCN1.1).
> PC-DMIS scans the Basic scan (SCN1.1).
> PC-DMIS moves the ClearanceCube based on the ending face of the Basic scan (SCN1.1).
>
> PC-DMIS repeats these Basic scan SCN1.1 steps for SCN1.2 through SCN1.n.

PC-DMIS executes the Auto Move after scanning as defined in the scan command (SCN1).
PC-DMIS moves the ClearanceCube based on the ending face of the scan (SCN1).

Tip Moves Using Constrained Edges

Tip has activated ClearanceCube Moves set to ON

When you define constraints for some feature edges, PC-DMIS does not allow the tip to cross those edges. PC-DMIS does allow the tip to move freely at the T1A0B0 rotation height. Because of this, you must use a shorter fixture and clamp configuration for your part than this height. In some cases, you may have to use a `MOVESET` command to move the tip to the desired location.

If all four edges of a ClearanceCube face are constrained, PC-DMIS shows the edges in a light blue color in the Graphic Display window. PC-DMIS does not allow you to use this face for ClearanceCube motion.

Tip has activated ClearanceCube Moves set to OFF

When the TIP command becomes active with ClearanceCube move set to OFF, you must ensure that the tip crosses the correct edges and only edges that are not constrained. You may have to use `MOVESET` commands to achieve the correct path for the tip's motion.

ClearanceCube Settings

You can use the ClearanceCube Settings window to define the ClearanceCube motion for each feature and tip in the measurement routine.

ClearanceCube Settings Window

PC-DMIS displays the **ClearanceCube Settings** window automatically when you activate ClearanceCube for the measurement routine.

From the **ClearanceCube** toolbar, you can use the **ClearanceCube Settings Window** icon to show or hide this window. You can also open this window from the **View | Other Windows | ClearanceCube Settings** menu option.

You can dock the ClearanceCube Settings window at the top or bottom of the Graphic Display window. To float the ClearanceCube Settings window, press the Ctrl button and then click and drag the window to the desired location.

- **ID(s)** - This list displays one or more feature IDs based on your selection of features in the Edit window. To do this, follow these steps:

  1. Place your cursor on a feature in Command Mode or select a feature in Summary Mode to display or edit the settings for the ClearanceCube.
  2. To edit the ClearanceCube settings for more than one feature at the same time, select them in the Edit window and then make the necessary changes.
  3. From the **ID(s)** list, select Features or Tips. This box appears when you select multiple features and tips in the Edit window.
  4. Based on your selection from the **ID(s)** list, the list shows the features or tips. Select the item in the list for configuration.

If you position your mouse cursor over a command in Command Mode or select a command in Summary Mode that does not require ClearanceCube settings, PC-DMIS displays the message “ClearanceCube is not required for the selected commands”.

- **Activate ClearanceCube Moves** - This switch determines whether you want to use ClearanceCube motion for the features you selected in the **ID(s)** list.

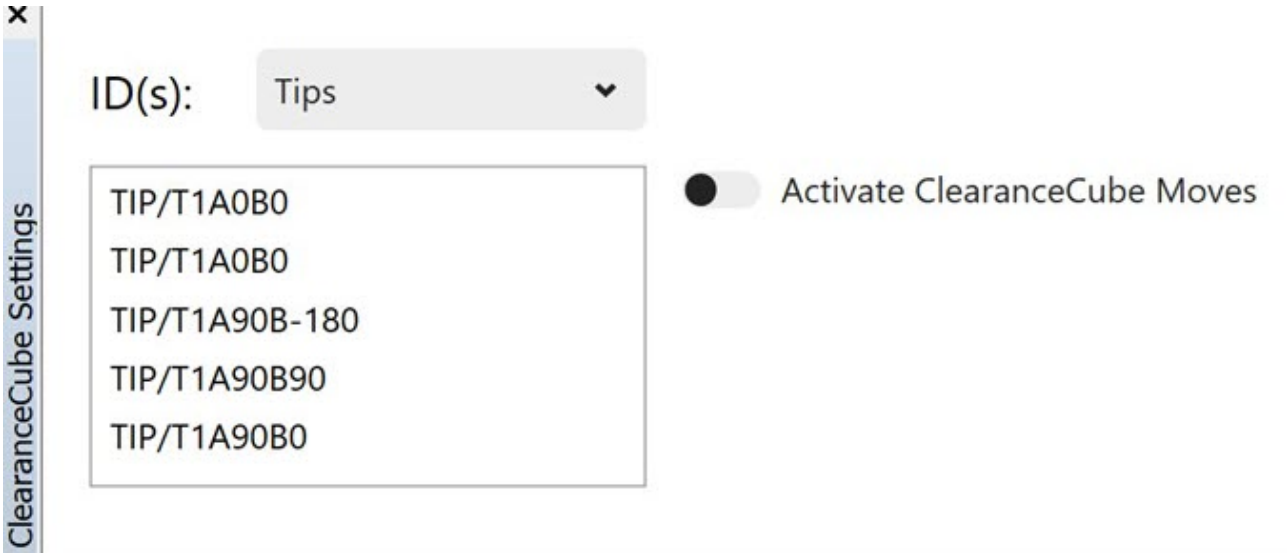
- **Starting Face** - This determines the ClearanceCube face that PC-DMIS moves to before it measures the feature. You have these options:
  - Select a face
  - Use Tip Vector
  - Set the Starting Face to OFF if you only want the ending motion to occur for the feature.
- **Ending Face** - This determines the ClearanceCube face that PC-DMIS moves to after it measures the feature. You have these options:
  - Select a face
  - Use Tip Vector
  - Set the Ending Face to OFF if you only want the starting motion to occur for the feature.

The feature vector determines the default setting for the ClearanceCube **Starting Face** and **Ending Face** options. PC-DMIS marks the **ClearanceCube uses Tip Vector for Start/End Face** check box in the **General** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) by default. It is recommended to use the tip vector.

Based on constraints set in the **Constraints** tab, the selection of some axes for the starting and ending faces may not be available.

For Tip Commands

From the **ID(s)** list, select the features or tips. This box appears only when you select multiple features and tips in the Edit window.



ClearanceCube Settings Window - Tips Option

When you select the **Tips** option from the **ID(s)** list, the **ClearanceCube Setting** window only shows the **Activate ClearanceCube Moves** setting.

The ClearanceCube Shortcut Menu

In addition to the ClearanceCube Settings dialog box, PC-DMIS provides a right-click menu. To access this menu, follow these steps:

1. If you are in Command Mode, place your cursor on a feature. If you are in Summary Mode, click a feature to select it.
2. Right-click the feature and select the **Path | ClearanceCube** option.
3. From the **Starting Face** and **Ending Face** list, select the face to align the ClearanceCube to. The available options are -X, +X, -Y, +Y, -Z, +Z, Use Tip Vector, and Off.

ClearanceCube Toolbar

ClearanceCube Toolbar

Use the **ClearanceCube** toolbar (**View | Toolbars | ClearanceCube**) to enable or disable options related to ClearanceCube functionality. See "Using the ClearanceCube" in the "Editing the CAD Display" chapter.

The following icons are available:

ClearanceCube Definition - This icon displays the **ClearanceCube Definition** dialog box. For more details on the dialog box, see the "(Simple) ClearanceCube Definition" topic. You can also press Alt + C to open the dialog box.

Activate ClearanceCube Motion - This icon enables or disables ClearanceCube motion.

- If enabled, your measurement routine uses the ClearanceCube to control its motion and *disables clearance planes*.
- If disabled, your measurement routine uses existing clearance plane commands to control its motion.

You cannot deactivate ClearanceCube if any feature in a measurement routine uses ClearanceCube motion.

Show ClearanceCube - This icon shows or hides ClearanceCube in the Graphic Display window.

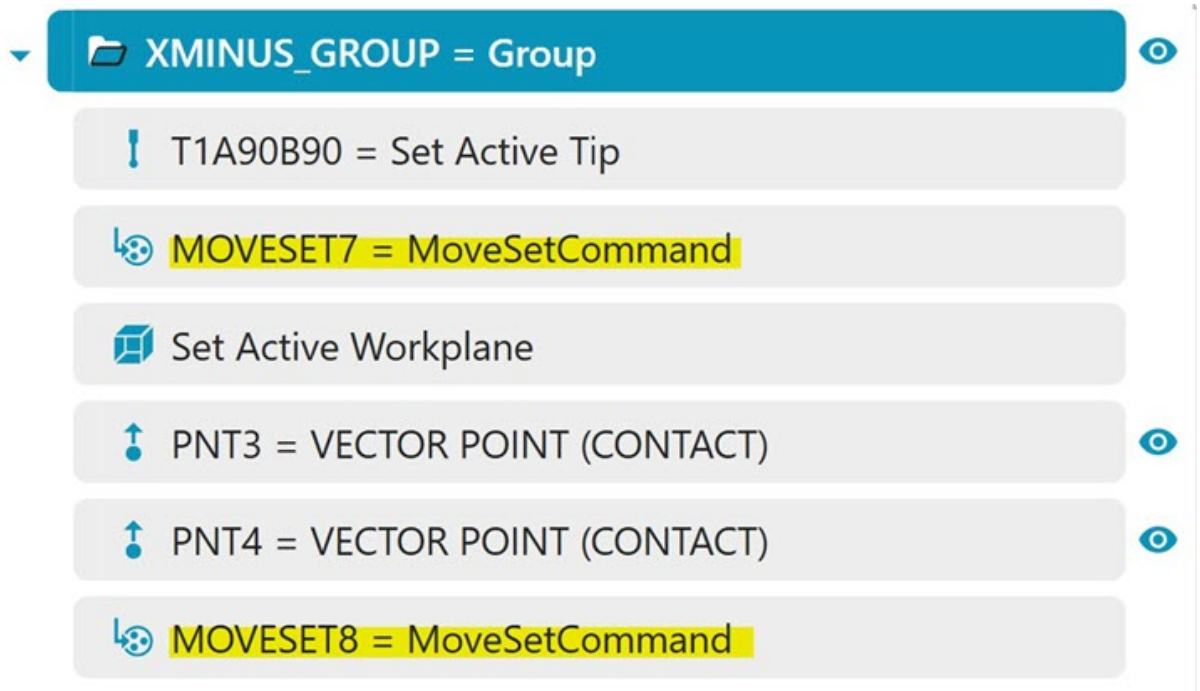
To increase transparency through ClearanceCube in the Graphic Display window, select the **High quality transparency** check box in the **OpenGL** tab of the **CAD and Graphic Setup** (**Edit | Graphic Display Window | OpenGL**) dialog box.

ClearanceCube Settings window - This icon shows or hides ClearanceCube Settings window.

Important Notes

PC-DMIS provides the ClearanceCube moves along with these additional commands to help you safely navigate around your part during execution:

- Avoidance Move - Use Avoidance Move commands with Auto features to safely move to the next feature in your measurement routine. You can select the Avoidance Move command along the feature or tip vector direction.
- Moveset command - The Moveset command is a set of move points. You must select the ClearanceCube for the Moveset command. Use this command to move the tip so that it is in front of the surface that is inside a complex geometry. You can also use this command at the beginning of a group and at the end of a group to bring the tip to the front of a part face where features are located.



Important Information When Using ClearanceCube

- PC-DMIS supports using ClearanceCube only on single arm systems. It *does not* support ClearanceCube on Dual Arm systems.
- When you activate ClearanceCube, PC-DMIS disables any Clearance Plane commands in the current measurement routine. You can only use Clearance Plane or ClearanceCube in a measurement routine.
- Be sure to check for collisions before you execute your measurement routine. For more information, see "Detecting Collisions".
- Make sure you place the probe tip outside the ClearanceCube before you start the execution of your measurement routine.
- You can start execution or partial execution or block execution with the tip located inside the ClearanceCube if required. This can happen when execution stops due to a chip in a bore. You can clear the error and then continue execution of the measurement routine from where it stopped.
- The width of ClearanceCube in any direction cannot be smaller than 1 mm. This helps you to ensure the correct definition of the ClearanceCube.
- You must ensure the safe execution of a measurement routine, desired ClearanceCube motion settings, and scan command when you open a measurement routine created in a version before PC-DMIS 2023.2.

Inserting Clearance Moves Automatically

Once you import or create features and perform a path optimization, you need to ensure that your probe won't collide with your part during measurement. PC-DMIS contains menu options in the **Operation | Graphic Display Window | Clearance Moves** submenu that insert clearance moves automatically as `MOVE/POINT` commands to help avoid unintentional collisions with the part. These menu items under the **Clearance Moves** submenu are discussed below.

For more information on how to detect collisions, see "Detecting Collisions" in the "Editing the CAD Display" chapter.

More:

Clearance Moves Menu Items

Using Auto Insert Moves

Related Topics:

Inserting Move Commands

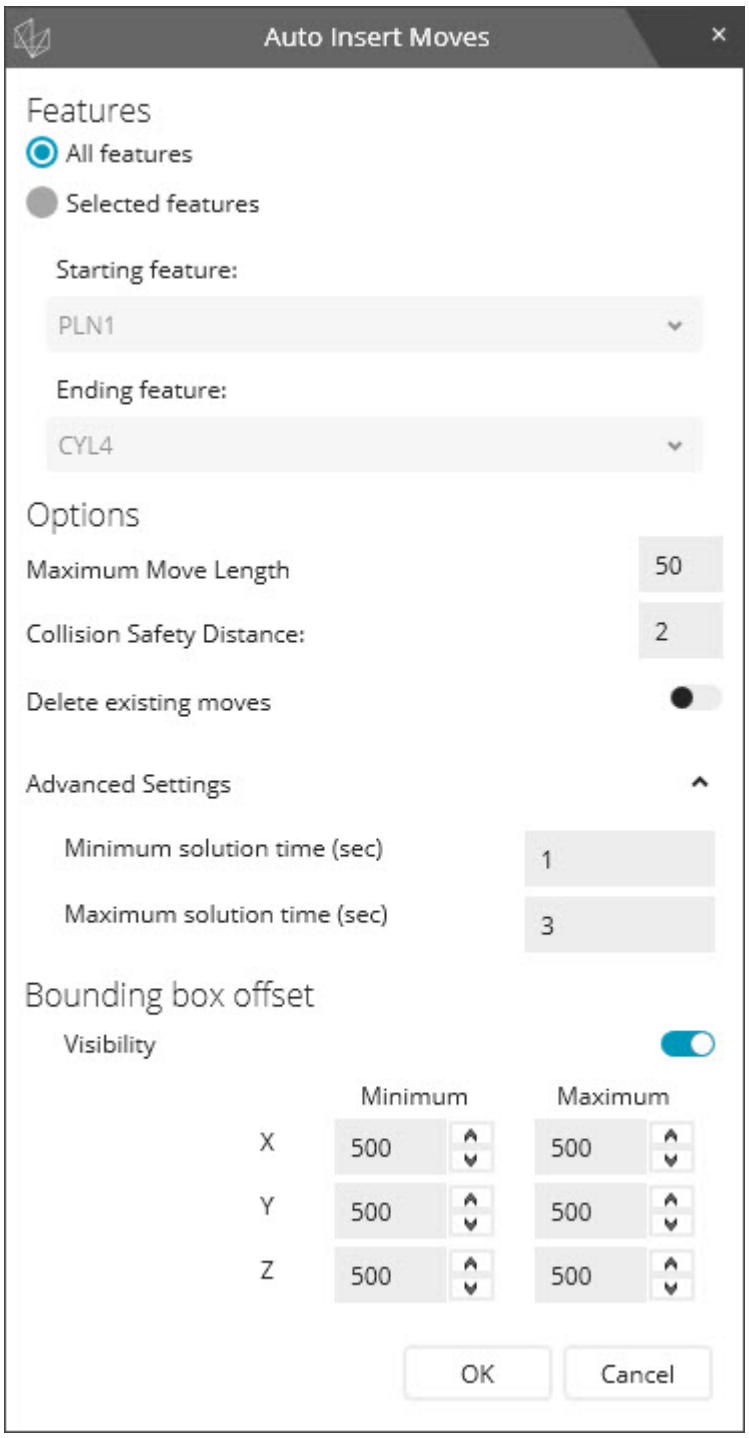
Detecting Collisions

Clearance Moves Menu Items

These are the menu items under the **Operation | Graphic Display Window | Clearance Moves** submenu:

Auto Insert Moves

The **Auto Insert Moves** menu option (**Operation | Graphic Display Window | Clearance Moves | Auto Insert Moves**) opens the **Auto Insert Moves** dialog box. You can use this dialog box to add clearance moves (`MOVE/POINT` commands) between all features or between selected features. For information on how to do this, see the "Using Auto Insert Moves" topic below.



Auto Insert Moves Dialog Box

When you click **OK**, PC-DMIS inserts the `MOVE/POINT` commands.

PC-DMIS does not place move commands prior to the first feature; make sure there is a clear path between the probe's start position and the first feature.

All features - PC-DMIS places the move commands between all features.

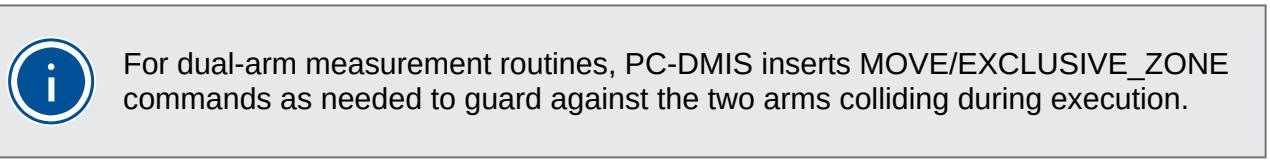
Selected features - PC-DMIS places the move commands between a range of two features that you can select from this area. The **Starting feature** and **Ending feature** lists beneath this option show the features. (You can also select the two features from the Edit window before you open this dialog box.)

Maximum Move Length - This value defines the maximum length of a move. This value is in mm or inches, based on the measurement routine's units. This means any `MOVE/POINT` commands that the algorithm calculates to move the probe from one location to another cannot exceed this value.

PC-DMIS attempts to go directly to the next feature, but compensates for collisions by inserting clearance move commands. It continues this process until it defines a collision free path to the next feature. More than one move command may be inserted between features. The higher the value, the lower the amount of time PC-DMIS spends calculating these moves. The recommended value is 100 mm (3.93 inches).

Collision Safety Distance - This value defines a safety distance around the part. A move is considered a safe move if it moves the probe this distance away from the part. To prevent the creation of too many moves, the value of **Maximum Move Length** should be greater than the collision safety distance value.

Delete existing moves - This switch determines whether to delete any pre-existing move commands in the measurement routine.



Advanced Settings

Minimum Solution time (sec) - This value is the minimum number of seconds that the algorithm uses to plan a move between two feature commands. A higher value increases the accuracy of the move, but it also increases the computation time. The recommended value is 1 second.

Maximum solution time (sec) - This value is the maximum number of seconds that the algorithm uses to plan a move between two feature commands. A higher value increases the accuracy of the move, but it also increases the computation time. The recommended value is 3 seconds.

Bounding box offset - This area lets you define a cubic region or cubic-like rectangular region that PC-DMIS uses to calculate the automatic insert moves. *This region is not a clearance cube.* You may find this functionality useful if you have a fixture or some other obstruction around certain regions of your part and you don't want PC-DMIS to calculate moves near those areas. In that case, you can define a region, and PC-DMIS calculates the moves only within this region.

The maximum and minimum ranges for each axis let you define a more precise space for PC-DMIS to calculate the moves.

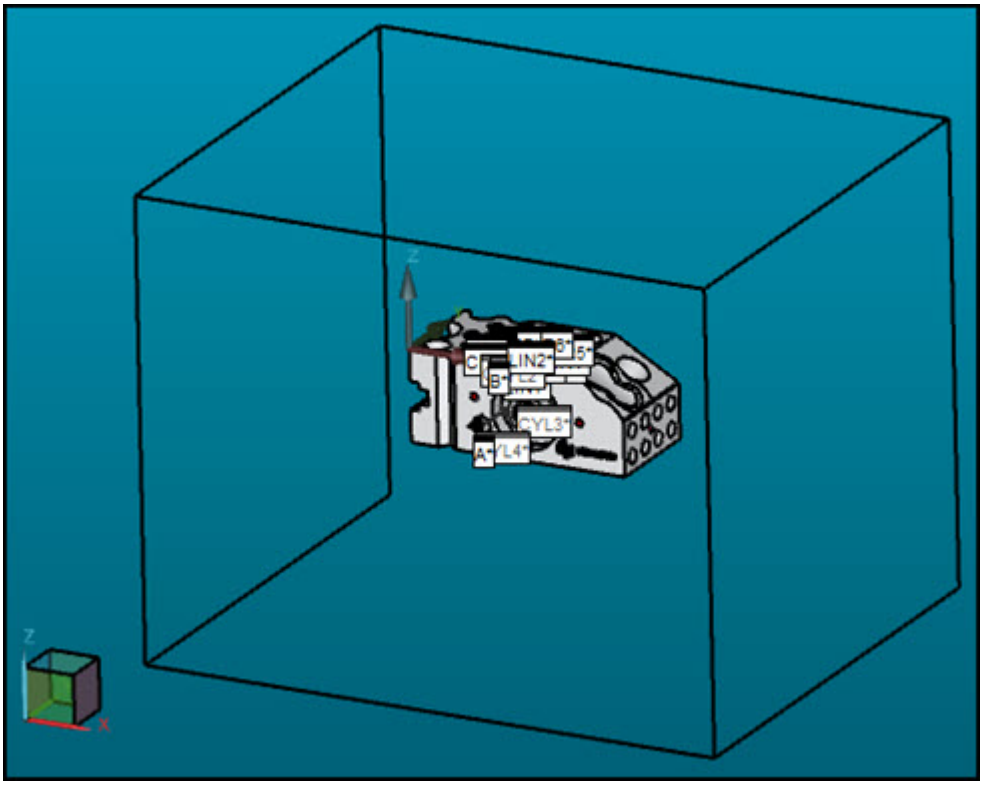
The offset values are based from on the surface sides, not from the alignment origin. This value is in mm or inches, based on the measurement routine's units.

The software adds this value to the CAD model's existing bounding box offsets. The recommended value is 1000 mm (39.3 inches).

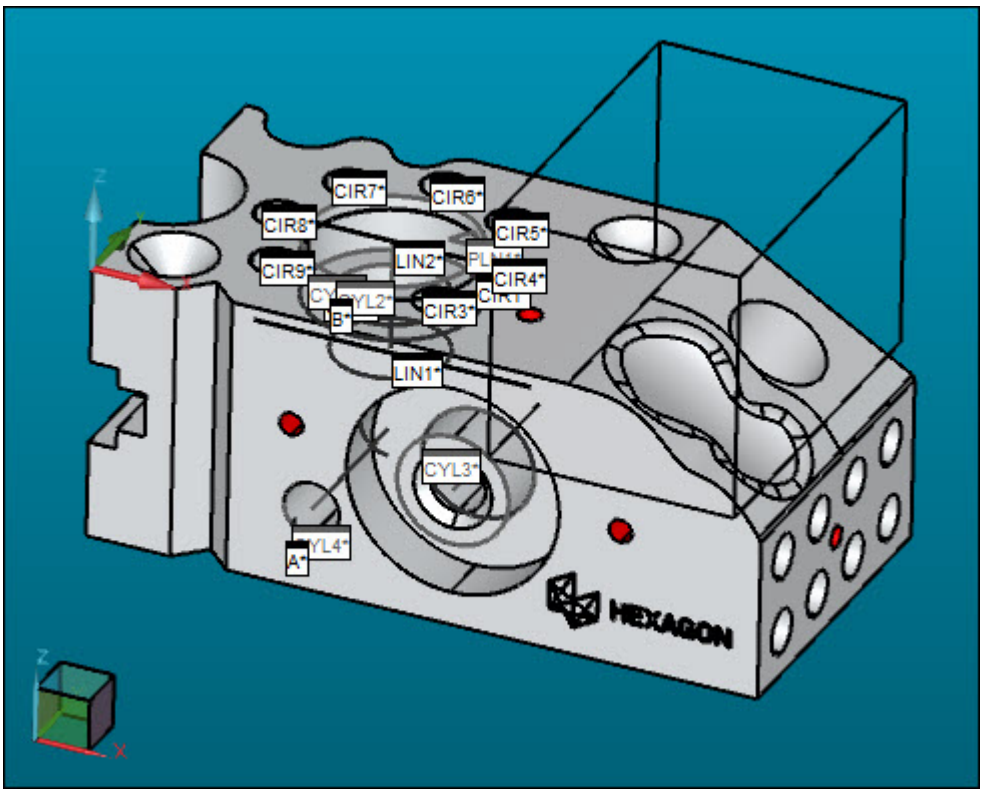
The small up and down arrows in each box adjust the value by 10 percent of the maximum size of the bounding box with each click.

Visibility - This switch lets you show or hide the bounding box region in the Graphic Display window while the dialog box remains open.

The image below shows the bounding box at 100 mm in all axes:



The image below shows the bounding box with minimum X, Y, and Z values of -80, 20, and -45 and maximum X, Y, and Z values of 4.0, -20, and 40:



Within Feature

This menu item (**Operation | Graphic Display Window | Clearance Moves | Within Feature**) only works in QuickFeatures mode. If you select this menu item from the check box and you create a QuickFeature, PC-DMIS checks the probe path inside the feature and adds clearance moves as needed *within* the feature. This is different from the **With Feature Creation** menu item below, which adds clearance moves *between* features.

With Feature Creation

You can choose **With Feature Creation** menu item (**Operation | Graphic Display Window | Clearance Moves | With Feature Creation**) to enable it or disable it. The menu item shows a check mark when enabled.

With this menu item enabled, PC-DMIS uses a simple algorithm to automatically insert `MOVE/POINT` commands as you add features to the measurement routine. The angle between the features must be within a range of 30 to 150 degrees. The algorithm cannot solve anything outside this range. For example, two circle features at different depths but on the same face, results in a 0 degree angle between them and can't be solved. When you insert move commands, this menu item behaves similarly to the **Auto Insert Moves** dialog box and uses one feature above and below the newly inserted feature as the start and end features. This menu item also uses the distance defined in the `CollisionMoveClearanceInMM` entry.

The QuickFeatures menu item activates a new algorithm to generate safe clearance moves as you add QuickFeatures. Automatic safe moves between features that use different probe tip angles are not yet supported. You must manually define those moves.

This functions only in Offline mode.

With Collision Detection

The **With Collision Detection** menu item is not supported for QuickFeatures.

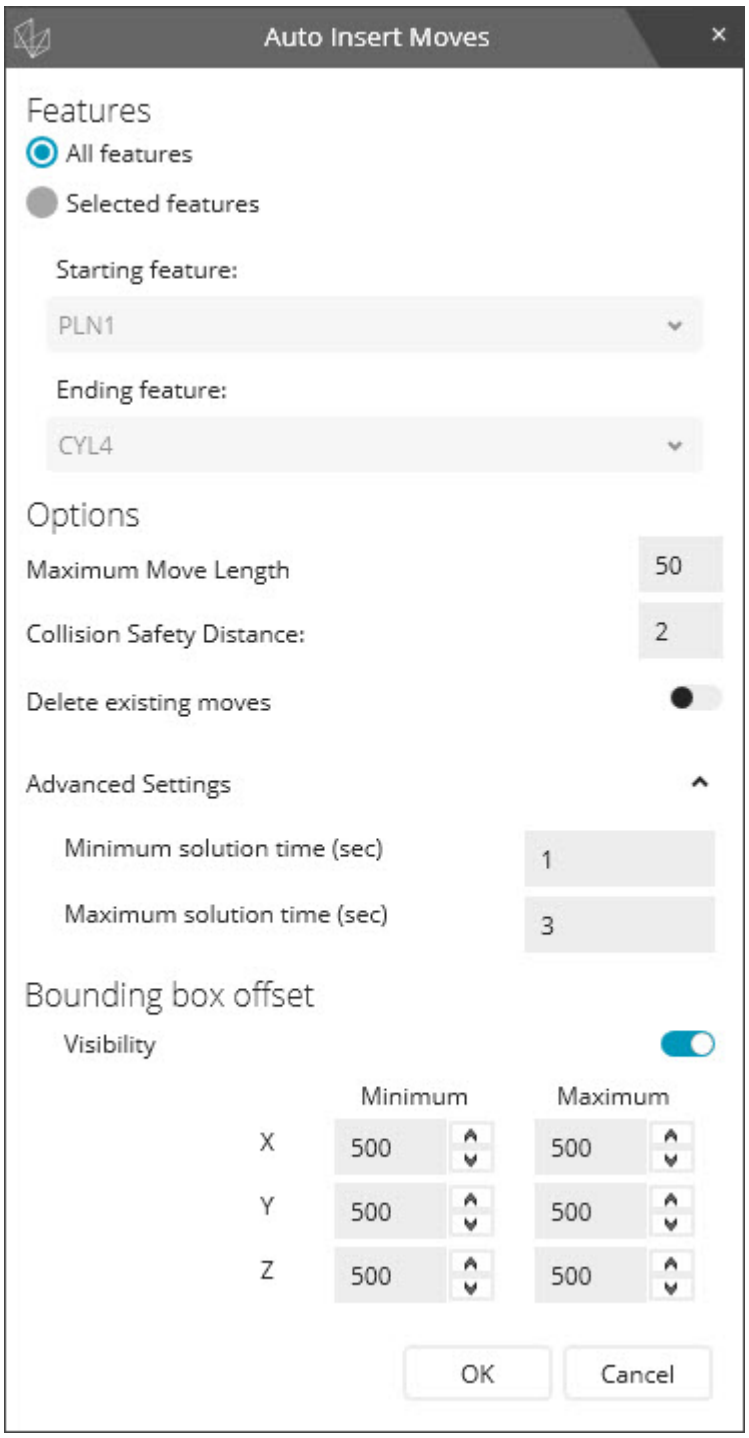
You must select the **With Feature Creation** menu item (**Operation | Graphic Display Window | Clearance Moves | With Feature Creation**) to enable the **With Collision Detection** option. When enabled, this option shows a check mark.

If you enable the **With Collision Detection** menu item, the software extends the functionality of **With Feature Creation** so that you aren't limited by the angle restrictions between features. PC-DMIS uses the collision detection algorithm to detect collisions instead of the simple algorithm. It provides clearance moves between the range of features. For each collision found, it then inserts the appropriate clearance moves.

Using Auto Insert Moves

This procedure helps you insert safe clearance moves for your probe between all features or some features in your measurement routine. This procedure uses information discussed in the "Auto Insert Moves" subtopic in the "Clearance Moves Menu Items" topic above. Refer to that subtopic if you need more information on anything in this procedure.

1. Choose **Operation | Graphic Display Window | Clearance Moves | Auto Insert Moves**. The software opens the **Auto Insert Moves** dialog box.



Auto Insert Moves Dialog Box

2. From the **Auto Insert Moves** dialog box, define whether to insert moves between **All features** or between a range of **Selected features**.

3. If you choose **Selected features**, define a range between two features. Use the **Starting feature** and **Ending feature** lists to choose the two features.

4. If you want a different maximum distance value for your moves, you can modify the default **Maximum Move Length** value. The higher the value, the lower the amount of time PC-DMIS spends calculating these moves. To prevent the creation of too many moves, this value should be greater than the **Collision Safety Distance** value.

5. If you want a different safety distance, you can modify the default **Collision Safety Distance** value.

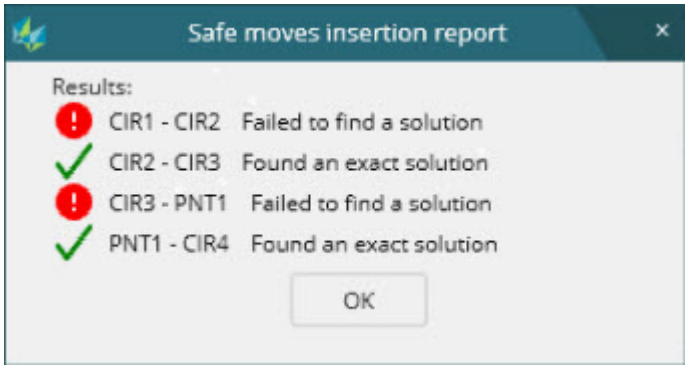
6. If you want to delete all existing moves in your measurement routine before this routine inserts new move commands, move the **Delete existing moves** switch to on.

7. If you need to make changes to the **Advanced Settings**, expand **Advanced Settings**, and make your changes.

8. Click **OK** to start the move insertion process. The software's algorithm calculates and generates safe moves between the features. Once it finishes, it shows a **Safe Moves Insertion Report** dialog box.

   You can press ESC at any time during the generation process to cancel the process and remove any moves the software inserted.

9. From the **Safe Moves Insertion Report** dialog box, hover your pointer over the items to review the results. Then click **OK** to close the dialog box.



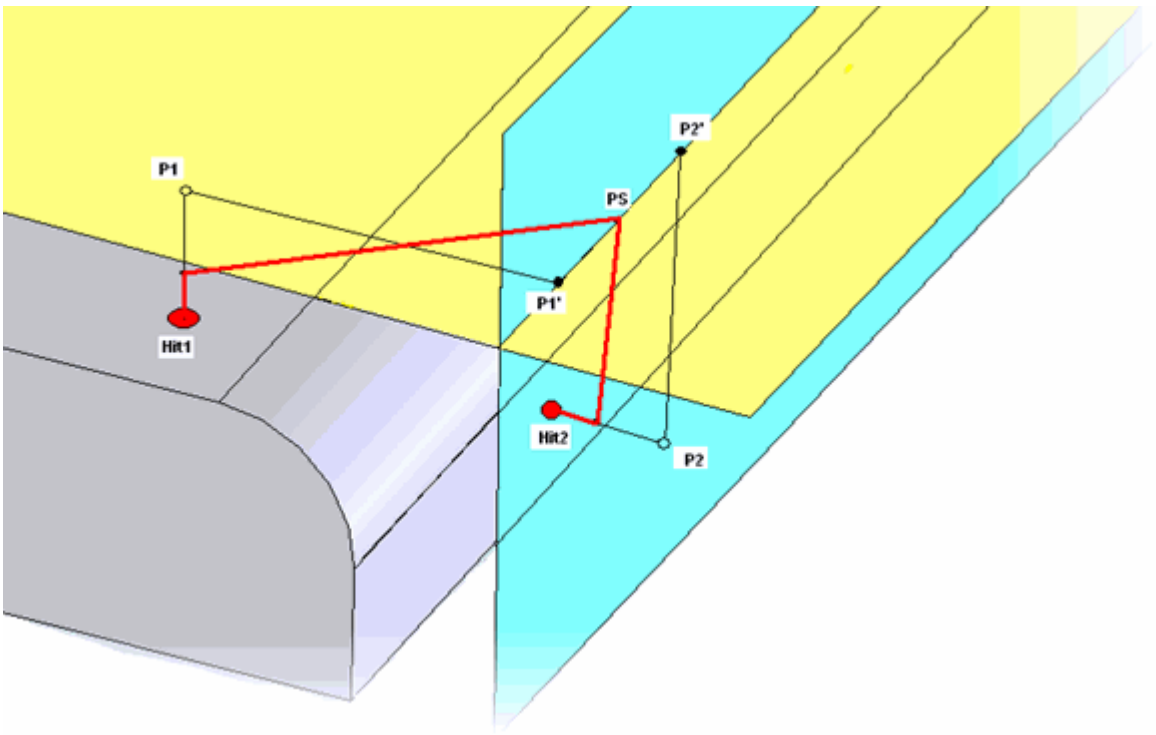
Safe moves insertion report Dialog Box

If the **Safe Moves Insertion Report** failed to find a solution between features, you may need to increase the **Bounding box offset size** under **Advanced Settings**. You may also need to increase the solution times. You can do that by increasing the **Minimum solution time (sec)** and **Maximum solution time (sec)** values.

If the results say you have an invalid start or target command position, this may be due to an incorrect probe tip assigned to measure those commands.

Diagram of MOVE/POINT Calculations

This diagram shows where PC-DMIS calculates a `MOVE/POINT` command between two hits:



- **Hit1** - The last hit
- **P1** - Offset point from Hit1 in vector direction
- **Hit2** - The new hit
- **P2** - Offset point from Hit2 in vector direction. This is the created MOVE/POINT.
- **Yellow Plane** - The plane through P1
- **Blue Plane** - The plane through P2
- **P1'** - Projected P1 in the vector direction of P2
- **P2'** - Projected P2 in the vector direction of P1
- **PS** - Symmetry Point between P1' and P2': This is the new move point.

If the angle between the vector of Hit1 and the vector of Hit2 is bigger than 150°, no move point is needed.

Branching by Using Flow Control

Branching by Using Flow Control: Introduction

Suppose you have a part with many features, but you just want to measure a few features over and over to get a comprehensive statistical set of data for those features. Suppose you want to jump to a particular part in your measurement routine dependent on a response from the user. You can accomplish tasks such as these, and many others, by using flow control commands. By setting up conditions for certain commands, you can control the flow of your measurement routine.

This chapter provides you with the information you need to accomplish such tasks. It explains the syntax for conditional statements, loops, and subroutines. It also provides many code samples.

The main topics covered in this chapter include the following:

- Using Control Pairs
- Using Generic Loops
- Using Labels
- Branching on an Error
- Branching with Subroutines
- Ending a Measurement Routine

Command Mode Commands after Comments

Because many of the code samples in this chapter use typed `COMMENT` commands, consider the following:

After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

Using Control Pairs

The **Insert | Flow Control Command | Control Pairs** submenu offers various paired commands that work within the Edit window to govern or "control" the proper flow of the measurement routine. To insert a control pair type command into the Edit window, simply type the command, or choose a command from this submenu.

When you use a conditional branching statement to test for a the value of a YES / NO comment, be aware that your test should look for an uppercase "YES" or "NO" value. A lowercase "Yes" or "No" will not work. For information on comments, see the "Inserting Programmer Comments" topic in the "Inserting Report Commands" chapter.

If / End If

The **Insert | Flow Control Command | Control Pairs | If / End If** menu option allows you to add a conditional block to the measurement routine. The items between the IF and the END IF commands will only execute if the expression for the IF command evaluates to true (nonzero). Otherwise, flow of execution will jump to the first command after the `END/IF` command.

The Edit window command line for an IF / END IF statement reads:

```
IF/expression

END_IF/
```

To insert the If / End If commands, follow these steps:

1. Place the cursor in the desired location of the Edit window.
2. Select **If / End If** from the menu bar. The IF / END IF statement will appear in the Edit window.

Code Sample of If / End If

Consider the following example that asks the user if he or she would like to measure a point feature.

```
C1= COMMENT/YESNO,Would you like to measure the point
feature, PNT1?
IF/C1.INPUT=="YES"
   PNT1=FEAT/POINT,RECT
   …
   …
   ENDMEAS/
END_IF/
```

Explanation of Sample Code

C1=COMMENT/YESNO

This line takes and stores the YES or NO response from the user.

IF/C1.INPUT=="YES"

This line is the expression. It tests to see if the input of comment 1 is a YES. If it's a YES then the IF statement is TRUE and continues executing the statements after the IF statement, in this case it measures the `PNT1` feature. If NO it moves to the END_IF statement.

END_IF

This line ends the execution of commands inside the IF / END IF block of code. Any command following this line is where PC-DMIS will go to if the user clicks **No** at the comment.

Else If / End Else If

The **Insert | Flow Control Command | Control Pairs | Else If / End Else If** menu option allows you to add a conditional block to the measurement routine. The items between the ELSE IF and the END ELSE IF commands will only execute if the expression for the ELSE IF command evaluates to true (nonzero). The ELSE IF / END ELSE IF block must be positioned directly *after* an IF / END IF block or another ELSE IF / END ELSE IF block. If all IF / ELSE IF expressions above the current block have evaluated to false, then the expression will be evaluated. If the expression evaluates to false (zero), then execution will jump to the next command following the END ELSE IF command. If any of the IF / ELSE if expressions above the current block evaluate to true, all subsequent ELSE IF / END ELSE IF blocks in this sequence will be skipped.

The Edit window command line for a ELSE IF / END ELSE IF statement reads:

```
ELSE_IF/expression

END_ELSE_IF/
```

To insert the ELSE IF / END ELSE IF commands:

1. Place the cursor in the desired location of the Edit window, after an existing IF/END IF statement or ELSE IF/END ELSE IF statement.
2. Select **Else If / End Else If** from the menu bar. The ELSE IF / END ELSE IF statement will appear in the Edit window.

This type of block is only valid when positioned *after* an IF / END IF or ELSE IF / END ELSE IF block. Invalidly positioned control pairs are shown in red text in the Edit window.

Code Sample of Else If / End Else If

Consider the following example that displays a message notifying the user when any one of the X, Y, or Z values for a measured point exceeds defined tolerances:

```
PNT2=FEAT/POINT,RECT
…
…
ENDMEAS/
IF/PNT2.X<6.9 OR PNT2.X>7.1
   COMMENT/OPER,"The measured X value of PNT2: " + PNT2.X +
" is out of tolerance."
END_IF/
ELSE_IF/PNT2.Y<3.3 OR PNT2.Y>3.5
   COMMENT/OPER,"The measured Y value for PNT2: " + PNT2.Y +
" is out of tolerance."
END_ELSEIF/
ELSE_IF/PNT2.Z<.9 OR PNT2.Z>1.1
   COMMENT/OPER,"The measured Z value for PNT2: " + PNT2.Z +
" is out of tolerance."
END_ELSEIF/
```

Explanation of Sample Code

This code first tests the X value of the point. If the condition evaluates to false, then the code tests for the Y value. If the condition for the Y value evaluates to false, then it tests for the Z value.

If any of these conditions evaluates to true, PC-DMIS displays the comment associated with it and skips the remaining conditional statements.

```
IF/PNT2.X<6.9 or="" pnt2.x="">7.1
```

This line is the expression. It tests to see if the measured X value is less than 6.9 or greater than 7.1. If it exceeds either of these boundaries it executes the first comment.

```
END_IF
```

This line ends the execution of commands inside the IF / END IF block of code. Any command following this line is where PC-DMIS will go to if the IF THEN condition evaluates to false.

```
ELSE_IF/PNT2.Y<3.3 or="" pnt="" 2.y="">3.5
```

This line is the expression for the first ELSE_IF command. It only gets executed if the IF / END IF block above it returns false. It tests to see if the measured Y value is less than 3.3 or greater than 3.5. If it exceeds either of these boundaries it executes the second comment.

```
END_ELSEIF/
```

This line ends the execution of commands inside the first ELSE IF / END ELSE IF block of code.

```
ELSE_IF/PNT2.Z<.9 or="" pnt2.z="">1.1
```

This line is the expression for the second ELSE IF command. It only gets executed if the ELSE IF / END ELSE IF block above it returns false. It tests to see if the measured Z value is less than .9 or greater than 1.1. If it exceeds either of these boundaries it executes the third comment.

```
END_ELSEIF/
```

This line ends the execution of commands inside the second ELSE IF / END ELSE IF block of code.

Else / End Else

The **Insert | Flow Control Command | Control Pairs | Else / End Else** menu option allows you to add a conditional block to the measurement routine. The items between the ELSE and the END ELSE commands will execute only if all other if / end if and else if / end else if blocks above the else block have failed (All evaluated to zero). ELSE / END ELSE blocks must be located at the end of a set of IF / END IF or ELSE IF / END ELSE IF blocks in order to be valid.

The Edit window command line for a ELSE / END ELSE statement reads:

```
ELSE/

END_ELSE/
```

To insert Else / End Else commands:

1. Place the cursor in the desired location of the *Edit window*. Note that Else / END ELSE blocks must be positioned after an IF / END IF or ELSE IF / END ELSE IF blocks.
2. Select **Else / End Else** from the menu bar. The ELSE / END ELSE statement will appear in the Edit window.

Code Sample of Else / End Else

Consider the following example that asks the user if he or she would like to measure a point feature.

```
C1= COMMENT/YESNO,Would you like to measure the point
feature, PNT1? Clicking No measures the next feature.
IF/C1.INPUT=="YES"
   PNT1=FEAT/POINT,RECT
   …
   …
   ENDMEAS/
END_IF/
ELSE
   PNT2=FEAT/POINT,RECT
   …
   …
   ENDMEAS/
END_ELSE
```

Explanation of Sample Code

C1=COMMENT/YESNO

This line takes and stores the YES or NO response from the user.

IF/C1.INPUT=="YES"

This line is the expression. It tests to see if the input of comment 1 is a YES. If it's a YES then the IF statement is TRUE and continues executing the statements after the IF statement, in this case it measures the `PNT1` feature. If NO it moves to the END_IF statement.

END_IF

This line ends the execution of commands inside the IF / END IF block of code. Any command following this line is where PC-DMIS will go to if the user clicks **No** at the comment.

ELSE

If the above IF / END IF block evaluates to false then command lines falling after this line and before the END_ELSE line will be executed. In this case, PNT2 gets executed.

END_ELSE

This line ends the execution of commands inside the ELSE / END_ELSE block of code.

While / End While

The **Insert | Flow Control Command | Control Pairs | While / End While** menu option allows you to add a conditional loop to the measurement routine. The items between the WHILE and the END WHILE command will continue to execute in a loop until the condition (or expression) keeping the loop activated is no longer met, meaning the expression for the while loop evaluates to FALSE (i.e. zero). The WHILE command can be added anywhere in the measurement routine. The expression is tested at the start of each loop.

The Edit window command line for a WHILE / END WHILE statement reads:

```
WHILE/expression

END_WHILE/
```

To insert a While / End While option, follow these steps:

1. Place the cursor in the desired location of the Edit window.
2. Select **While / End While** from the menu bar. The WHILE / END WHILE statement appears in the Edit window.

Code Sample of While / End While

Consider the following example that measures a feature an amount specified by the measurement routine user.

```
C1=COMMENT/INPUT,How many times would you like to measure
PNT1? Please type an integer only.
ASSIGN/COUNT=0
WHILE/COUNT<C1.INPUT
   PNT2=FEAT/POINT,RECT
   …
   …
   …
   ENDMEAS/
   ASSIGN/COUNT=COUNT+1
   COMMENT/OPER,"Measured "+COUNT+" out of "+C1.INPUT+"
times."
END_WHILE/
```

Explanation of Sample Code

C1=COMMENT/INPUT

This line takes and stores the integer input from the user into the variable `C1.INPUT` .

`ASSIGN/COUNT=0`
This line initializes `COUNT` , a user-defined variable, and gives it an initial value of 0. The code uses this variable to count the number of times PC-DMIS measures the feature inside the loop.

`WHILE/COUNT<C1.INPUT`
This line is the expression. It tests to if the value of `COUNT` (initially set to 0) is less than the integer selected by the user. If this tests true, then the statements in following `WHILE/` and before `END_WHILE/` are executed

`ASSIGN/COUNT=COUNT+1`
This line increments the `COUNT` variable by one so that it eventually exits the loop when it fails the condition test.

`COMMENT/OPER,"Measured "+COUNT+" out of "+C1.INPUT+" times."`
This line displays a message showing the number of times, out of the total, that the loop is running.

`END_WHILE`
This line ends the execution of commands inside the `WHILE / END WHILE` block as long as the condition is false. Otherwise, when PC-DMIS encounters this command it loops back to the `WHILE` statement.

Do / Until

The **Insert | Flow Control Command | Control Pairs | Do / Until** menu option allows you to add a conditional loop to the measurement routine. The items between the DO and the UNTIL commands will continue to execute in a loop until the expression of the UNTIL command evaluates to TRUE (nonzero). The DO/ UNTIL commands can be added anywhere in the measurement routine. The expression is tested at the end of each loop.

The Edit window command line for a DO / UNTIL statement reads:

```
DO/

UNTIL/ expression
```

To insert DO / UNTIL commands:

1. Place the cursor in the desired location of the Edit window.
2. Select **Do / Until** from the menu bar. The DO / UNTIL statements will appear in the Edit window.

Code Sample of Do / Until

Consider the following example that measures a feature an amount specified by the measurement routine user. This is similar to the example given under the While / End While topic, except that PC-DMIS tests for the condition at the end of the loop instead of at the beginning.

```
C1=    COMMENT/INPUT,Type the number of times PC-DMIS should
measure the PNT1 feature:(type an integer only)
ASSIGN/COUNT=0 DO/
   PNT1=FEAT/POINT,RECT
   …
   …
   ENDMEAS/
   ASSIGN/COUNT=COUNT+1
   COMMENT/OPER,"Measured "+COUNT+" out of "+C1.INPUT+"
times."
UNTIL/COUNT==C1.INPUT
```

Explanation of Sample Code

C1=COMMENT/INPUT

This line takes and stores the integer input from the user into the variable `C1.INPUT` .

ASSIGN/COUNT=0

This line initializes `COUNT` , a user-defined variable, and gives it an initial value of 0. The code uses this variable to count the number of times PC-DMIS measures the feature inside the loop.

DO/

Begins the `DO / UNTIL` loop. All statements are executed at least once and measurement routine flow exits out of the loop once the expression evaluates to false.

ASSIGN/COUNT=COUNT + 1

This line increments the `COUNT` variable by one so that it eventually exits the loop when it fails the condition test.

COMMENT/OPER,"Measured " + COUNT + " out of " + C1.INPUT + " times."

This line displays a message showing the number of times, out of the total, that the loop is running.

UNTIL/COUNT == C1.INPUT

This line ends the execution of commands inside the `DO / UNTIL` loop as long as the condition evaluates to false. Otherwise, when PC-DMIS encounters this command it loops back to the `DO` statement.

Select / End Select

The **Insert | Flow Control Command | Control Pairs | Select / End Select** menu option allow for the addition of a conditional block that is used in conjunction with the CASE / END CASE and Default Case / End Default Case pairs. The expression for the Select command provides data that is compared against the expression in the Case statements. If the two expressions evaluate to the same thing, then the statements within the Case / End Case Block will execute. The SELECT / END SELECT block surrounds the sets of CASE / END CASE and DEFAULT CASE / END DEFAULT CASE blocks.

The Edit window command line for a SELECT / END SELECT statement reads:

```
SELECT/expression

END_SELECT/
```

To insert the Select / End Select commands:

1. Place the cursor in the desired location of the Edit window.
2. Choose **Select / End Select** from the menu bar. The SELECT / END SELECT statements will appear in the Edit window.

Code Sample of Select / End Select

The pairs, `SELECT / END_SELECT`, `CASE / END_CASE`, `DEFAULT CASE / END_DEFAULT CASE`, all work together to evaluate multiple conditions providing a wide range of alternatives.

Suppose you have five circles, labeled CIR1 through CIR5, and you want the operator to be able to measure a circle by simply pressing a key on the keyboard. You could use code similar to the following:

Entire Code

```
DO/
   C1=COMMENT/INPUT,Type a number to measure that circle:
   ,FOR CIR1 - Type 1
   ,FOR CIR2 - Type 2
   ,FOR CIR3 - Type 3
   ,FOR CIR4 - Type 4
   ,FOR CIR5 - Type 5
   ,Any other character exits the loop
   SELECT/C1.INPUT
      CASE/1
         CIR1=FEAT/CIRCLE
         …
         …
         ENDMEAS/
      END_CASE
      CASE/2
         CIR2=FEAT/CIRCLE
         …
         …
         ENDMEAS/
      END_CASE
      CASE/3
         CIR3=FEAT/CIRCLE
         …
         …
         ENDMEAS/
      END_CASE
      CASE/4
         CIR4=FEAT/CIRCLE
         …
         …
         ENDMEAS/
      END_CASE
      CASE/5
         CIR5=FEAT/CIRCLE
         …
         …
         ENDMEAS/
      END_CASE
      DEFAULT CASE
         COMMENT/OPER,Now exiting loop.
      END_DEFAULT CASE
   END_SELECT
UNTIL C1.INPUT < 1 OR C1.INPUT > 5
```

Explanation of Sample Code

SELECT/C1.INPUT

This line of code takes a number or string value (in this case a number) typed by the user and determines which `CASE/END_CASE` block will execute from the input. Notice that `SELECT/END_SELECT` pair surrounds the entire list of code. All `CASE/END_CASE` and `DEFAULT CASE/END_DEFAULT CASE` pairs must reside within these two lines.

`END_SELECT`

This marks the end of the code held inside the `SELECT/END SELECT` pair.

`CASE/1 through CASE/5`

Depending on the value of `C1.INPUT`, one of the CASE code blocks executes. For example, if `C1.INPUT` evaluates to 1, the CASE 1 block of code executes, measuring CIR1. If it evaluates to 2, then the CASE 2 block of code executes, measuring CIR2, and so forth.

`END_CASE`

These lines end the specific case blocks of code.

`DEFAULT CASE`

If the value of the `C1.INPUT` doesn't match any of the defined CASE statements (if the value isn't a number one through five) then the `DEFAULT CASE` code block executes. In this case it displays a message letting you know that you are exiting the loop.

Notice how the `DO / UNTIL` loop surrounds the entire code sample. This allows the user to continue to choose from the menu created from the `COMMENT/INPUT` line until the user selects a character not recognized by the CASE statements.

Case / End Case

The **Insert | Flow Control Command | Control Pairs | Case / End Case** menu option allows you to add a conditional block to the measurement routine. The items between the CASE and the END CASE commands will execute if the expression for the case statement evaluates to a value equal to the expression of the corresponding SELECT command. Otherwise, the block of statements will be skipped. The CASE / END CASE statement block must be located directly after a SELECT command or an END CASE command of a previous CASE / END CASE block. Also, PC-DMIS cannot compare multiple expressions on a single case statement.

The Edit window command line for a CASE / END CASE statement reads:

```
CASE/expression

END_CASE/
```

To insert the Case / End Case option:

1. Place the cursor in the desired location of the Edit window. Note the positional requirements stated above.

2. Select **Case / End Case** from the menu bar. The CASE / End CASE statements will appear in the Edit window.

Default Case / End Default Case

The Insert | Flow Control Command | Control Pairs | Default Case / End Default Case menu option allows you to add a conditional block to the measurement routine. The items between the DEFAULT CASE and the END DEFAULT CASE commands will execute if all other expressions in previous CASE / END CASE blocks within the corresponding SELECT / END SELECT block evaluated to false. Only one DEFAULT CASE / END DEFAULT CASE block is allowed within a SELECT/ END SELECT block. The DEFAULT CASE / END DEFAULT CASE block must be located after all CASE / END CASE blocks within the SELECT / END SELECT block.

The Edit window command line for a DEFAULT CASE / END DEFAULT CASE statement reads:

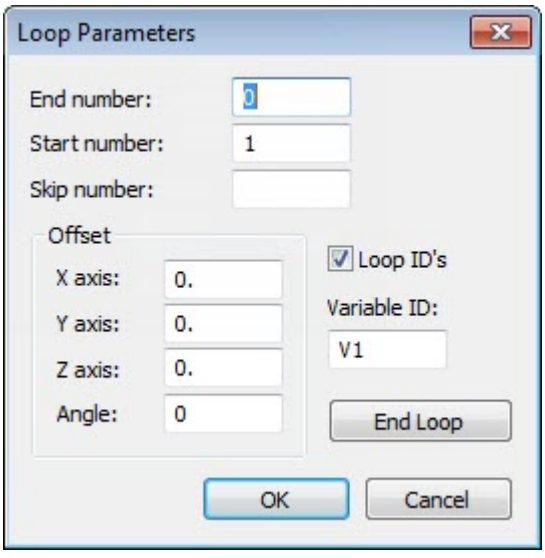
```
DEFAULT CASE/

END_DEFAULT_CASE/
```

To insert DEFAULT CASE/ END DEFAULT CASE commands:

1. Place the cursor in the desired location of the Edit window noting positional limitations as stated above.

2. Select **Default Case / End Default Case** from the menu bar. The DEFAULT CASE / END DEFAULT CASE statements will appear in the Edit window.

Using Generic Loops



Loop Parameters Dialog Box

The **Insert | Flow Control Command | Looping** menu option displays the **Loop Parameters** dialog box. You can use this dialog box to create a `LOOP` command that repeats the measurement routine (or portions of the measurement routine) with or without any of the offsets. The `LOOP` command can be added anywhere in the measurement routine, although this function is most useful at the beginning and end of the measurement routine.

Start Number - This box tells PC-DMIS the starting position number in a series of parts.

Suppose you have 10 parts, and you want to start with position number 5. You would enter 10 for the total number of parts and 5 for the starting position.

End Number - This box tells PC-DMIS how many times to loop through the measurement routine. This number is usually the same as the number of parts that the fixture holds (or patterns on the part) in the X, Y, or Z direction. PC-DMIS also asks for the starting part (pattern) number.

Suppose you have 10 parts in the X direction, and you want to start with position number 5. For the **End Number** box, type 10. For the **Start Number** box, type 5.

Skip Number - In a loop, PC-DMIS repeats a measurement routine the indicated number of times. The **Skip Number** box allows you to skip a specified increment. Note that you cannot *skip* the *first* increment using this box, but you can change the **Start Number** to 2 and effectively skip over it that way.

Suppose you want to set the parameter to skip every third increment of the loop. If the number three is indicated, PC-DMIS measures the first and second part and then skips to the fourth part.

Offset - The offset area contains these boxes:

X Axis, Y Axis, Z Axis - These boxes define the X, Y, and Z offset distances between parts, or patterns on the same part. These values offset the part by the defined offset distances once the loop executes.

Angle - This box sets up the angular offset between parts, or patterns on the same part. The first offset is based on the part's origin. PC-DMIS offsets the part by the angle value once the execution loops.

The software bases the offset distances on the part's origin. The first time through the loop, PC-DMIS uses the part's origin. The second time through the loop, it moves

by the offset distances. The third time through the loop, it moves by the offset distances multiplied by two, and so on.

If you have an alignment command inside a loop, and the loop uses offsets, you *must* define all axes for that alignment. Additionally, the alignment inside the loop must use features measured inside the loop.

Loop IDs - This check box determines whether PC-DMIS displays the current loop value in square brackets in the feature ID.

Suppose you have a circle feature of CIR1. It becomes CIR1[1] on the first time through the loop, CIR1[2] on the second time through the loop, and so on.

Feature IDs in Statistical Databases

If you select the **Loop IDs** check box and send statistical data to a database, in some cases, PC-DMIS may not display these loop IDs in the database.

Consider the following:

If you have a `STATS/ON` command and a `STATS/UPDATE` command *inside a loop block*, then loop IDs are not displayed inside the database.

If you have a `STATS/ON` command *outside a loop block* and a `STATS/UPDATE` command *inside a loop block*, then loop IDs are not displayed inside the database.

If you have a `STATS/ON` command *outside a loop block* and a `STATS/UPDATE` command *outside a loop block*, then loop IDs are displayed inside the database.

If you have a `STATS/ON` command *inside a loop block* and a `STATS/UPDATE` command *outside a loop block*, then loop IDs are displayed inside the database.

Variable ID - This box allows you to define the variable name used to track the loop's current iteration (or current loop within the number of specified loops). During the measurement routine's execution, this variable is equal to the current iteration number of the loop.

End Loop - This button completes the looping process. You must follow the command `LOOP/START` with the command `LOOP/END` in the Edit window.

Uses for Looping

There are three main uses for the looping functionality:

- You have a multiple part fixture that holds a grid of parts. The fixture should use consistent spacing between the rows. The translation / rotation offsets allow you to index from one part to the next in the grid of parts.
- You have a fixture that holds one part and you want to swap in a new part before each loop of the measurement routine. A `COMMENT` command is helpful to stop the CMM when the part is being replaced with a new one. The command can be at the beginning or end of the loop.
- You want to use the loop functionality to rotate the measurement routine to measure a different portion of the same part. For example, you could create a measurement routine to measure a complicated hole pattern that was duplicated ten times on the part. Your measurement routine would only need to measure one of the hole patterns. You could then use the loop functionality to offset the measurement routine to measure the other nine occurrences of the pattern.

If you're using an alignment inside of a loop, PC-DMIS allows you to use the active alignment in the `ALIGNMENT/START` command line instead of always recalling a previously stored alignment. See the "Using an Alignment Inside Loops" topic in the "Creating and Using Alignments" chapter.

Creating a Loop

1. Select **Insert | Flow Control Command | Looping** from the menu bar to display the **Looping Parameters** dialog box.
2. Define the values in the dialog box.
3. Select parameters as needed (such as **Number of Parts**, **Start Number**, **Skip Number**, **Offsets Angle**).
4. Place the cursor in a location in the Edit window where you want to begin the loop.
5. Click the **OK** button to insert the `LOOP/START` command block.

   The Edit window command line for looping reads:

```
VARNAME = LOOP/START, ID = Y/N, NUMBER = 0, START
          = 1, SKIP = , OFFSET: XAXIS = 0, YAXIS = 0,
ZAXIS = 0, ANGLE = 0
```

To complete the looping procedure, you must finish the loop with a `LOOP/END` command. PC-DMIS loops any Edit window commands encompassed by the `LOOP/START` and `LOOP/END` commands.

You can insert the `LOOP/END` command in one of these ways:

- In the Edit window, type `LOOP/END.`
- Select the **Insert | Flow Control Command | End Loop** menu item.
- From the **Looping Parameters** dialog box, click the **End Loop** button.

Using the Print Command in a Loop

To use the `PRINT` command in a loop in PC-DMIS, do this:

1. Insert the `LOOP/START` and `LOOP/END` commands in your measurement routine.
2. Insert the `PRINT` command between the loop commands where you want it.
3. In the Edit window, click on the `PRINT` command. Then press the F9 key to show the **Output Configuration** dialog box for that `PRINT` command. For details on how to set the output and printer options for the Report window, see "Setting Output and Printer Options for the Report Window" topic in the "Using Basic File Options" chapter.
4. Select one of the output format tabs (**Report**, **DMIS**, or **Excel**) to define the format for your `PRINT` command.
5. Select the check box in the upper left of the tab page to enable the options for the output.
6. Define where to send the generated report:
    a. Click the browse (...) button to the right of the path and file name box to display the **Save As** dialog box.
    b. On the **Save As** dialog box, define the file name and click **Save**.
7. Make any other changes that you need. For example, if you want to view each report after PC-DMIS generates them, select the **Show Report** check box.
8. If you want to generate outputs in multiple formats, press the next tab and repeat steps 4 through 7.
9. Click the **OK** button to save your changes to the `PRINT` command.

The `PRINT` command works almost the same inside of loops as it does outside of loops. The difference is that internally, PC-DMIS keeps track of what was output, and only outputs data that was executed after executing the last `PRINT` command.

For example, consider the code snippet below. When PC-DMIS executes the `PRINT` command for the first time, the software creates the first output file with *all* the collected data up to that point. This means the first output file includes data collected before the `LOOP/START` command as well. The next time the `PRINT` command executes, PC-DMIS only executes the contents within the loop. This means the second data file only contains the data from within the loop. This repeats until the measurement routine ends.

```
.
.
.
 Loop Start
      CIR1 = CIRCLE (CONTACT)
      CIR2 = CIRCLE (CONTACT)
      Move Point
      Move Point
      Move Point
      Move Point
      CIR3 = CIRCLE (CONTACT)
      Move Point
      Move Point
      Move Point
      CIR4 = CIRCLE (CONTACT)
      CIR5 = CIRCLE (CONTACT)
      CIR6 = CIRCLE (CONTACT)
      CIR7 = CIRCLE (CONTACT)
      CIR8 = CIRCLE (CONTACT)
      LOC1 Passed: CIR1
      Print Report
 Loop End
.
```

```
.
.
```

Using Labels

You can use a `LABEL` command with a `GOTO` or an `IF GOTO` command to control where the execution flow gets sent based on certain conditions. The label name cannot have spaces and must not exceed 230 characters. PC-DMIS displays the label name using all capital letters.

To create a `LABEL` command, do one of the following:

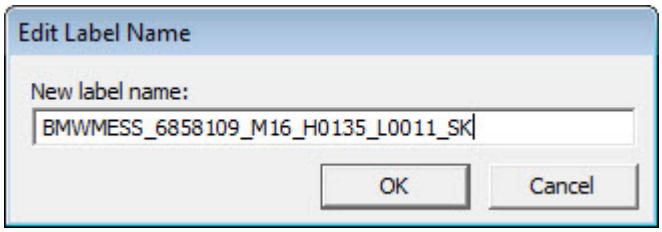
Type the Command

1. Position the cursor on a blank line in the Edit window.
2. Type LABEL and then press the Tab key.
3. In the highlighted field, type a new label name.

or

Choose the Command

1. Position the cursor in the Edit window.
2. From the menu bar, select **Insert | Flow Control Command | Label** to access the **Edit Label Name** dialog box.
3. In the **New label name** box, type the name for the label.



Edit Label Name Dialog Box

4. Click **OK** to insert the label name in the next possible location in the Edit window.

The LABEL command in the Edit window's Command Mode reads:
`ID = LABEL/`

Where ID represents the specified label name.

For information on using the `GOTO` command with labels, see the "Jumping to a Label Using GOTO" topic.

For information on using the `IF_GOTO` command with labels, see the "Jumping to a Label Based on Conditions" topic.

Jumping to a Label Using GOTO

You can tell your measurement routine to jump to a specific label by creating a GOTO command.

To create a GOTO command, do one of the following:

Type the Command

1. Position the cursor on a blank line in the Edit window.
2. Type GOTO and then press the Tab key.
3. Type the label name to jump to.

or

Choose the Command

1. Position the cursor in the Edit window.
2. Select the menu option **Insert | Flow Control Command | Goto** to open the **GoTo** dialog box.

GoTo Dialog Box

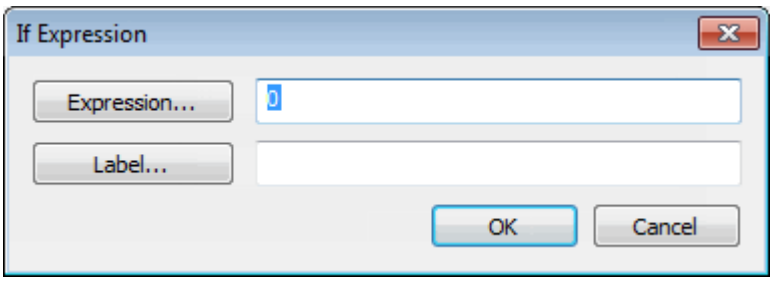
3. If labels already exist in the measurement routine, they appear in the **Current labels** box.
4. From the **Current labels** box, select the desired label, or in the **GoTo Label** box, type a label name.
5. Click **OK** to insert the GOTO command in the next possible location in the Edit window.

The GOTO command in the Edit window's Command Mode reads:
```
GOTO/label_ID
```

Where label_ID represents the name of the specified label. The label name cannot have spaces and must not exceed 230 characters.

If the label has not yet been created, the label name appears highlighted in red in the Edit window's Command Mode and that GOTO command is ignored during execution.

Jumping to a Label Based on Conditions



If Expression Dialog Box

With the **Insert | Flow Control Command | If Goto** option, you can use the **If Expression** dialog box to create IF GOTO statements within your measurement routine. When the measurement routine is executed and PC-DMIS encounters an IF GOTO statement, the routine flow moves to the label identification if the specified expression evaluates to a non-zero value.

The Edit window command line for an IF_GOTO statement reads:

```
IF_GOTO/expression, GOTO=Label
```

Expression - If you click **Expression**, the expression builder opens. With the expression builder, you can create a variety of different expressions that you may need within your measurement routine. Once you create the expression it appears in the **Expression** box. For information on creating expressions, see the "Using Expressions and Variables" chapter.

Label - If you click **Label**, the **Goto** dialog box opens. From the **Goto** dialog box, you can choose what label you want the routine flow to move to when the defined expression is met. The chosen label appears in the **Label** box. Or if you know the label name, you can type it in the box. For more information on labels, see "Jumping to a Label".

Reset Loop

The Reset Loop command resets the execution tree in a loop. You can add the Reset Loop command to support multiple part measurements using these command pairs:

- While/End While
- Do/Until
- Loop/End Loop

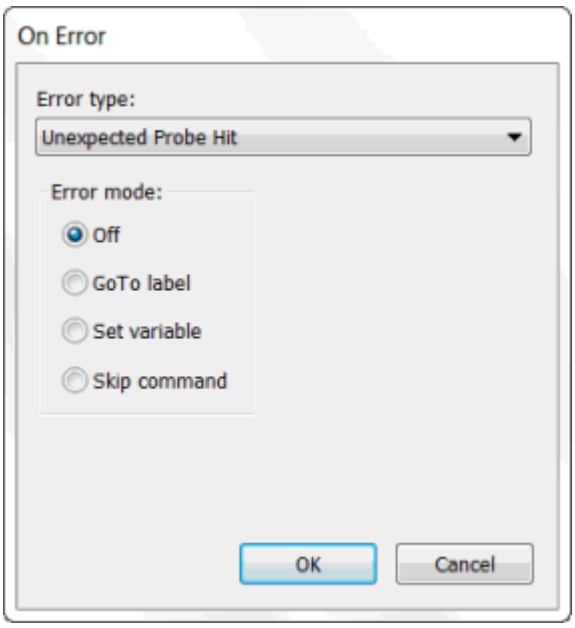
You must use the Reset Loop command within one of these loops. The command has no effect if you place it outside a loop. To insert a Reset Loop command, select **Insert | Flow Control Command | Reset Loop** menu option. The Edit window command line for a Reset Loop command would read: `RESET LOOP/`

For multiple part measurements, you can get a report of the last part measured in the loop. To do this, insert the Reset Loop command at the start of the loop as shown in this image:

The reset loop removes all the objects from the execution tree.

Branching on an Error

The **Insert | Flow Control Command | On Error** option opens the **On Error** dialog box:



On Error Dialog Box

You can use this dialog box to tell PC-DMIS the action to take when a machine error occurs.

Error type - PC-DMIS tracks these error conditions:

- Unexpected probe hit
- Missed probe hit
- Reflector not found - This is used with the Tracker in PC-DMIS Portable.
- Laser error - If you are using a laser configuration, see the "Handling Laser Sensor Errors with On Error Command" topic in the PC-DMIS Laser documentation.
- Temperature Outside Limits - The Temperature Compensation command in the measurement routine raises this error if one or more of the temperatures for the X axis, Y axis, or Z axis scale or part are above the maximum limit or below the minimum limit that is defined for the Temperature Compensation command.

  The On Error command must be placed above the Temperature Compensation command in the measurement routine.

- Calibration Outside Limits - The Check Calibration Limits command in the measurement routine raises this error if one or more calibration error checks are outside the limits. For more information on the Check Calibration Limits command, see "About Check Calibration Limits" in the PC-DMIS Toolkit Modules documentation.

The On Error command must be placed above the Check Calibration Limits command in the measurement routine.

Error mode - PC-DMIS can take these possible actions depending on the error type:

- **Off** - PC-DMIS does nothing.
- **GoTo label** - The measurement routine flow moves to a defined label (see "Using Labels"). These options become available:
  - **Label ID** - Type a reference to a label that doesn't exist yet.
  - **Current labels** - Lists all the labels in the measurement routine.
- **Set variable** - Sets a variable's value to one.
- **Skip command** - The measurement routine flow skips over the current command and moves to the next marked command in the measurement routine.

For details on how PC-DMIS reports dimensions that use skipped features during execution, see the "Reporting a Dimension Using a Feature Skipped During Execution" topic in this documentation.

By default, all measurement routines start with the action for both types of errors set to **Off** (PC-DMIS does nothing). The action mode for each error type can be changed throughout the measurement routine.

During execution, if PC-DMIS encounters an `ONERROR/UNEXPECTED HIT/JUMPTOLABEL` command, any unexpected hits that occur after that point in the measurement routine cause the execution to jump to the specified label. The action to "set a variable's value to one" causes the variable to be set as soon as the specified error type occurs. This value of the variable can then be tested using an IF statement to cause execution to jump to a new point in the measurement routine.

Using the On Error Command

1. Select **Insert | Flow Control Command | On Error** to open the **On Error** dialog box.
2. In the **Error type** list, select the error type as the condition for branching.
3. In the **Error mode** area, select the option that defines the action to take place when that condition is met.
4. To add the On Error command, click **OK**. To close the dialog box without applying any changes, click **Cancel**.

On Error Supported Interfaces

Some interfaces support the On Error command. To see if your interface supports the command, refer to the following table.

- If your interface is in the table, a small black box indicates the error type that the interface supports.
- If your interface is not in the table, then it's not able to use the On Error command.

Supported Interfaces	Unexpected Probe Hit	Missed Probe Hit	Reflector Not Found
DEA	-	■	-
FDC	■	■	-
I++ DME Client	■	■	-
Leica Tracker	-	-	■
Leitz	■	■	-
LK Direct (also known as LKRS232)	■	■	-
LK Driver	■	■	-
Mora	■	■	-
Sharpe	■	■	-
Sheffield	■	■	-
Wenzel	■	■	-
Zeiss	■	■	-

Branching with Subroutines

Subroutines are blocks of code in your measurement routine or in an external measurement routine that are usually referenced repeatedly, allowing for more concise programming. PC-DMIS allows you to pass information to "arguments" (or local variables) in the subroutine. The types of arguments that can be passed into a subroutine are numeric values, variables, text strings, and feature names.

Subroutine command blocks are enclosed within SUBROUTINE and ENDSUB commands.

Once you have created a subroutine in your measurement routine you can "call" it from your current measurement routine or from another measurement routine, causing the measurement routine execution flow to go into the specified subroutine, executing commands contained within the subroutine command block. Measurement routine flow will then return to the statement immediately following the calling statement.

Subroutines are called by using the CALLSUB command.

External Subroutines

External subroutines, or subroutines located in a measurement routine outside of the calling measurement routine, do not have access to features, variables, or alignments from the calling measurement routine. The subroutine would still have access to items within its own measurement routine. The external measurement routine and the calling measurement routine must use the same units of measurement.

Nesting Subroutines

You can nest subroutines within other subroutines. The only limitation to the number of arguments and nested subroutines is the amount of available memory.

Creating a New Subroutine

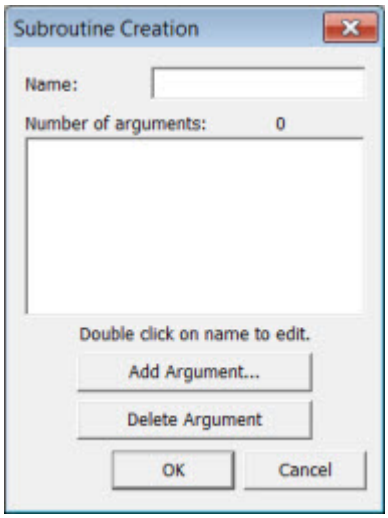
Creating a Subroutine by Typing SUBROUTINE

You can insert this command by typing SUBROUTINE in the Edit window's Command Mode, and then pressing TAB. Once the command is inserted, you will need to specify the subroutine's name and any arguments it has. See the subroutine syntax and example below for this information.

Type the ENDSUB command and press TAB to end the command block. Any Edit window commands placed within this command block will be considered part of the subroutine and will be executed when the subroutine is called.

Creating a Subroutine by Using the Subroutine Menu Item

1. Select **Insert | Flow Control Command | Subroutine** from the submenu. This displays the **Subroutine Creation** dialog box. For information on this dialog box, see the "Understanding the Subroutine Creation Dialog Box" topic.

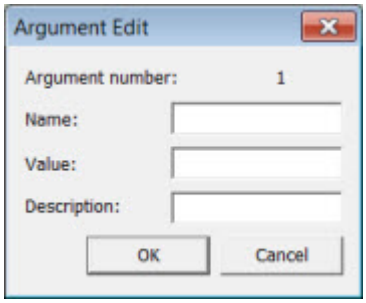


Subroutine Creation Dialog Box

2. Give the subroutine a name by typing it in the **Name** box. The name is limited to 180 or fewer characters. If you enter 181 or more characters a "Line too long" error message appears.

   When you click the **OK** button, the name is automatically shortened to the first 180 characters, and the **Subroutine Creation** dialog box closes.

3. If your subroutine uses arguments (place holders for information passed into the subroutine), add them one by one by clicking the **Add Argument** button. The **Argument Edit** dialog box appears. For information on this dialog box, see the "Understanding the Argument Edit Dialog Box" topic.



Argument Edit Dialog Box

4. Give your argument a name by typing it in the **Name** box.

5. Give your argument a default value by typing it in the **Value** box. The subroutine will use the default value if no values are passed into the subroutine from the `CALLSUB` statement. Valid argument values can be numeric values, variables, text strings, and feature names.

6. If you want to give the argument a description, type it in the **Description** box.

7. Click **OK** in the **Argument Edit** dialog box to create the argument.
8. Repeat steps 3 through 7 for each argument you want in your subroutine.
9. Click the **OK** button in the **Subroutine Creation** dialog box to finish creating your subroutine. This subroutine will appear inside the Edit window with any defined arguments.
10. End your subroutine by selecting the **Insert | Flow Control Command | End Sub** menu option. This places an `ENDSUB/` command in the Edit window, completing the subroutine's command block. Any other measurement routine commands you want in your subroutine must be added inside the subroutine's command block, before the `ENDSUB` command.

Syntax for a Subroutine Command Block

The Edit window command line syntax for a sample subroutine command block would read:

```
SUBROUTINE/<Name>,
<A1> = <Arg1> : <Description>,
<A2> = <Arg2> : <Description>,
=
<Commands>
ENDSUB/
```

SUBROUTINE/ is the command used to start the subroutine command block.

<Name>= The name of the subroutine. The name is limited to 256 or fewer characters. If you enter 257 or more characters, the name is automatically shortened to 256 characters.

<A1>= The first argument (or local variable) used in the subroutine. This variable generally cannot be accessed outside of the subroutine.

<A2>= Second argument used in the subroutine. This variable generally cannot be accessed outside of the subroutine. Additional arguments can be added as needed.

<Arg1> = The default value for the first argument.

<Arg2>= The default value for the second argument.

<Description> = The description for the argument.

<Commands> = Other Edit window commands can be inserted as needed after the arguments and before the ENDSUB command.

ENDSUB/ is the command used to end the subroutine command block.

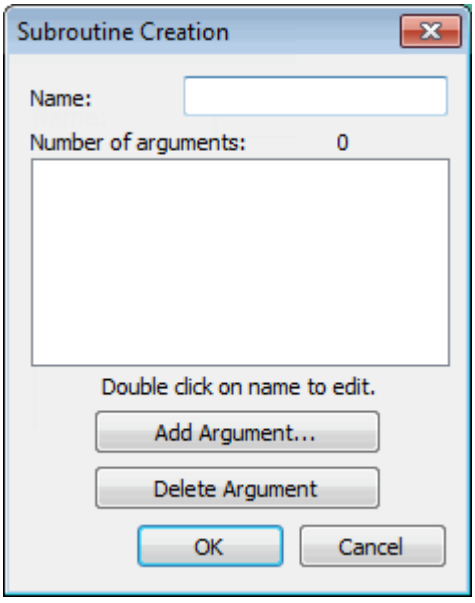
Example Subroutine Command Block

For example, a finished subroutine that takes operator data and displays it to the report might look like this:

```
SUBROUTINE/GET_OPERATOR_INFO,
    OPNAME = <Operator> : OPERATOR NAME,
    SHIFT = <Shift> : SHIFT TIME,
    =
   COMMENT/REPT,OPNAME
   COMMENT/REPT,SHIFT
ENDSUB/
```

Understanding the Subroutine Creation Dialog Box

To access the **Subroutine Creation** dialog box, select **Insert | Flow Control Command | Subroutine**.



Subroutine Creation Dialog Box

The following options are available in the **Subroutine Creation** dialog box:

Name - This box defines your subroutine. This is the name to use when you call the subroutine later. If you have multiple subroutines in a single measurement routine, each name must be unique.

> The name is limited to 180 or fewer characters. If you enter 181 or more characters in the **Name** box, a "Line too long error" message appears.
>
> When you click the **OK** button, the name is automatically shortened to the first 180 characters, and the **Subroutine Creation** dialog box closes.

Number of arguments - This list shows the arguments for the subroutine you are creating. Arguments appear in this area in this form:

```
<NAME> = <VALUE> : <DESCRIPTION>
```

For example, if one of your arguments is named "Diameter" with a default value of 3, your argument in this list may appear as:

```
DIAMETER = 3 : The hole's diameter
```

PC-DMIS uses the default value whenever another value is not passed from the `CALLSUB` command.

To edit an argument, double-click on the argument you want to change. The **Argument Edit** dialog box opens, allowing you to make the changes. See "Understanding the Argument Edit Dialog Box" for information on this dialog box.

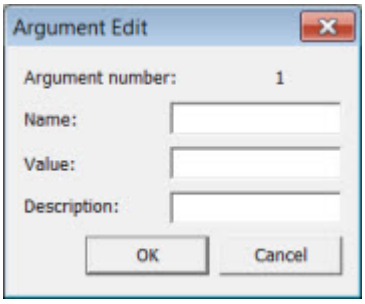
Add Argument - This button adds new arguments to your subroutine. Click on this button to open the **Argument Edit** dialog box appears. For information on this dialog box, see "Understanding the Argument Edit Dialog Box".

Delete Argument - This button allows you to delete arguments from your subroutine. Select the argument from the list and then click the **Delete Argument** button to remove it.

Editing an Existing Subroutine

To edit an existing subroutine you can always use the Edit window's Command Mode and edit the subroutine directly. Alternately, you can access the **Subroutine Creation** dialog box by placing your cursor on the subroutine in the Edit window and pressing F9. This brings up the **Subroutine Creation** dialog box. For information on this dialog box, see the "Understanding the Subroutine Creation Dialog Box" topic.

Understanding the Argument Edit Dialog Box



Argument Edit Dialog Box

The **Argument Edit** dialog box appears whenever you choose to create or edit an argument within either the **Subroutine Creation** dialog box (**Insert | Flow Control Command | Subroutine**) or the **Call Subroutine** (**Insert | Flow Control Command | Call Sub**) dialog box.

The **Argument Edit** dialog box can be used in these two contexts:

- To define a subroutine's arguments and their default values in a `SUBROUTINE` command block.
- To define the values that will be passed into the subroutine from a `CALLSUB` command.

The following options are available in the **Argument Edit** dialog box:

Name - This box defines the name for the argument you are creating or editing.

Value - This box defines the value of the argument.

If creating or editing the `SUBROUTINE` command, this is the default value used if no value is passed into the subroutine by the `CALLSUB` statement.

If creating or editing the `CALLSUB` command, this is the value passed into the subroutine.

Valid values can be:

- Numeric
- Variable
- Text String - Text strings must be enclosed in double quotation marks.
- Feature Name - The feature name must be bounded by curly brackets, for example {F1}.

Description - Defines the description of the argument for the subroutine. This description will appear next to the argument in the Edit window's `SUBROUTINE` command block.

Calling a Subroutine

To call a subroutine, you need to insert a `CALLSUB` command into your measurement routine to call an existing subroutine from the current measurement routine or a subroutine from an external measurement routine.

When a CALLSUB command executes, PC-DMIS executes all the commands in the subroutine. This includes even unmarked commands.

The called subroutine must have been created in a valid PC-DMIS version (versions 2021.1 through 2026.2).

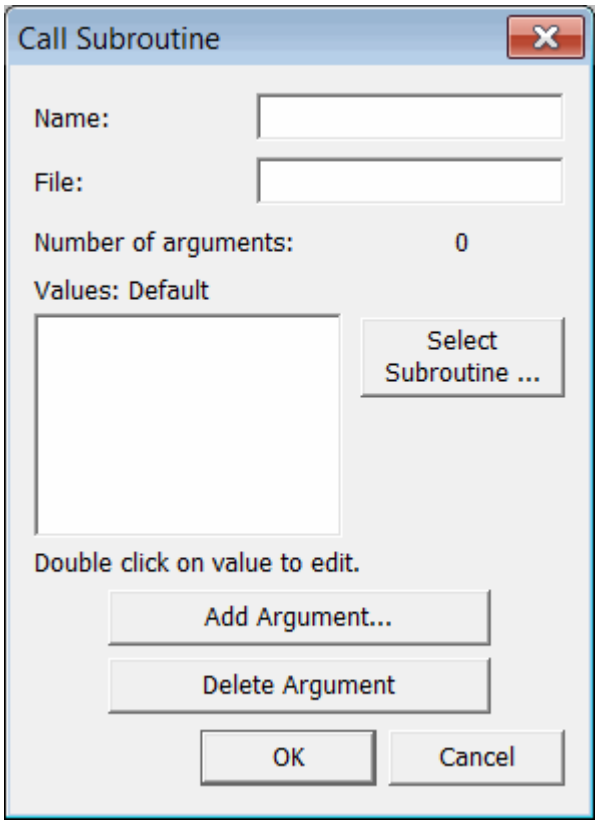
Calling a Subroutine by Typing CALLSUB

You can insert this command by typing CALLSUB in the Edit window and then pressing the Tab key on your keyboard where you want the command to appear in the Edit window.

Once the command is inserted, you will need to specify the subroutine's name, its location if it is in an external measurement routine, as well as any values to pass to available arguments. See "Passing Arguments into a Subroutine" for examples of passing arguments.

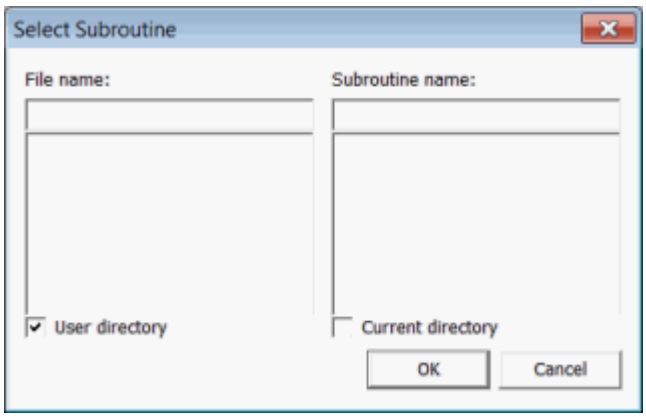
Calling a Subroutine Using the Call Sub Menu Item

1. Select the **Insert | Flow Control Command | Call Sub** option from the submenu. The Call Subroutine dialog box opens. See "Understanding the Call Subroutine Dialog Box" for information on this dialog box.



Call Subroutine Dialog Box

2. Click the **Select Subroutine** button. The Select Subroutine dialog box opens.



Select Subroutine Dialog Box

3. Select either the **User directory** check box or the **Current directory** check box, or both. If the measurement routine the subroutine is from resides in the directory specified to be searched for subroutines, select the **User directory** check box. If it is from the current directory, select the **Current directory** check box. PC-DMIS lists all the measurement routines available for selection.

4. Select the measurement routine that contains the subroutine you want. You will see all subroutines associated with the selected measurement routine appear in the **Subroutine name** box.

5. Select the subroutine you want to call.

6. Click the **OK** button. The subroutine information you are going to call will appear in the **Name** and **File** boxes of the **Call Subroutine** dialog box.

7. If you want to pass information into the subroutine, click the **Add Argument** button, and use the **Argument Edit** dialog box to define arguments and values to pass. See "Understanding the Argument Edit Dialog Box" for information on this dialog box. See "Passing Arguments into a Subroutine" for examples of passing arguments.

8. Click the **OK** button again. PC-DMIS adds the `CALLSUB` command in the selected location of the Edit window.

Syntax for the CALLSUB Command

The Edit window command line syntax for calling a subroutine is this:

```
CS1         =CALLSUB/<Name>, <File>:<Arg1>,<Arg2>,
```

CS1 = the label ID given to the `CALLSUB` command.

<Name> = the name of the subroutine to be called.

<File> = the full path way to the measurement routine that contains the subroutine to call. If this field is blank, PC-DMIS looks in the current measurement routine for the subroutine.

<Arg1> = the value to be passed to the first argument in the subroutine. If this field is blank, PC-DMIS uses the default value defined for the first argument in the subroutine instead.

<Arg2> = the value to be passed to the second argument in the subroutine. If this field is blank, PC-DMIS uses the default value defined for the second argument in the subroutine instead. This syntax sample shows only two arguments. You can pass other arguments to your subroutine if needed.

In your `CALLSUB` command, you should keep a set of pointers to all of the objects made for the subroutine so that you can easily refer to them afterwards, using the subroutine's ID. For more information on pointers, see "Pointers" in the "Using Expressions and Variables" chapter.

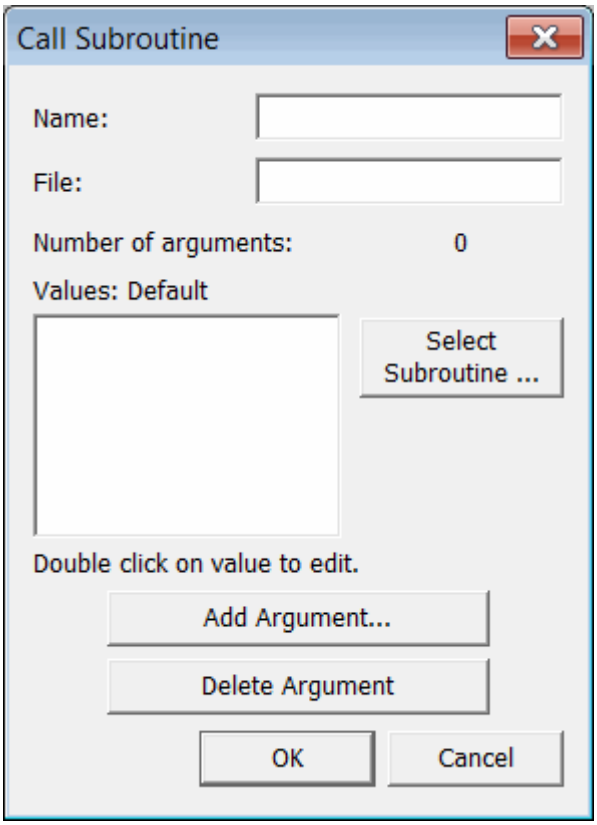
Example CALLSUB Command

```
CS1         =CALLSUB/GET_OPERATOR_INFO,D:
\MEASROUTINES\V42SUBROUTINETEST.PRG:V1,V2,,
```

This example `CALLSUB` command, CS1, calls a subroutine named GET_OPERATOR_INFO located within the measurement routine V42SUBROUTINETEXT.PRG located in the D:\MEASROUTINES\ directory.

It passes two values - in this case the variables V1 and V2 - into the subroutine.

Understanding the Call Subroutine Dialog Box



Call Subroutine Dialog Box

The following describes the options available in the **Call Subroutine** dialog box.

Name box

The **Name** box contains the name of the subroutine you have selected after using the **Select Subroutine...** button.

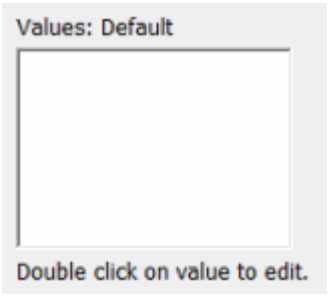
The name is limited to 180 or fewer characters. If you enter 181 or more characters in the **Name** box, a "Line too long error" message appears.

When you click the **OK** button, the name is automatically shortened to the first 180 characters, and the **Call Subroutine** dialog box closes.

File box

The **File** box contains the directory pathway to the subroutine file you have called.

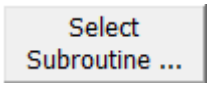
Values box



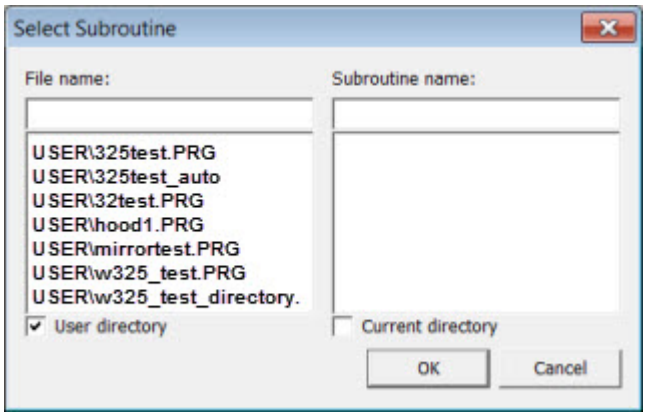
The **Values** box contains a list of the values of each argument associated with the subroutine. These values will get passed into the subroutine when the subroutine is executed.

To change these values, double-click on the value you want to change. The **Argument Edit** dialog box appears. See "Understanding the Argument Edit Dialog Box" for information on this dialog box.

Select Subroutine button



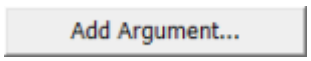
The **Select Subroutine** button displays the **Select Subroutine** dialog box



Select Subroutine Dialog Box

This dialog box allows you to call previously-created subroutines by searching in the user directory or the current directory. See "Calling a Subroutine" for information on the **Select Subroutine** dialog box.

Add Argument button



This button defines a value to pass to the arguments for the subroutine.

Delete Argument button

This button allows you to delete arguments from the **Values** box. Select the value displayed, and then click the **Delete Argument** button. The argument associated with that value is then deleted.

To add a new argument using the Call Subroutine Dialog box:

To add a new argument into your `CALLSUB` command to pass into the subroutine:

1. Inside the **Call Subroutine** dialog box, click the **Add Arg** button to open the **Argument Edit** dialog box.
2. Click in the **Value** box.
3. In the **Value** box, type the value for the argument.
4. Click **OK**.

To edit existing arguments using the Call Subroutine Dialog box:

To edit an existing argument inside your `CALLSUB` command:

1. In the **Call Subroutine** dialog box, double-click on the value of the argument you want to change. A value box appears and shows the default value for the called subroutine.
2. Type a new value.
3. Click the **OK** button.

For additional information on how to edit or create new arguments for a subroutine, see "Creating a New Subroutine" and "Editing an Existing Subroutine".

To delete arguments from a Call Subroutine command:

1. Place the cursor on the `CALLSUB` command.
2. Press F9 to access the **Call Subroutine** dialog box.
3. From the list of arguments, select one or more arguments.
4. Click the **Delete Arg** button.
5. Click **OK**.

You can also delete an argument in the Edit window text directly. To do this,

1. Place PC-DMIS in Command Mode.

2. Place the cursor on the `CALLSUB` command, and press TAB until you highlight the desired argument.
3. Type the letters "del" to delete the argument. Note that pressing Delete or Backspace doesn't truly delete the argument, they merely change the argument to an empty argument.

Using CALLSUB Statements in Multiple Arm Mode

If you assign a `CALLSUB` *statement to Arm1*; all of the commands in the subroutine are assigned to Arm1 when the subroutine is called.

If you assign a `CALLSUB` *statement to Arm2*; all of the commands in the subroutine are assigned to Arm2 when the subroutine is called.

If you mark a `CALLSUB` *statement for both arms*, PC-DMIS leaves the subroutine markings as they were originally set.

If a subroutine contains a `MOVE/SYNC` *command in it*, and you assign the `CALLSUB` statement to Arm1 or Arm2, at execution time PC-DMIS displays an error indicating that this is invalid and the subroutine is not called.

For information on assigning a command to execute for a specific arm, see the "Assigning a Command to an Arm" topic in the "Using Multiple Arm Mode" chapter.

Subroutine Examples

Consider the information in the following topics for some examples of passing arguments and subroutines in general.

- Passing Arguments into a Subroutine
- Code Sample of a Subroutine

Passing Arguments into a Subroutine

The types of arguments that you can pass into a subroutine are numeric values, variables, text strings, and feature names. To pass values into arguments, type the value within the **Value** box of the **Argument Edit** dialog box, or directly into an inserted `CALLSUB` command in Command Mode.

For details on how to do this, see "Creating a New Subroutine" and "Understanding the Argument Edit Dialog Box".

Passing Variables into a Subroutine

Arguments that can pass data back are variables. When you use a variable as an argument to a subroutine, any changes that occur to the corresponding variable in the subroutine are passed back and become the value of the variable that was passed in.

Example of Passing Variables:

This example shows how a variable's value, used as an argument to a subroutine, is modified and passed back from a subroutine:

```
            ASSIGN/V1=6
CS4        =CALLSUB/MYSUB,:V1,,
            SUBROUTINE/MYSUB,
                A1 = 0 : FIRST ARGUMENT,
                 =
            ASSIGN/A1=A1+1
            COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
            The value of A1 is:
            A1
            ENDSUB/
```

Code Explanation with Execution Flow

First, the V1 variable is assigned a value of 6.

The `CALLSUB` statement then calls the subroutine and passes the V1 variable as the first argument.

The `SUBROUTINE/MYSUB` statement defines MYSUB as a subroutine.

A1 is the name for the first argument; so when the call is made, A1 initially has the same value that V1 has at the time of the call, 6.

The subroutine then executes its two statements:

- The first statement is an `ASSIGN` command that increments the value of A1 to 7.
- The second statement is a `COMMENT/OPER` command that shows the updated A1 value.

The subroutine then ends with the `ENDSUB/` command.

The flow of execution then returns to the statement that directly follows the `CALLSUB` command. This is another `COMMENT/OPER` command that shows the final value of V1.

As execution jumps back, any variables that were used as arguments, V1 in this case, are updated to the value of the corresponding variables in the subroutine, A1 in this case. So, after returning from the subroutine, V1 now has a value of 7. That value was passed back from the subroutine.

Passing Number Values into a Subroutine

Arguments can also take numeric characters.

Example of Passing Number Values:

This example shows how to pass number values into a subroutine. It passes up to two numbers and then adds them together.

```
CS1         =CALLSUB/SUM_NUMBERS,:,
CS2         =CALLSUB/SUM_NUMBERS,:5,10,,
             SUBROUTINE/SUM_NUMBERS,
                 NUM1 = 1 : FIRST NUMBER,
                 NUM2 = 1 : SECOND NUMBER,
                  =
             ASSIGN/TOTAL=NUM1+NUM2
             COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
             NUM1 + "+" + NUM2 + "=" + TOTAL
             ENDSUB/
```

In the first `CALLSUB` command, (CS1), no number values are passed into the subroutine. The default numbers, 1 for NUM1 and 1 for NUM2 are used instead, and the sum total is 2.

In the second `CALLSUB` command, (CS2), two number values are passed, 5 and 10. So, NUM1 is 5 and NUM2 is 10, and the sum total is 15.

Passing Text Strings into a Subroutine

Arguments can also take a text string. To pass a text string, make sure the alphanumeric characters are placed within double quotation marks.

Example of Passing Text Strings:

This example shows how to pass string values into a subroutine. It passes two string values into the two parameters and then displays them in the report:

```
CS1         =CALLSUB/GET_OPERATOR_INFO,:"BOB
JONES","MORNING",,,
             SUBROUTINE/GET_OPERATOR_INFO,
                 OPNAME = "<NO PASSED OPERATOR>" :
OPERATOR NAME,
                 SHIFT = "<NO PASSED SHIFT>" : SHIFT
TIME,
                  =
             COMMENT/REPT,
             OPNAME
             COMMENT/REPT,
             SHIFT
             ENDSUB/
```

The first argument, `OPNAME` , receives the passed value of "BOB JONES" and the second argument, `SHIFT` , receives "MORNING". The two `COMMENT/REPT` commands then send the passed strings to the inspection report.

Passing Feature Names into a Subroutine

You can pass feature names within curly brackets. For example, in the `CALLSUB` command, if you use {PNT1} as a parameter, the software passes PNT1 into the subroutine. When you pass the feature name in this manner, the subroutine has full access to that feature.

Example of Passing Feature Names:

This example passes in the PNT1 feature name into the subroutine, giving your subroutine complete access to the feature. If no value is passed then the default feature name of F1 is used. This subroutine then queries you for a new value and changes the theoretical X value for the feature.

```
CS1         =CALLSUB/CHANGE_THEO_X,:{PNT1},,
C1          =COMMENT/INPUT,NO,FULL SCREEN=NO,
             Passed feature:
             FEAT1
             The current theo X is:
             FEAT1.TX
             Type a new theo X value:
             ASSIGN/FEAT1.TX=C1.INPUT
             COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
             "Feature updated to " + FEAT1.TX
```

Because the subroutine has access to the passed feature, the statement `ASSIGN/FEAT1.TX=C1.INPUT ?>` modifies the theoretical X value of the actual PNT1 feature. PNT1 will now permanently have its theoretical X value changed.

Code Sample of a Subroutine

The following code sample allows the operator to have a choice of changing the theoretical X, Y, and Z values of a feature after its measurement. Subsequent runs then use the updated theoretical values.

```
PNT1        =GENERIC/POINT,DEPENDENT,CARTESIAN,$
             NOM/XYZ,<5,10,15>,$
             MEAS/XYZ,<7,12,17>,$
             NOM/IJK,<0,0,1>,$
             MEAS/IJK,<0,0,1>
C1          =COMMENT/YESNO,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,
             Do you want to change the theoretical values for
PNT1?
             IF/C1.INPUT=="YES"
CS1         =CALLSUB/CHANGETHEO,:,
             END_IF/
             COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,
             The XYZ theoretical and actual values for PNT1
are:
             "Theo X= "+PNT1.TX
             "Theo Y= "+PNT1.TY
             "Theo Z= "+PNT1.TZ
             --------------------
             "Actl X= "+PNT1.X
             "Actl Y= "+PNT1.Y
             "Actl Z= "+PNT1.Z
             ROUTINE/END
             SUBROUTINE/CHANGETHEO,
                 POINT1={PNT1}:,
                  =
             DIMINFO/PNT1;ICON,DIMID,FEATID,VERT,HORIZ,,$
                    HEADINGS,;MEAS,,,,,,,,
C2          =COMMENT/INPUT,NO,FULL SCREEN=NO,
             Type the new X theo value for PNT1.
             "Its current value is "+PNT1.TX
             ASSIGN/PNT1.TX=C2.INPUT
C3          =COMMENT/INPUT,NO,FULL SCREEN=NO,
             Type the new Y theo value for PNT1.
             "Its current value is "+PNT1.TY
             ASSIGN/PNT1.TY=C3.INPUT
C4          =COMMENT/INPUT,NO,FULL SCREEN=NO,
             Type the new Z theo value for PNT1.
             "Its current value is "+PNT1.TZ
             ASSIGN/PNT1.TZ=C4.INPUT
             ENDSUB/
```

Explanation of Sample Code

C1=COMMENT/YESNO

This line takes and stores the YES or NO response from the user.

```
IF/C1.INPUT=="YES"
```

This line is the expression. It tests to see if the input of comment 1 is a YES. If it's a YES then the IF statement is TRUE and continues executing the statements after the IF statement, in this case it measures the `PNT1` feature. If NO it moves to the END_IF statement.

```
CS1=CALLSUB/CHANGETHEO,:,
```

This line calls the subroutine named CHANGETHEO. The flow of the measurement routine now jumps to the `SUBROUTINE/CHANGETHEO` line.

```
SUBROUTINE/CHANGETHEO
```

This line initializes the CHANGETHEO subroutine. Measurement routine flow continues with the execution of code between this line and the `ENDSUB/` line.

```
POINT1={PNT1}:,
```

This is the only argument of the subroutine. It allows the subroutine to access information from the PNT1 feature.

```
C2=COMMENT/INPUT,C3=COMMENT/INPUT,C4=COMMENT/INPUT
```

These input comments all take the new theoretical X, Y, and Z values from the user and store them in C2.INPUT, C3.INPUT, and C4.INPUT respectively.

```
ASSIGN/PNT1.TX=C2.INPUT
```

This line takes the theoretical X value from C2.INPUT and assigns it to the `PNT1.TX` variable. `PNT1.TX` is a PC-DMIS variable that holds the theoretical X value (denoted by TX) for the point with the ID label of PNT1.

```
ASSIGN/PNT1.TY=C3.INPUT
```

This line takes the theoretical Y value from C3.INPUT and assigns it to the `PNT1.TY` variable. `PNT1.TY` is a PC-DMIS variable that holds the theoretical Y value (denoted by TY) for the point with the ID label of PNT1.

```
ASSIGN/PNT1.TZ=C4.INPUT
```

This line takes the theoretical Z value from C4.INPUT and assigns it to the `PNT1.TZ` variable. `PNT1.TZ` is a PC-DMIS variable that holds the theoretical Z value (denoted by TZ) for the point with the ID label of PNT1.

```
ENDSUB/
```

This line ends the subroutine, and measurement routine flow returns to the line immediately following the subroutine call. In this case, the `END_IF/` statement. The measurement routine flow then continues with the next operator comment which displays the theoretical and actual X, Y, and Z values, and then the measurement routine ends with the `ROUTINE/END` command.

Command Mode Commands after Comments

After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

Ending a Measurement Routine

The **Insert | Flow Control Command | End Routine** menu item inserts a `ROUTINE/END` command into the Edit window. Whenever PC-DMIS encounters this command during routine execution, it immediately stops measurement routine execution.

This command is useful when you want to end a measurement routine earlier than usual, based on defined conditions.

Tracking Statistical Data

Tracking Statistical Data: Introduction

PC-DMIS allows you to track and manage statistical data for your measured parts. Information in any dimension or tracefield can be sent to a selected statistical software package by inserting a `STATS` command before the dimension or tracefield. Alternately, you can send statistics to a text-based XML file by using the `XMLSTATS` command.

The main topics in this chapter include the following:

- Using Tracefields
- Sending Current Stats to a File
- Using the Statistics Options Dialog Box
- Sending Data to a Local DataPage+ Database
- Sending Statistics to an XML file using DataPageStats.exe
- Sending Statistics to an XML File using XMLSTATS/ON
- Table of Supported Methods

Using Tracefields

PC-DMIS uses tracefields to send additional information into your statistical database along with your measurement results. A tracefield consists of a name, a message, and a value.

You can use the following items with tracefields:

Trace Field dialog box - You can use this dialog box to insert a `TRACEFIELD` command in the Edit window. For more information on this dialog box, see "Inserting Tracefields".

Read Trace Value in a Variable - You can use the `GETTRACEVALUE()` function to get trace values from a variable. For more information on how to read a trace value from a variable, see "GETTRACEVALUE()" under "String Functions" in the "Using Expressions and Variables" chapter.

Q-DAS Tracefields - You can insert Q-DAS tracefields and send K-Field information to your database. For more information on how to insert Q-DAS tracefields, see "Inserting Q-DAS Tracefields".

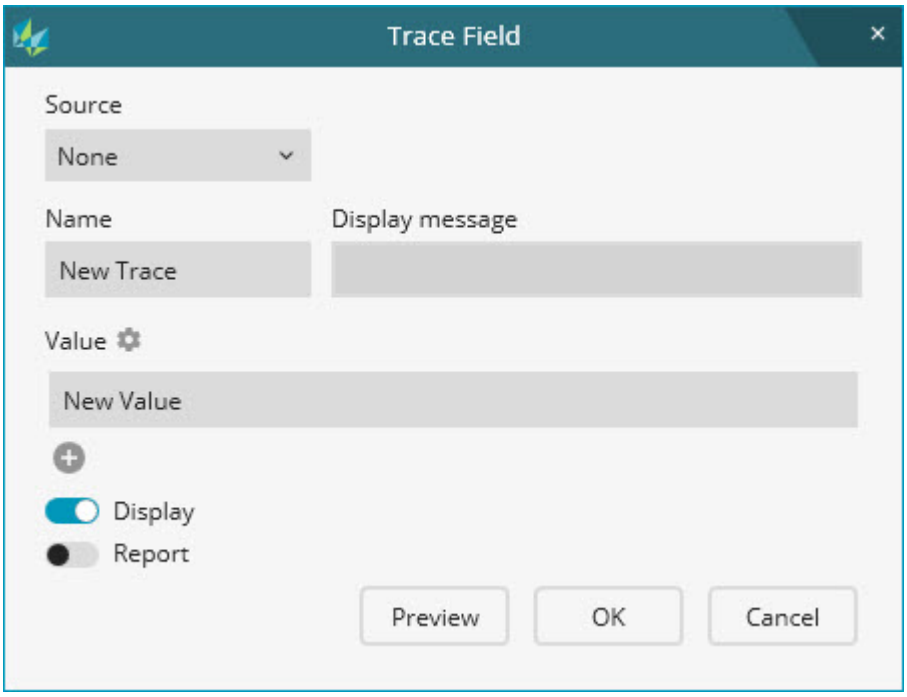
Trace used in Report Header - This item lets you display the values in the report header. For more information on how to use tracefields in the report header, see "Variables and Headers" under "Displaying a Variable's Value" in the "Reporting Measurement Results" chapter.

Inserting TRACEFIELD Commands

With the **Trace Field** dialog box, you can insert a `TRACEFIELD` command in the Edit window. (You can also manually type the command into the Edit window.)

Trace Field Dialog Box

Select **Insert | Statistics Command | Tracefield** to open the **Trace Field** dialog box.



Trace Field Dialog Box

Source - This list defines the location from where PC-DMIS gets the tracefield information.

None - This option lets you type the tracefield name, display message, and value in the dialog box.

Q-DAS - This option reads tracefield information from the Q-DAS configuration file. This option is only available if you insert the `TRACEFIELD` command after the `STATS/ON,Q-DAS` command in a measurement routine. For more information on how to create a Q-DAS configuration file, see the Q-DAS Configurator help file.

Name - This box defines the tracefield's name.

If you select **None** from the **Source** list, type the trace name in the box.

If you select **Q-DAS** from the **Source** list, the **Name** box becomes a drop-down list. From the list, select the trace name.

The drop-down list displays the trace names from the Q-DAS configuration file. (You can choose a different configuration file from the **Statistic/ON Q-DAS** dialog box.)

Each trace name displays a tooltip that provides information about the K-field description.

Display message - This message is the message that PC-DMIS shows for the trace during execution.

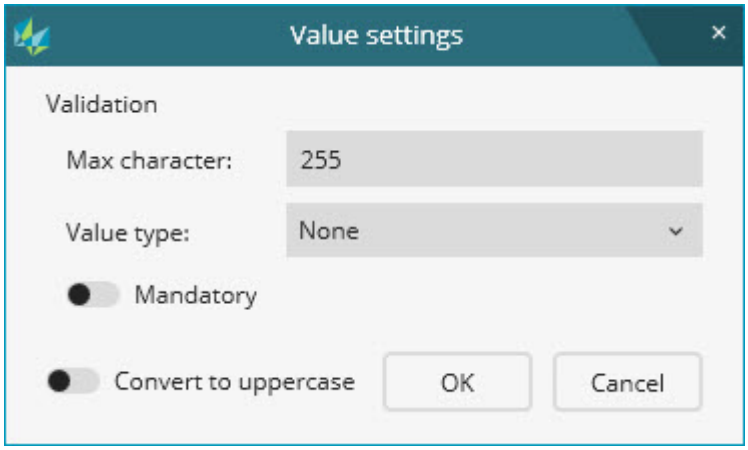
If you select **None** from the **Source** list, type the message in the box.

If you select **Q-DAS** from the **Source** list, the **Display message** box displays a trace name's description from the Q-DAS configuration file. You cannot edit this description.

Value - This box defines the tracefield's value. You can define the settings for the value.

Click the **Value settings** icon to open the **Value settings** dialog box. This dialog box lets you define settings for the value. When you type the value in the **Value** box, PC-DMIS validates the value with the settings.

If you select **Q-DAS** from the **Source** list, you cannot define the settings in the **Value settings** dialog box. PC-DMIS validates the value based on the selection of the configuration file.



Value settings Dialog Box

Max character - This option defines the maximum number of characters for the value. You can type any number between 1 and 255, inclusive. The default value is 255.

Value type - This option defines the type of value. From the list, you can select the type.

None - This type lets you enter any value with the limited number of characters set in the **Max character** box.

Integer - This type lets you enter the whole number.

Float - This type lets you enter the decimal value.

Date/Time (dd.MM.yyyy[/HH:mm:ss]) - This type lets you enter the date and time in the format:

dd.MM.yyyy/HH:mm:ss or dd.MM.yyyy

where

dd - Day of the month as digits. Use a leading zero for single-digit dates.

MM - Month as digits with a leading zero for single-digit months.

yyyy - Year represented by four digits.

HH - Hours with a leading zero for single-digit hours; 24-hour clock. For example, 9 A.M. appears as 09:00. 1 P.M. appears as 13:00.

mm - Minutes with leading zero for single-digit minutes.

ss - Seconds with leading zero for single-digit seconds.

Alphanumeric - This option lets you enter characters such as letters and decimal digits.

File name - This type lets you enter the file name based on your computer's operating system.

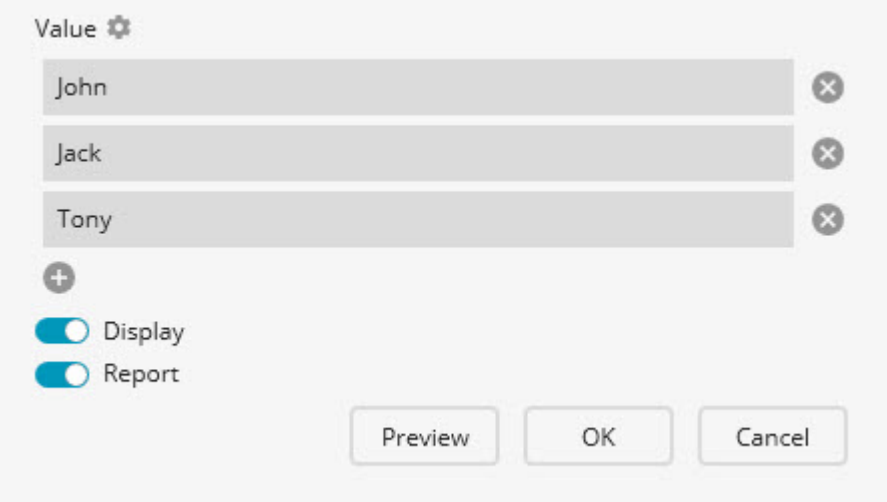
In the Windows 10 operating system, the file name does not allow characters such as \, /, :, *, ?, ", <, >, and |.

Mandatory - This switch determines whether the operator can skip the trace during execution. If you turn on the switch, the operator cannot skip the trace during execution and must type a value.

Convert to uppercase - This switch converts the value characters into uppercase.

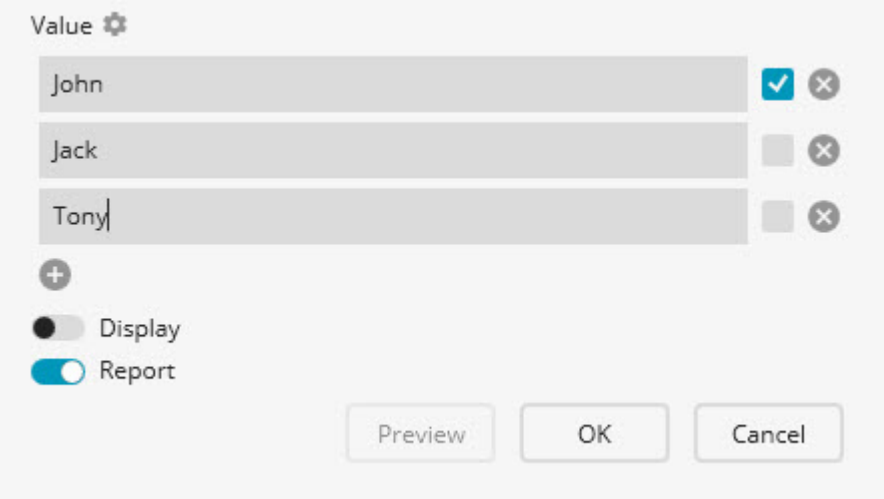
You can have a single value, or you can have multiple rows of values. To add a row for a value, click the **Add** button .

- If the **Display** switch is on, PC-DMIS adds a **Remove** button ( ) for each row.



During execution, the value in the first row becomes the selected value. You can change the value from the list when PC-DMIS displays the dialog box during execution. Otherwise, PC-DMIS sends the selected value to the statistical database.

- If the **Display** switch is off, PC-DMIS adds a check box and **Remove** button ( ) for each row.



To set a value as the selected value, mark the check box next to the value. During execution, PC-DMIS displays this value as the selected value.

To hide the dialog box during execution, and send the selected value to the statistical database, you can turn off the **Display** switch in the **Trace Field** dialog box.

To remove a row for a value, click the **Remove** button .

Display - This switch shows or hides the **Trace Field** dialog box during execution. If you hide the **Trace Field** dialog box, PC-DMIS doesn't display the dialog box for the operator to type or select the value. PC-DMIS sends the current value to the statistical software.

Report - This switch shows or hides the name of the trace and its value in the measurement report.

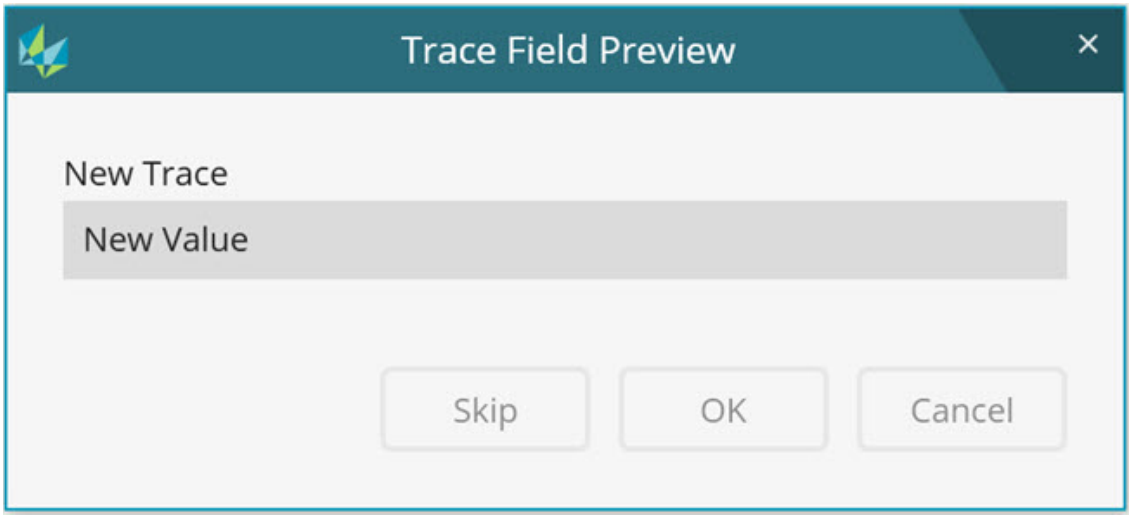
- If the tracefield name is Operator, the value is Sam, and the source is None, then the report shows:

```
TRACEFIELD Operator = Sam
```

- If the trace name is K0008, the value is Sam, and the source is Q-DAS, then the report shows:

```
TRACEFIELD Source = Q-DAS, K0008 = Sam
```

Preview - This button lets you preview the tracefield in a **Trace Field Preview** dialog box.



Trace Field Preview Dialog Box

The preview dialog box mimics the actual **Trace Field** dialog box that PC-DMIS shows during execution. Click × to close the preview.

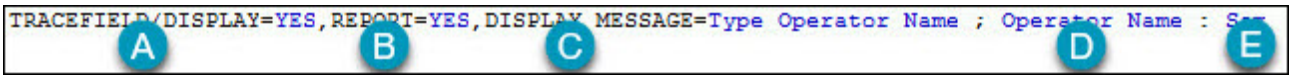
You can see the `TRACEFIELD` command in these Edit window code snippets:

```
TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
MESSAGE=Type Operator name ; Operator : Sam

TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
MESSAGE=Batch number / Ident number ; K0006 : 4
```

TRACEFIELD Command in the Edit window

You can also use the Edit window to change the tracefield's name, display message, and current value.

The Edit window command line for a TRACEFIELD command looks like this:



Example showing a TRACEFIELD command.

A. **Display** - This toggle field controls whether to display the **Trace Field** dialog box during execution. This field switches between YES and NO.

B. **Report** - This toggle field controls whether to display the name of the trace and its value in the measurement report. This field switches between YES and NO.

C. **Display Message** - This field holds the message that appears for the tracefield during execution.

D. **Name -** This field holds the name of the tracefield.

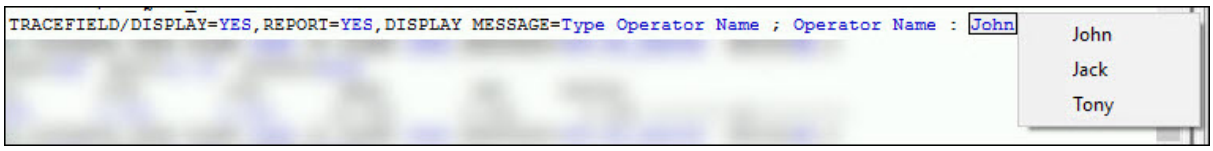
E. **Value -** This field holds the current value of the tracefield.

If you select **Source** as **None**, you can type or edit the trace name, display message, and value.

If you select **Source** as **Q-DAS**, the **Name** becomes a drop-down list. From the drop-down list, select the trace name.

If you have one row with a single value, you can type or edit the value.

If you have multiple rows each with its own value, you can select a value from a list of available values.



Using Variables

You can use variables in a `TRACEFIELD` command's fields under these conditions:

- You must define the variables that you want to use earlier in the routine above any `TRACEFIELD` commands.
- You must type the variable names into the corresponding fields in the `TRACEFIELD` command in the Edit window or in the dialog box.

You can type a variable into all the fields (name, message, or value) if **Source** is set to **None** in the dialog box. If **Source** is set to **Q-DAS**, you can type variables only in the value

field. Also, you can type a variable for a value only if the `TRACEFIELD` command contains a single value.

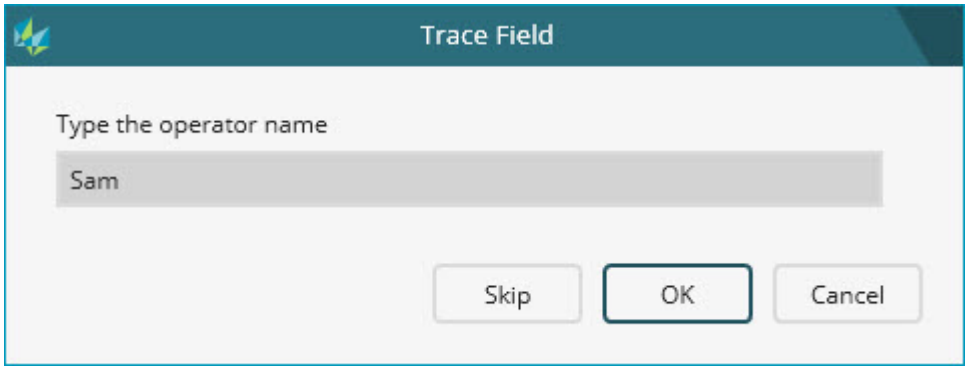
Use the GETTRACEVALUE() function to read a value from a `TRACEFIELD` command in the measurement routine. For information on this and all string functions, see "String Functions" in this documentation.

You can see the `TRACEFIELD` command after you set **Source** to **None** in the dialog box:

```
ASSIGN/V1="OPERATOR"
ASSIGN/V2="TYPE THE OPERATOR NAME"
ASSIGN/V3="Sam"
TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY MESSAGE=V2 ;
V1 : V3
```

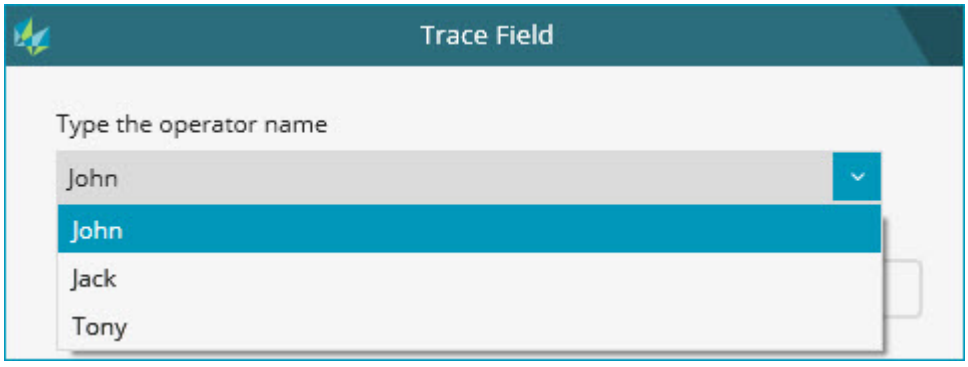
The Trace Field Dialog Box During Execution

For a single value, the **Trace Field** dialog box displays a box to type or edit a value:



Example of the Trace Field dialog box with a single tracefield value.

For multiple values, the **Trace Field** dialog box displays a list to select a value:



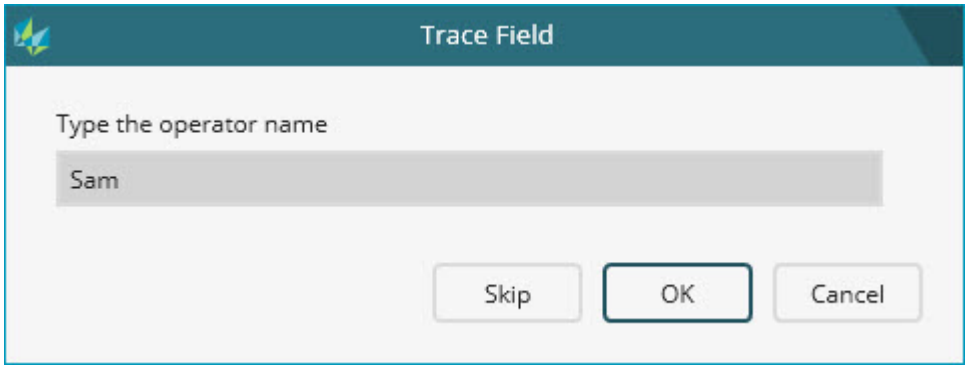
Example of the Trace Field dialog box with multiple tracefield values.

Execution Example - A Single TRACEFIELD Command

```
LIN1        =FEAT/LINE,CARTESIAN,UNBOUNDED
             THEO/<4.113,0,-6.449>,<1,0,0>
             ACTL/<4.113,0,-6.449>,<1,0,0>
             MEAS/LINE,2,ZPLUS
               HIT/BASIC,NORMAL,<4.113,0,-5.307>,<0,-1,0>,<4.113,0,-5.307>,USE THEO=YES
               HIT/BASIC,NORMAL,<87.972,0,-7.592>,<0,-1,0>,<87.972,0,-7.592>,USE THEO=YES
             ENDMEAS/
             TRACEFIELD/DISPLAY=YES,REPORT=NC,DISPLAY MESSAGE=Type the operator name ; Operator : Sam
LIN2        =FEAT/LINE,CARTESIAN,UNBOUNDED
             THEO/<0,4.366,-7.211>,<0,1,0>
             ACTL/<0,4.366,-7.211>,<0,1,0>
             MEAS/LINE,2,ZPLUS
               HIT/BASIC,NORMAL,<0,4.366,-7.302>,<-1,0,0>,<0,4.366,-7.302>,USE THEO=YES
               HIT/BASIC,NORMAL,<0,57.394,-7.12>,<-1,0,0>,<0,57.394,-7.12>,USE THEO=YES
             ENDMEAS/
```

Example showing a single tracefield command.

During execution, PC-DMIS displays a single `TRACEFIELD` command with the usual **Trace Field** dialog box:



Example of the Trace Field dialog box with a single tracefield value.

Skip - This button skips the tracefield and does not use the value. If the tracefield is mandatory, this button is not available.

OK - This button accepts the values for any executed tracefields.

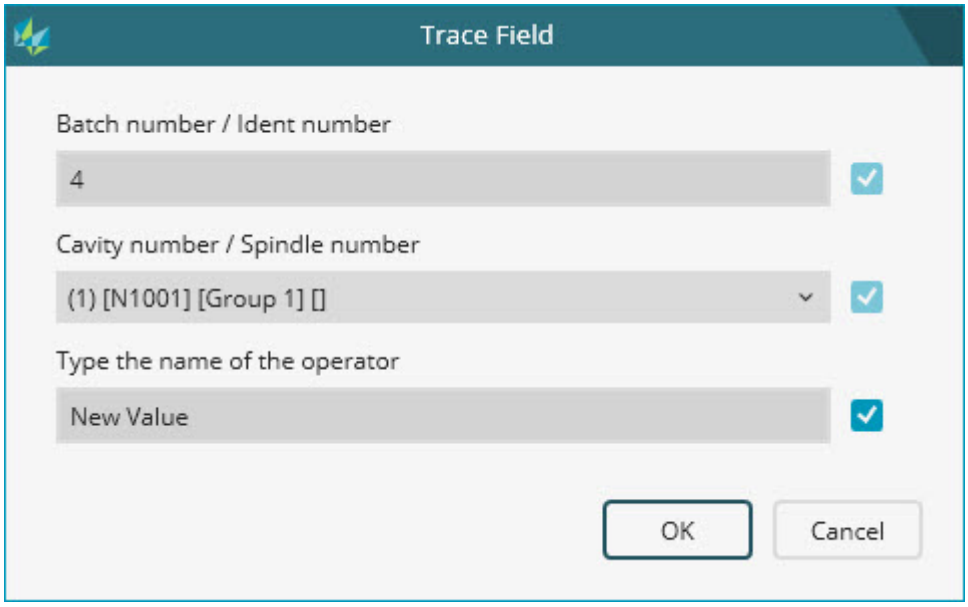
Cancel - This button cancels the execution of the PC-DMIS measurement routine.

Execution Example - Multiple Consecutive TRACEFIELD Commands

```
PLN1        =FEAT/PLANE,CARTESIAN,TRIANGLE
             THEO/<42.901,28.561,0>,<0,0,1>
             ACTL/<42.901,28.561,0>,<0,0,1>
             MEAS/PLANE,4
               HIT/BASIC,NORMAL,<77.785,5.533,0>,<0,0,1>,<77.785,5.533,0>,USE THEO=YES
               HIT/BASIC,NORMAL,<80.143,57.7,0>,<0,0,1>,<80.143,57.7,0>,USE THEO=YES
               HIT/BASIC,NORMAL,<9.926,48.571,0>,<0,0,1>,<9.926,48.571,0>,USE THEO=YES
               HIT/BASIC,NORMAL,<3.751,2.439,0>,<0,0,1>,<3.751,2.439,0>,USE THEO=YES
             ENDMEAS/
             TRACEFIELD/DISPLAY=YES,REPORT=NC,DISPLAY MESSAGE=Batch Number ; K0006 : 4
             TRACEFIELD/DISPLAY=YES,REPORT=NC,DISPLAY MESSAGE=Cavity Number ; K0007 : (1) [N1001] [Group 1]
             TRACEFIELD/DISPLAY=YES,REPORT=NC,DISPLAY MESSAGE=Type the name of the operator ; Operator : New Value
COP1        =COP/DATA,TOTAL SIZE=0,REDUCED SIZE=0,
             REF=,
COPFCOLORMAP1=COP/OPER,SURFACE COLORMAP,SHOW PARAMETERS=NO
             SIZE=0,REF=COP1,,
COPFCOLORMAP2=COP/OPER,SURFACE COLORMAP,SHOW PARAMETERS=NO
             SIZE=0,REF=COP1,,
```

Example showing three consecutive TRACEFIELD commands.

During execution, PC-DMIS combines multiple consecutive commands so that they appear in a single **Trace Field** dialog box. This combination of commands can save you time because PC-DMIS collects the information from a single dialog box instead of a dialog box for each command:



Example of the Trace Field dialog box with multiple values for consecutive tracefields.

Check Box - The blue check boxes allow you to include or skip a tracefield during execution.

- If you select any check boxes, PC-DMIS executes those tracefields.
- If you clear any check boxes, PC-DMIS skips those tracefields and does not use those values. If a tracefield is mandatory, the check box is not available for that tracefield.

OK - This accepts the values for any executed tracefields.

Inserting Q-DAS Tracefields

Select **Insert | Statistics Command | Q-DAS Tracefields** to insert the Q-DAS tracefields. This option inserts the tracefields for K-fields whose "**Value From**" properties are set to "**TRACE**" in the **Setup K-Field Configuration** window of the Q-DAS Configurator.

Q-DAS tracefields help you create measurement routines that support Q-DAS statistical software. For information on the Q-DAS Configurator, see the Q-DAS Configurator documentation.

Before you insert Q-DAS tracefields, you need to create a `STATS/ON,Q-DAS,DFQ` command. For information, see "Sending Statistics to a Q-DAS Database".

After you insert Q-DAS tracefields, if you update the Q-DAS configuration file (<name>.qdascfg.xml), you must re-insert the Q-DAS tracefields.

Here's an example of some Q-DAS tracefields in the Edit window:

```
QDAS_TRACEFIELDS=GROUP/SHOWALLPARAMS=No

EXECUTION CONTROL=AS MARKED

ENDGROUP/ID=QDAS_TRACEFIELDS
```

To expand the group, you can change `SHOWALLPARAMS` from No to Yes as shown in this code snippet:

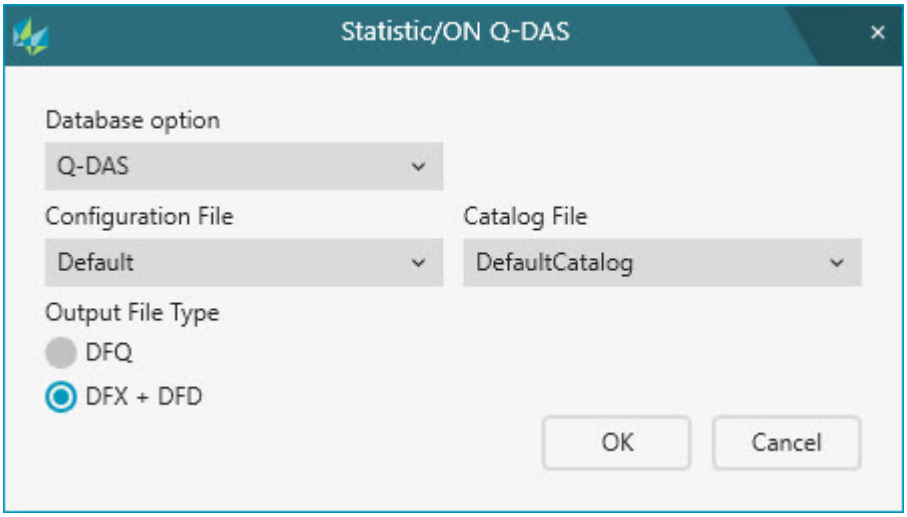
```
QDAS_TRACEFIELDS=GROUP/SHOWALLPARAMS=YES

EXECUTION CONTROL=AS MARKED

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Batch Number ; K0006 :

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Cavity Number ; K0007 : (1) [N1001] [Group
    1]

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Operator Name ; K0008 : (1) [P001] [Operator
    1]

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Machine Number ; K0010 : (1) [M001] [Machine
    1]

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Gage Number ; K0012 : (1) [PM001] [CMM 1]
    [CMM Grp. 1]

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Part ID Number ; K0014 :

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Order ; K0053 :

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Part Number ; K1001 :

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Contract ; K1053 :

    TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
    MESSAGE=Work Cycle / Operation Number ; K1086 :
```

```
                TRACEFIELD/DISPLAY=YES,REPORT=NO,DISPLAY
               MESSAGE=Remark ; K1900 :

          ENDGROUP/ID=QDAS_TRACEFIELDS
```

Sending Statistics to a Q-DAS Database

The **Statistic/ON Q-DAS** dialog box lets you send the measurement data to the Q-DAS statistical software package.

The **Insert | Statistics Command | Statistics/ON, Q-DAS** menu option opens the **Statistic/ON Q-DAS** dialog box.



Statistic/ON Q-DAS Dialog Box

Database option - This option generates a file that the Q-DAS statistical software can read. The list only displays Q-DAS.

Configuration File - This file defines the type of measurement data and the additional information that PC-DMIS needs to collect during the execution of a measurement routine. From the list, select the configuration file. PC-DMIS provides a Default.Qdascfg.xml file.

You can use Q-DAS Configurator to create a configuration file. For more information on Q-DAS Configurator, see the Q-DAS Configurator help file.

You can find the configuration files in this folder:

C:\Users\Public\Documents\Hexagon\PC-DMIS\<version>\Q-DAS\Settings

Where <version> is the PC-DMIS version.

Catalog File - Q-DAS statistical software uses a catalog file to define information that is related to your manufacturing environment. This file is a text file that contains administrative and other data that K-fields can reference. Your organization creates this

catalog according to the requirements of the Q-DAS software and provides it for use. This file is location specific. For more information on the catalog file, see the Q-DAS Configurator help file. From the list, select the catalog file. PC-DMIS provides a DefaultCatalog.dfd file.

You can find the catalog files in this folder:

C:\Users\Public\Documents\Hexagon\PC-DMIS\<version>\Q-DAS\Catalogs

Where <version> is the PC-DMIS version.

Q-DAS Configurator creates a Q-DAS data file for each measurement routine. The Q-DAS data file contains information that Q-DAS statistical software needs to generate output files. Q-DAS Configurator saves the Q-DAS data file in the same folder as the measurement routine. The file name has the following format:

<name of measurement routine>.QdasData.xml

Where <name of measurement routine> is the name of the PC-DMIS measurement routine.

If a QdasData.xml file exists for the current measurement routine, then the **Configuration File** and **Catalog File** lists display the file names from the QdasData.xml file. You cannot change the file names from this dialog box.

To select the different configuration file or catalog file, or both, you need to use Q-DAS Configurator. For more information on Q-DAS Configurator, see the Q-DAS Configurator help file.

Output File Type - Q-DAS statistical software supports the Q-DAS ASCII Transfer Format. This option lists the formats that Q-DAS supports:

DFQ - This option sends the output to a .dfq file.

DFX + **DFD** - This option sends the output to .dfd and .dfx files.

Select the required output format.

OK - This button inserts the `STATS/ON,Q-DAS,DFX+DFD` or `STATS/ON,Q-DAS,DFQ` command into the Edit window.

Cancel - This button discards the changes and closes the dialog box. It does not insert the `STATS/ON,Q-DAS,DFX+DFD` or `STATS/ON,Q-DAS,DFQ` command into the Edit window.

Sending Current Stats to a File

The functionality of the **Operation | Send Current Stats To File** menu option depends on the statistical package you use, as described below.

If the `STATS/ON` command uses DataPage or DataPage+, PC-DMIS shows a **Save Statistics** dialog box asking if you want to save your statistical data.

- Click **No** to cancel the operation, no data is saved.
- If you click **Yes**, PC-DMIS shows an **Update** dialog box asking if you want to update the DataPage+ database.
- If you click **Yes** on this second dialog box, PC-DMIS updates the database.
- If you select **No**:
  - If the `STAT/ON` command uses DataPage, PC-DMIS creates an "XSTAT11.tmp" file.
  - If the `STAT/ON` command uses DataPage+, PC-DMIS creates an XML file.

If the `STATS/ON` command uses Q-DAS, PC-DMIS shows a **Save Statistics** dialog box to confirm that you want to save your statistical data or not.

If the current measurement routine was the last measurement routine you executed, then PC-DMIS creates the DFQ/DFX+DFD files. If not, you may not have the correct data set for this measurement routine. If the `STATS|ON` command is set to Q-DAS, to ensure your data set is correct before you save it, run the measurement routine and then select the **Operation | Send Current Stats to File** menu option. You can find the files in the "C:\Users\Public\Documents\Hexagon\PC-DMIS\Q-DAS\Results" folder. If you have not executed your measurement routine, PC-DMIS does not create DFQ/DFX+DFD files.

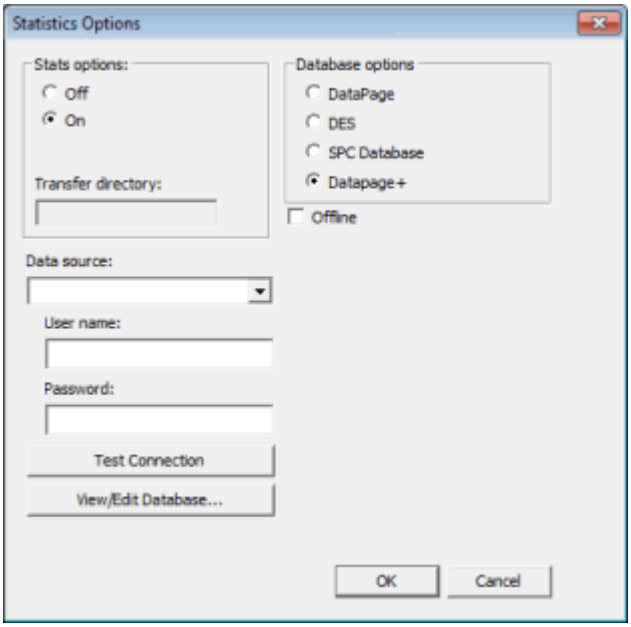
For details on sending statistical data to a Q-DAS database, see the "Sending Statistics to a Q-DAS Database" topic in the PC-DMIS Core documentation.

If you want to use your XSTATS11.tmp file with Q-DAS, be sure to execute the measurement routine first so that it sends the generated Q-DAS data to the .tmp file.

If you select **Send Current Stats To File**, PC-DMIS immediately sends all of the dimensions and tracefields that follow a `STATS/ON` command to the selected database. Statistics only get sent to a database if you have a database registered and have the `STATS/ON` command marked. With the `STATS/ON` command set to Datapage, PC-DMIS also sends statistical data to the XSTATS11.tmp file.

For details on the locations of PC-DMIS files, see "Understanding File Locations".

Using the Statistics Options Dialog Box



Statistics Options Dialog Box

The **Insert | Statistics Command | Statistics** menu option displays the **Statistics Options** dialog box. Use this dialog box to tell PC-DMIS what it should do with the results of a measurement. You can tell it to send that information to a specific statistical software package database.

Under **Stats options**, if you select **On** and click **OK**, it inserts a `STATS/ON` command into the measurement routine. If `STATS/ON` is inserted before any desired dimension or tracefield commands, then that dimension or tracefield information is sent to the selected statistical software package.

The dialog box changes depending on the database option selected from the **Database Options** area. For information on the changeable areas of the dialog box, see the items discussed in the **Database Options** sub-topics.

Stats Options

The **Stats Options** area is always available on the **Statistics Options** dialog box. The **Stats Options** area allows you to use the following commands.

- Off - Turning off statistical gathering
- On - Turning on statistical gathering
- Transfer - Transferring a stats file to a new directory

- Update - Updating the database
- Record - Recording specific data

Off - Turning Off Statistical Information (STATS/OFF)

Selecting the **Off** option stops statistical information from being sent from your measurement routine by inserting a `STATS/OFF` command. With stats off, Dimension or tracefield commands will *not* send statistical information to the database unless you select the Stats On option.

On - Turning on Statistical Gathering (STATS/ON)

Selecting the **On** option allows statistical information to go from your measurement routine to the database. With stats on, you can send data to the stats file, transfer the information to a different directory, or store the information in a database application.

Any dimensions that follow a `STATS/ON` command and precede the next `STATS/OFF` command are sent to the stats file or database upon execution of the next `STATS/TRANSFER`, `STATS/UPDATE`, or final command of the measurement routine. With the `STATS/ON` command, you can send the output information to multiple databases, even within the same measurement routine.

Upon each execution of a measurement routine that includes at least one `STATS/ON` command, PC-DMIS asks if it should send the output information to any statistical database directories. This prompt can be bypassed by selecting the **ALWAYS Update Database** check box accessible from the **Dimension** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**). For information on the **Dimension** tab, see the "Setup Options: Dimension tab" topic in the "Setting Your Preferences" chapter.

If you select **DataPage** from the **Database Options** area, PC-DMIS automatically calls the DPUPDATE.exe executable to insert the data in the XSTATS11.tmp file into any selected DataPage directories. You must run PC-DMIS with administrator privileges for DPUPDATE.exe to work.

Similarly, selecting the **DataPage+** option calls DPPLUSUPDATE.exe which in turn calls DataImporter to import the XML file into the DataPage+ database.

Turning stats on inside a loop can result in a slightly different output for the dimension and feature IDs in generated stats files such as the XSTATS11.tmp text file or the XML file.

For details on the locations of PC-DMIS files, see "Understanding File Locations".

The Edit window command line for a `STATS/ON` command block changes based on the specified database type. All `STATS/ON` commands start with this on the initial line: `STATS/ON,database option;`

database option = This toggle field indicates the database type to which the stats will be sent. This field can be DATAPAGE, DES, SPC_DATABASE, or DATAPAGE+. Depending on what you select here, the rest of the command can change.

With the DATAPAGE option, the command block looks like this:

```
STATS/ON,DATAPAGE,$

DIRECTORY=directory_name,$

READ=n1,WRITE=n2,MEMPAGES=n3,variable_name1,variable
_name2,$

STATS/END
```

With the SPC option, the command block looks like this:

```
STATS/ON,SPC_DATABASE,database_name
```

With the DES option, the command block looks like this:

```
STATS/ON,DES,DIRECTORY=directory
```

With the DATAPAGE+ option, the command block looks like this:

```
STATS/ON,DATAPAGE+,database_name,OFFLINE=y/n

RECORD=data_list
```

DIRECTORY=directory_name If the database option is DATAPAGE or DES, the value of **directory_name** should be a string that indicates the database directory to which the statistics will be sent. If you have more than one database directory, this DIRECTORY field repeats itself for each additional directory. Each one must be a valid database directory.

database_name = If the database option is SPC_DATABASE or DATAPAGE+, this string indicates the Data Source Name (DSN) of the database. For an SPC database, this name can be created in the ODBC options of the computer's control panel. For a DATAPAGE+ database, the DSN is usually created inside of the DATAPAGE+ tool. The DSN must be created to link to a valid database.

READ=n1 This is a read lock. **n1** indicates the maximum number of seconds PC-DMIS will wait for database access when reading. The default setting is 10 seconds. This field is only used for the DataPage database option.

WRITE=n2 This is a write lock. **n2** indicates the maximum number of seconds PC-DMIS will wait for database access when writing. The default setting is 20 seconds. This field is only used for the DataPage database option.

MEMPAGES=n3 This stands for memory pages. **n3** indicates the number of 4K memory pages to reserve for database tables when opening the database. This option will affect performance. The minimum number that can be entered is 4 (also default). This field is only used for the DataPage database option.

variable_name1 = This field indicates the variable name setting. If set to DIMENSION_NAME then dimension names will be displayed. If set to FEATURE-NAME then feature IDs will be displayed. This field is only used with the DataPage database option.

variable_name2 = This field determines whether or not control calculations should be used. If set to CONTROLCALC ON then control calculations are used. If set to CONTROLCALC OFF then they are not used.

OFFLINE = This YES/NO field determines if the offline options should be used. If set to YES, then the `RECORD=` field appears, specifying the info to write to the XML file. This field is only used with the DataPage+ database option.

RECORD=data_list This determines what data to record to the XML file. The value of **data_list** is either ALL, meaning all data is sent to the XML file, or a comma-separated listing of items. For example, if you wanted just dimension and hit information recorded, you would choose DIMENSION, HIT. This field is only used with the DataPage+ database option, and it is only available if `OFFLINE=YES` .

PC-DMIS and DataPage+ versions that support STATS/ON command

For information on supported methods and versions, see "Table of Supported Methods".

DataPage+ XML Handling

If you're using `STATS/ON` with a DataPage+ database name and `OFFLINE=YES` , a generated XML is created in the C:\Users\Public\Documents\Hexagon\PC-DMIS\ directory immediately after execution finishes. *Note that this directory is version independent.* It contains the transaction date and time, the DSN, as well as other information. You can limit this other information by using the **Record** option with the **Offline** check box.

Once generated, dpplusupdate.exe moves the file into the %AppData%/DataPage+/ForStatsFile directory for file handling. The DataImporter tool, if running, then looks in that directory and automatically imports the data into DataPage+. For information on using DataImporter, see the documentation that came with your installation of DataPage+.

The name of the XML is based on the current local time when it was generated. It uses this naming format:

<YYYY><MM><dd><hh><mm><ss>.xml

<YYYY> - The four digit year

<MM> - The two digit month

<dd> - The two digit day of the month

<hh> - The hour in 24-hour format

<mm> - The minutes

<ss> - The seconds

Transfer - Transferring a Stats File to a New Directory (STATS/TRANSFER)

The **Transfer** option allows statistical information from a measurement routine to be transferred to a specified directory upon execution of the `STATS/TRANSFER` command.

This option may be especially useful with the **Auto Update** option of DataPage or with an XML file generated from the **Record** option of the **DataPage+** database option. See "Record".

To transfer your stats to a different directory:

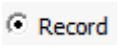
1. Click the **Transfer** option.
2. In the **Transfer Directory** box, enter the directory path of the directory to which you will transfer your stats file.

Update - Updating the Database (STATS/UPDATE)

The **Update** option creates a `STATS/UPDATE` command that tells PC-DMIS to update the statistical database every time the command is executed. The statistical software application must be installed and ON.

The **Always Update Database** check box from the **Dimension** tab of the **Setup Options** dialog box allows you to bypass the "Update Database Now?" message that PC-DMIS shows at the end of a measurement routine execution that contains a `STATS/ON` command. See the "Setup Options: Dimension tab" in the "Setting Your Preferences" chapter.

Record - Recording Specific Data (STATS/RECORD)



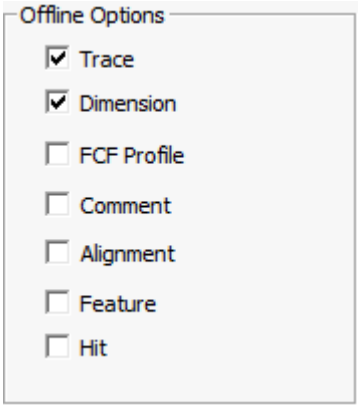
The original **DataPage+** output sends a lot of information to the database. This may bog down execution of some measurement routines. Some information may not even be desired.

The **Record** option in the **Stats options** area provides greater flexibility and speed for data output. It must follow a STATS/ON command to function. It does the following:

- It sends data by means of a dpplusupdate.exe utility that runs in the background. This means an active connection to a DataPage+ database is not used. In fact, you don't need to have a database defined at all.
- It writes data to an XML file.
- It lets you selectively target what data you want captured by editing the `STATS/ON` or `STATS/RECORD` commands to include the data you want the software to output to the XML file. You can do this using the **Offline** check box and the **Offline Options** area covered below.

How Record and the Offline Check Box Work

After the **Offline** check box is marked, the **Offline Options** area becomes available for edit to choose the specific items to record to the XML output.



Even if you don't have the **Record** option specifically selected, if **Offline** is marked and you click **OK**, the Edit window `STATS/ON` command is modified with a `RECORD` command to include the selected options.

For example, if you marked **Offline** and chose **Trace** and **Dimension**, your command would look something like this:

```
STATS/ON,DATAPAGE+,DATAPAGE_PLUS,OFFLINE=YES
RECORD=DIMENSION,TRACE,
```

This means during execution PC-DMIS would only send the **Dimension** and **Trace** information to the XML file until another `RECORD` command was encountered that told it to send something different or until it reached a `STATS/OFF` command.

Now, suppose later in the routine you chose the **Record** option and you wanted to include **Hit** information as well. The command would be this:

```
STATS/RECORD,DIMENSION,TRACE,HIT
```

This command, as long as it's following a `STATS/ON` command, would then limit the data sent to the XML file include trace, dimension, and hit information.

Database Options

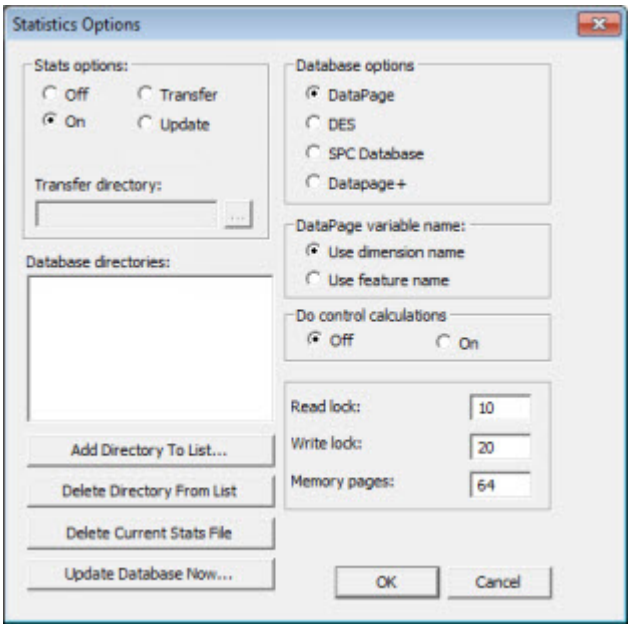
The options in the Database Options area determine which of these statistical software packages receives the measurement information:

- DataPage
- DES
- SPC Database
- DataPage+

Once you select an option, the **Statistics Options** dialog box changes to show specific information needed for that option.

The **DataPage+** option is the default option.

DataPage Option

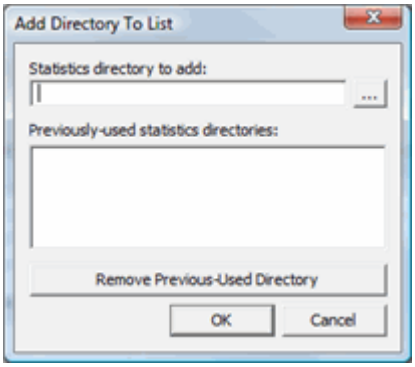


Statistics Options Dialog Box – DataPage Option

When you select the **Datapage** option, the **Statistics Options** dialog box changes to add the following items:

Database Directories - This list shows the different directories in which a database application could reside. You can add or delete directories from this list. A maximum of ten directories per `STATS/ON` command can be selected to which statistical information can be sent.

Add Directory to List - Displays the **Add Directory To List** dialog box that lets you specify a directory containing a database you want to use with DataPage:



Add Directory to List

When the dialog box appears, you can either type the pathway containing the database in the **Statistics directory to add** box, or click the ... button to browse to the directory. Then click **OK**. to add the specified directory into the **Database Directories** list.

Use the **Remove Previous-Used Directory** to remove previously defined directories.

Delete Directory from List - Deletes the selected database directory from the **Database Directories** list.

Delete Current Stats File - Deletes your xstats11.tmp file.

For details on the locations of PC-DMIS files, see "Understanding File Locations".

Update Database Now - Updates your database application with statistical data gained from your current measurement routine. Simply click this button and follow the instructions on screen. This uses DPUPDATE.exe to update the database.

Upon successful execution of a measurement routine, selected Dimension and tracefield information is stored within a file named XSTATS11.tmp. The information in this file can then be transferred to a different directory, printed out, or sent to a database application.

If a database application is unavailable, PC-DMIS displays a message informing you that it can't find the update executable (for the DataPage application, this is DPUPDATE.exe) to load XSTATS11.tmp into the database.

You must run PC-DMIS with administrator privileges for DPUPDATE.exe to work.

DataPage Variable Name - This area indicates whether the dimension name or feature name will be used to identify entries in DataPage. The maximum length of the name is 10 characters. To choose either the dimension name or the feature name, select the **Use Dimension Name** option or the **Use Feature Name** option.

Do Control Calculations - Contains options to indicate whether the control calculations that take place within Datapage should be used. Selecting the **On** option performs the out of tolerance control calculations. Additionally, it commands the DataPage statistics update program (DPUPDATE) to perform the control calculations and set the part/variable and in/out of tolerance/control indicators in the database. You can monitor this database using DataPage's Monitor mode by turning on the part/variables colors (red/yellow/green) based on those calculations. For additional information, see the help file that ships with DataPage's Monitor mode application.

To turn control calculations on or off simply select either the **On** or the **Off** option. The default setting is **Off**.

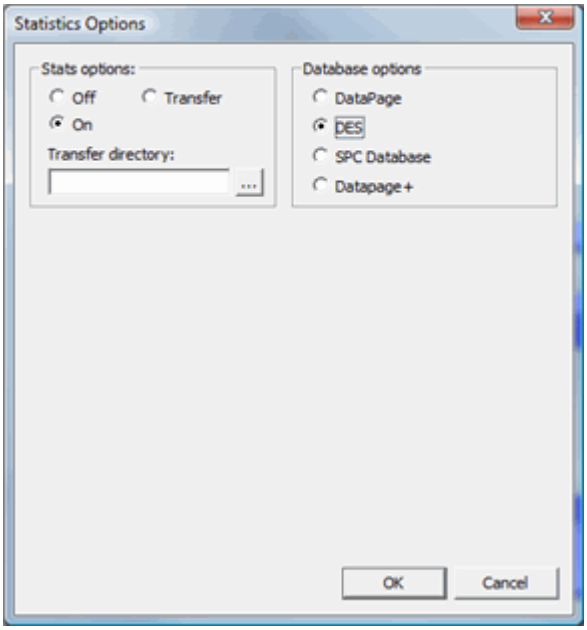
Read Lock - Determines the maximum number of seconds that PC-DMIS will wait for database access when reading. The default value is 10 seconds.

Write Lock - Determines the maximum number of seconds that PC-DMIS will wait for database access when writing. The default value is 20 seconds.

Memory Pages box - Determines the number of 4K pages to reserve for database tables when opening a database. The minimum (and default) number of pages that can be open is four.

DES Option

The DES option only functions with the statistical database program, DES. When you select DES, the **Statistics Options** dialog box changes so that it only includes the **Stats Options** area, the **Database Options** area, and the **OK** and **Cancel** command buttons.

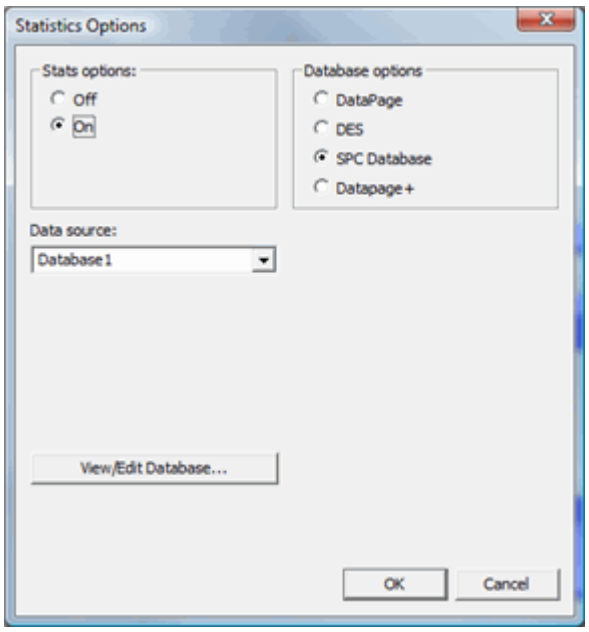


Statistics Options Dialog Box – DES Option

SPC Database Option

The **SPC Database** option only functions with PC-DMIS's own statistical database program, SPC. When you select the **SPC Database** option, the **Statistics Options** dialog changes to include the following:

- **Data source** list
- **View/Edit Database** button



Statistics Options Dialog Box - SPC Database Option

Data source list

This list contains predefined data sources. You can establish a connection between a selected data source and PC-DMIS in order to collect and store statistical data from PC-DMIS measurements results.

There are three ways you can establish a data source:

Use "ODBC Data Sources" from "Administrative Tools"

Use the **Data source** list to select a Data Source Name (DSN). A DSN is created initially by using the Open Database Connectivity (ODBC) options in the computer's Control Panel.

If you need to create a DSN:

1. Access the Windows **Control Panel**.
2. Select **Administrative Tools** (use the **Classic View** if using Vista).
3. Double-click on the **ODBC Data Sources** or **Data Sources (ODBC)** icon.
4. Follow the instructions in your operating system's Help file to create the DSN.

Use the Database Wizard

Click the **Database Wizard** icon from the **Wizards** toolbar inside of PC-DMIS to easily create a database. Simply follow the prompts to create the database. See "Wizards Toolbar" in the "Using Toolbars" chapter.

Use the Database Connection Properties dialog box within DataPage+

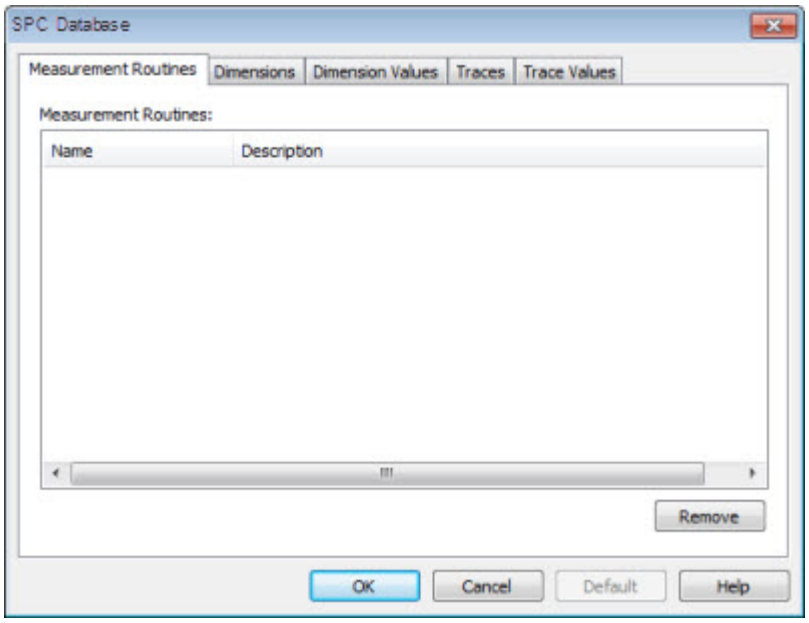
1. From within DataPage+, select **Tools | Database Connection**. The **Database Connection Properties** dialog box appears.

2. Following the documentation available within DataPage+, fill out the dialog box to create the DSN.

By using the ODBC, PC-DMIS can pass data directly to the SPC database, and the data can be viewed within a statistical package that supports the SPC database.

View/Edit Database button

The **View/Edit Database** button can only be used once a valid Data Source has been selected from the **Data Source** box. Clicking this button allows the data in the SPC database to be viewed and deleted from the **SPC Database** dialog box:



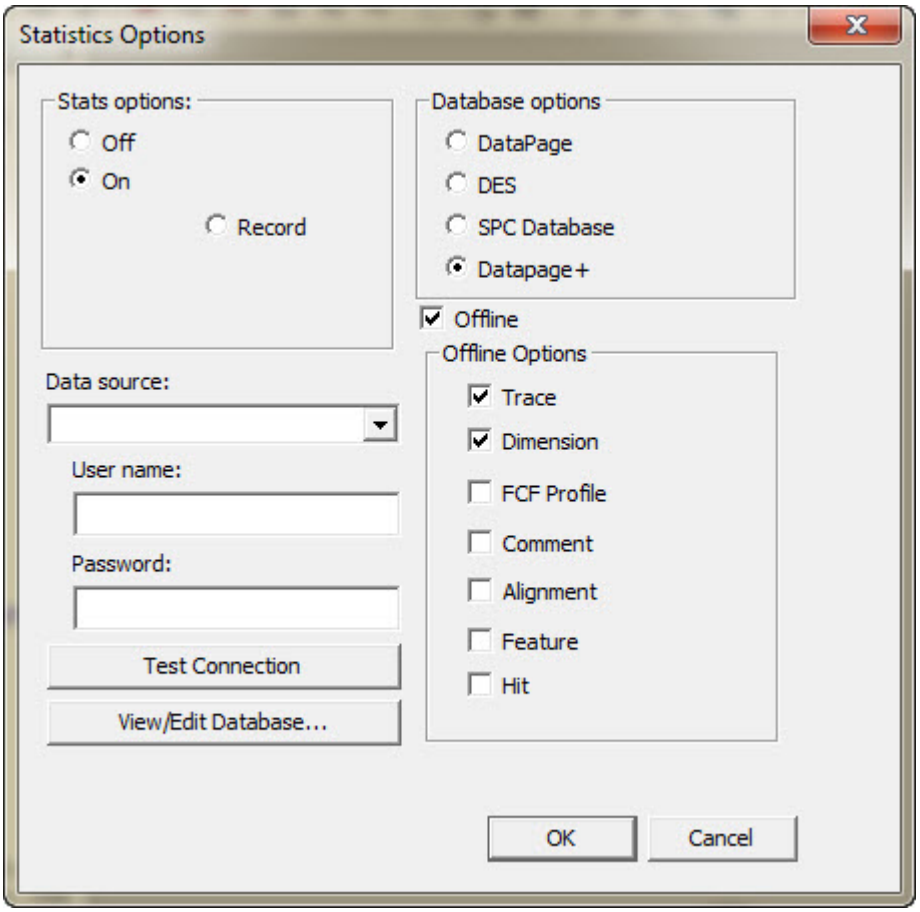
SPC Database Dialog Box

The SPC database can be viewed within PC-DMIS using the following tabs on the dialog box:

- Measurement Routines
- Dimensions
- Dimension Values
- Traces
- Trace Values

You can also view databases using the **SPC Database** menu option.

DataPage+ Option



Statistics Option - DataPage+ Option

If you select **DataPage+** from the **Database options** dialog box, the **Statistics Options** dialog box provides these options so that you can connect to a database used with DataPage+:

Stats options area
The off/on check box allows or stops statistical information to go from your measurement routine to the database. The **Record** option provides greater flexibility and speed for data output. See "Off", "On", and "Record" topics for more information.

Data source list
This list operates the same as the **Data Source** information discussed in the "SPC Database Option" topic.

User name box
In the **User name** box, type a user name for connecting to a password protecteddatabase.

Password box
In the **Password** box, type a password for connecting to a password protecteddatabase.

Test Connection button
This button attempts to establish a connection between PC-DMIS and the specified data source. It will display a success message if the connection successfully connects to the database.

View/Edit Database button
This button operates the same as the **View/Edit Database** information discussed in the table in "SPC Database Option" topic, except that the database is not an SPC database.

Database options area
This area determines which statistical software package receives the measurement information. See " Database Options" for more information.

Offline check box
Mark this check box if you want to send the data to a local DataPage+ database through Offline mode. See "Sending Data to a Local DataPage+ Database" and "Record".

See the "Sending Data to a Local DataPage+ Database" topic for more information.

Sending Data to a Local DataPage+ Database

To send statistical data from PC-DMIS to a local database used in DataPage+, you may need to create a new database.

> Legacy DataPage databases use a different database structure than DataPage+. Therefore, they cannot be used inside DataPage+. To continue building upon your existing data set, you need to perform a dump (or export) of your data and then import that data into a new DataPage+ database using the **Tools | Database Import** menu item in DataPage+. For more information, see the DataPage+ documentation on "Importing Data from an ASCII File".

After you have a database created, if you're using a supported version of PC-DMIS and DataPage+, you can then connect to that database from within your `STATS/ON` command by following the steps below. For information on supported methods and versions, see "Table of Supported Methods".

Step 1: Install .NET Framework v3.5

Before you can send data from PC-DMIS, your computer system must have the .NET Framework v3.5 installed. If needed, you can install it from this link:

http://msdn.microsoft.com/en-us/netframework/default.aspx

Step 2: Create a New DataPage+ Database

If you want to use a new database with DataPage+, you need to use the Database creation tools inside of DataPage+. To do this, start **DataPage+**, and select **Tools | Database connection**. See your DataPage+ documentation for information on creating the new database.

This automatically creates a database and data source for you to connect to from within PC-DMIS. After you have the database created, follow the instructions below to connect to an existing database.

Step 3: Connect to an Existing Data Source and DataPage+ Database

This step describes how to connect a `STATS/ON` command to an existing data source and database.

1. Select **Insert | Statistics Command | Statistics**. The **Statistics Options** dialog box appears.
2. From the **Database options** area, select **DataPage+**.
3. From the **Data source** list, select the name of your data source. If needed, specify a user name and password in the appropriate boxes. You can also send data offline by using the **Offline** check box. In that case, you don't need to establish a connection to the database. For more information, see "Sending DataPage+ Info without a Database Connection (Offline)" below.
4. Click **Test Connection**. If you have set things up correctly, a "Succeeded" message appears.

   The demo database that ships with DataPage+ is only for viewing purposes. You should not use it to send data from PC-DMIS.

5. Click **OK** to close the **Statistics Options** dialog box. Now, your `STATS/ON` command should send statistical data to the specified data source. Your final `STATS/ON` command might look something like this:

   ```
   STATS/ON,DataPage+,dpplus_database
   ```

   This connects to an existing database named dpplus_database.

Sending DataPage+ Info without a Database Connection (Offline)

See the "Record" topic.

Importing XML Data into your DataPage+ Database

You can also import statistical data from XML files into the DataPage+ database. You can send statistical data to XML files with these methods:

- Using DataPageStats.exe

- Using XMLSTATS/ON

You can then use a utility called DataImporter.exe (also in the DataPage+ installation directory). DataImporter.exe checks the directory that contains the generated XML files. It can then pull the data from the .xml files into the database.

For more information on DataImporter.exe, see "Using DataImporter.exe" in the DataPage+ documentation.

For more in-depth information, see the ReadMe.doc file that ships with DataPage+ and also the "XML Statistics Tools" section of the DataPage+ documentation.

Sending Statistics to an XML file using DataPageStats.exe

You can use the DataPageStats.exe utility in the DataPage+ installation directory to send statistics to an XML file. In your measurement routine, add an `External` command that points to this .exe file. Place the command either at the beginning or at the end of the measurement routine, depending on your requirements:

- If you want to receive all the data together, you should place the command at the end of your measurement routine.
- If you want to receive the data after the `External` command finishes execution, insert the command with a **-listening** parameter at the beginning of your measurement routine. For more information and examples, see "Using DataPageStats.exe" in the DataPage+ documentation.

The DataPageStats.exe utility creates an .xml file with the statistical data with each execution.

For information on supported methods and versions, see "Table of Supported Methods".

Sending Statistics to an XML File using XMLSTATS/ON

You can also send your statistical data to an XML file by using the `XMLSTATS/ON` command. This gives you access to the statistical data in a simple text format that you can then manually parse for use with third party tools. For information on supported methods and versions, see "Table of Supported Methods".

`XMLSTATS/ON` replaces the previous XML automation export utility named DataPageStats.exe that ships with DataPage+. You can then use the PcdmisXMLStatsToDatabase.exe utility in DataPage+ to monitor a directory and pull the data from the .xml files into the database. For information on how to do this, see

DataPage+ documentation. For information on how this differs from DataPageStats.exe, see "How XML STAT ON Differs from DataPageStats.Exe" below.

Using XMLSTATS/ON

To insert the `XMLSTATS/ON` command, type "XMLSTATS" and then press the Tab key on your keyboard in the Edit window's Command Mode; or select the **Insert | Statistics Command | XML Statistics** menu item. This command has the following syntax:

```
XMLSTATS/<TOG1>,<Directory>
```

<TOG1> - This field toggles between either ON or OFF, letting you choose whether or not to export statistical data for measurement commands that follow this command.

<Directory> - This field takes a string (without quotes) as the directory pathway in which the file will be created. The default pathway is the directory where you store your measurement routines.

`XMLSTATS/ON` only writes the XML during execution. It writes the file to the specified directory and automatically generates the .xml file's name using this format:

<Measurement Routine>_<Day>-<Month>-

-___
.XML

<Measurement Routine> - The measurement routine name without the PRG extension.

<Day> - The name of day the measurement routine was executed in local time.

<Month> - The name of the month the measurement routine was executed in local time.

<dd> - The day of the month in two digit number format in local time.

<YYYY> - The year the measurement routine was executed displayed in four digit format in local time.

<hh> - The hour the measurement routine was executed (in 24-hour local time).

<mm> - The minute the measurement routine was executed in local time.

<ss> - The second the measurement routine was executed in local time.

Thus, a measurement routine named "Test.prg" executed on August 12, 2010 at exactly 2:40:15 PM local time, would have this file name:
Test_Thursday_August-12-2010_14_40_15.XML

The XML file contains data for all commands that follow the `XMLSTATS/ON` until execution finishes or until execution encounters an `XMLSTATS/OFF` command. If execution encounters another `XMLSTATS/ON` command while writing to a file, it will write

a closing `</ExecutionTransaction>` tag in the XML file and then allow the next `XMLSTATS/ON` command to write another opening `<ExecutionTransaction>` tag.

For example, suppose you had this pseudo measurement routine:

```
XMLStats/ON
Dimension1
XMLStats/OFF
Dimension2
XMLStats/ON
Dimension3
```

Following execution, PC-DMIS would generate a single file with two `<ExecutionTransaction>` tags. The first tag would contain Dimension1:

```
<ExecutionTransaction DateTime=”Sometimestamp”>
   <XMLStats command data>
   <Dimension1 Data>
</ExecutionTransaction>
```

The second file generated would contain Dimension3:

```
<ExecutionTransaction DateTime=”Sometimestamp”>
   <XMLStats command data>
   <Dimension3 Data>
</ExecutionTransaction>
```

How XML STATS/ON Differs from DataPageStats.Exe:

- `XMLSTATS/ON` is faster than DataPageStats.exe.
- It outputs everything DataPageStats.exe did but includes more.
- You do not need to insert an external command at the end of your measurement routine as you did when using DataPageStats.exe.
- You can choose what data gets sent to the XML file by turning `XMLSTATS` commands on and off as needed.
- `XMLSTATS` adds a time stamp within each `<ExecutionTransaction>` tag pair. In addition, the filename itself, as discussed above, is comprised of the part name and the time of execution.

Table of Supported Methods

To import data from PC-DMIS, you can use DataPageStats.exe with any version of PC-DMIS and DataPage+. If you want to use `STATS/ON` command, consider the table below:

DataPage+	PC-DMIS	Supported Methods
Version 5.0 and earlier	Versions earlier than 2012 MR1	DataPageStats.exe
Version 2012 MR1 and later	DataPageStats.exe STATS/ON XMLSTATS/ON	
Version 5.1 and later	Versions earlier than 2012 MR1	DataPageStats.exe
	Version 2012 MR1 to 2015.0	DataPageStats.exe XMLSTATS/ON
	Version 2015.1 and later	DataPageStats.exe STATS/ON XMLSTATS/ON

Geometric Tolerance commands only support the STATS/ON method for statistical output to Datapage+ or Q-DAS.

Reporting Measurement Results

Reporting Measurement Results: Overview

For tutorials in your language, please contact your regional office or headquarters.

After you measure your part, it's important to be able to communicate your measurement results with others. PC-DMIS, by default, sends your measurement data to a standard textual report, called an inspection report. It includes comprehensive data about each feature measured by your measurement routine. You can send this report to the printer or to a file and later view stored reports by selecting **View | Inspection Report**. See "Viewing an Inspection Report" and "Printing the Report Window".

For many tasks, the standard inspection report may be exactly what you need. However, PC-DMIS also gives you some powerful tools to generate your own interactive reports. With these tools, you can use templates to determine exactly how your reports look and what information gets included in them. Also, with these templates, you can maintain a consistent look and feel across several reports.

In addition, you have the option to create reports for specific measurement routines called "Custom Reports". This is a useful solution when your reporting needs don't require the use of report templates.

You can only save report (.rpt) files in valid versions of PC-DMIS.

For this version of PC-DMIS, you can save your report file back to version 2021.1. That's the earliest valid version.

This chapter covers the following main topics:

- Reporting Tutorials
- Working with Legacy (HyperView) Reports
- Report Generation Sequence
- About the Report Window
- Using Standard Reports

- Viewing a Legacy Text Report
- Reported Tolerance Zones for Form Dimensions
- Rearranging Report Pages
- Modifying Standard Templates
- Using .DAT Files in Generated Reports
- Modifying the Report's Header
- Changing the Report's Text Colors
- Adding Notes to a Report
- About the Reporting and Form Editors
- Creating Templates
- Creating Forms
- Creating Custom Reports
- About Report Expressions
- Using PC-DMIS ActiveX Controls
- Tutorial - Using Scripting to Enhance your Report
- Working with PDF 3D Files
- QIF Report

Reporting Tutorials

This topic provides a list of available reporting tutorials. If you are new to the template-based reporting functionality that PC-DMIS uses, you should go through these tutorials in order to better understand how reporting is currently handled.

The tutorials below are displayed in the order that they appear in the overall documentation structure of the "Reporting Measurement Results" chapter. You may want to go through this chapter from the beginning to understand how the documentation topics are structured contextually.

- Tutorial - Creating a Report Template (Under the "Creating Templates" main topic)
- Tutorial - Creating Label Templates (Under the "Creating Templates" main topic)

- Tutorial - Customizing a Report Template (Under the "About Reports and Report Templates" main topic)
- Tutorial - Creating Forms (Under the "Creating Forms" main topic)
- Tutorial - Creating a Custom Report (Under the "Creating Custom Reports" main topic)

Additionally, you may find these other main reporting topics and procedures useful:

- Modifying Standard Templates
- Modifying the Report's Header
- Viewing a Legacy Text Only Report
- Changing the Report Window's Contents

Working with Legacy (HyperView) Reports

Legacy reporting (formerly known as HyperView reporting) existed in older 3.x versions of PC-DMIS. It provided the basis for the newer template and form and reporting environments used in 4.x and later versions.

By default, PC-DMIS uses the newer template approach to reporting. What does this mean for those of you with legacy Hyper Reports?

PC-DMIS provides the ability for you to:

- Migrate your HyperView reports into either a custom report or a form. See "Migrating Legacy (HyperView) Reports".
- Continue working directly with your HyperView Reports. You can create and edit reports. See "Using Legacy (HyperView) Reporting".

Migrating Legacy (HyperView) Reports

While you can now create and edit HyperView reports in version 4.3 MR1 and later, you may want to migrate your legacy HyperView reports into PC-DMIS's new reporting environment so that you can work with them there instead.

To convert an existing HyperView report:

1. Select **File | Reporting | Edit | Legacy Report** to open the HyperView report. The report will open in run mode.
2. Select **File | Convert To** and then select either **Report** or **Form**.

- If you select **Report**, PC-DMIS will convert your HyperView report to work with the **Custom Report Editor**. However, all objects may not be converted. Only objects supported by custom reporting will be converted. See "Creating Custom Reports".
- If you select **Form**, PC-DMIS will convert your HyperView report to work with the **Forms Editor**. However, all objects may not be converted. Only objects supported in Forms will be converted. See "Creating Forms".

You can then modify the report or form in the appropriate editor. The original HyperView report remains untouched.

Be aware that certain objects that used to appear in the HyperView Report Editor may not be supported by your selected migration route. For example, if your HyperView report contains button objects, and you migrate them into the Custom Report Editor, the buttons will appear in the editor, but they will not function in the Report window.

Using Legacy (HyperView) Reporting

PC-DMIS also supports the creation and editing of HyperView reports.

- To create a new legacy report, select **File | Reporting | New | Legacy Report**.
- To edit an existing legacy report, select **File | Reporting | Edit | Legacy Report**. The HyperView report will open in run mode. Press CTRL + E to toggle between editing the report in edit mode or testing it in run mode.

Since HyperView reporting is legacy functionality, the documentation doesn't actively exist as topics in this documentation. However, you can download a PDF file of the HyperView documentation by following the link below:

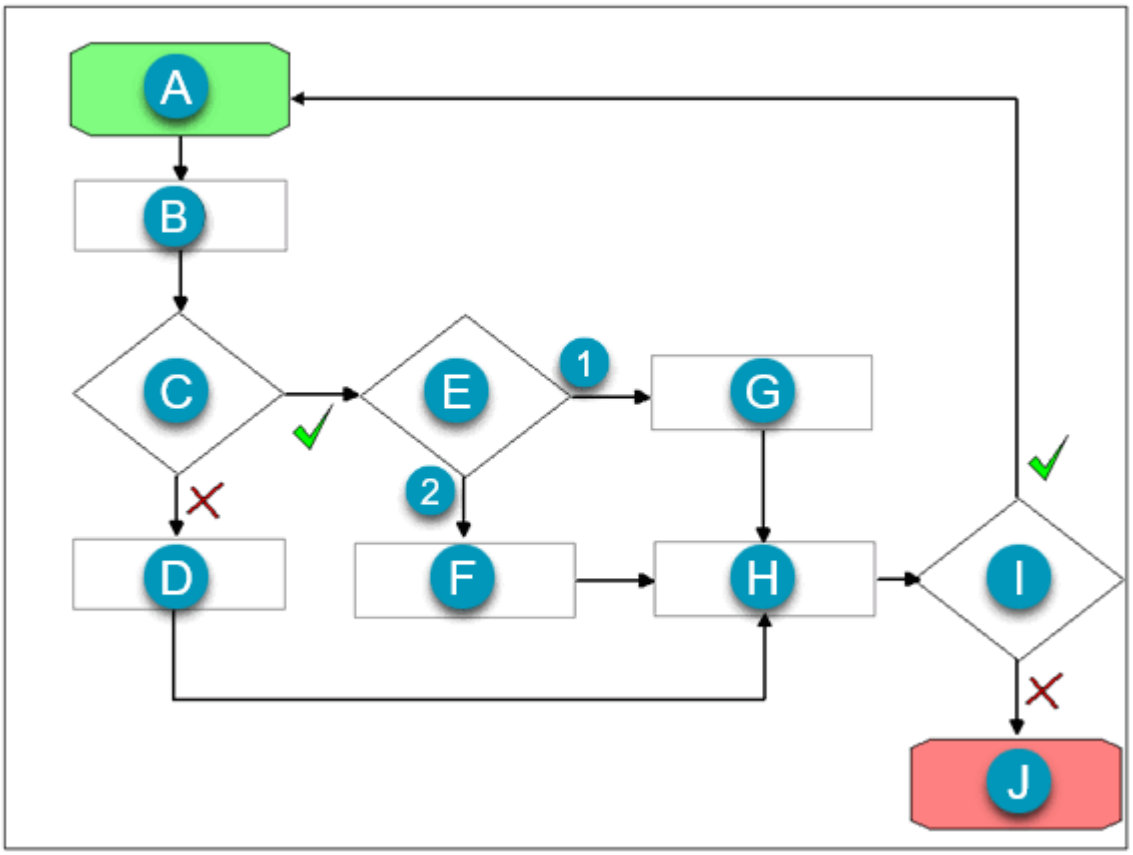
1. In your Internet browser, access this site: https://downloads.ms.hexagonmi.com/docs/FilesInHelps
2. Download the file "creating_hyperview_reports.pdf".

Report Generation Sequence

This topic describes the process by which the report objects in report templates are bound to measurement data and drawn in the Report window during the report generation process.

- PC-DMIS executes each command from the measurement routine.
- Information from each command goes to the report template for possible processing.

- Report objects on the report template are queried, and if the command from which the information came is defined in the **Rule Tree Editor** (for information, see "Defining a Rule in the Rule Tree Editor") to call a label template, then that label template is called. If not, the information will not be displayed in the final report.
- Data is sent to any called label templates to be formatted and displayed according to how the GridControlObject and other reporting objects in the label template are defined (for information see "The Object Bar").
- The report template finally displays the called label templates with their formatted data and displays its own information and any static elements in the Report window.



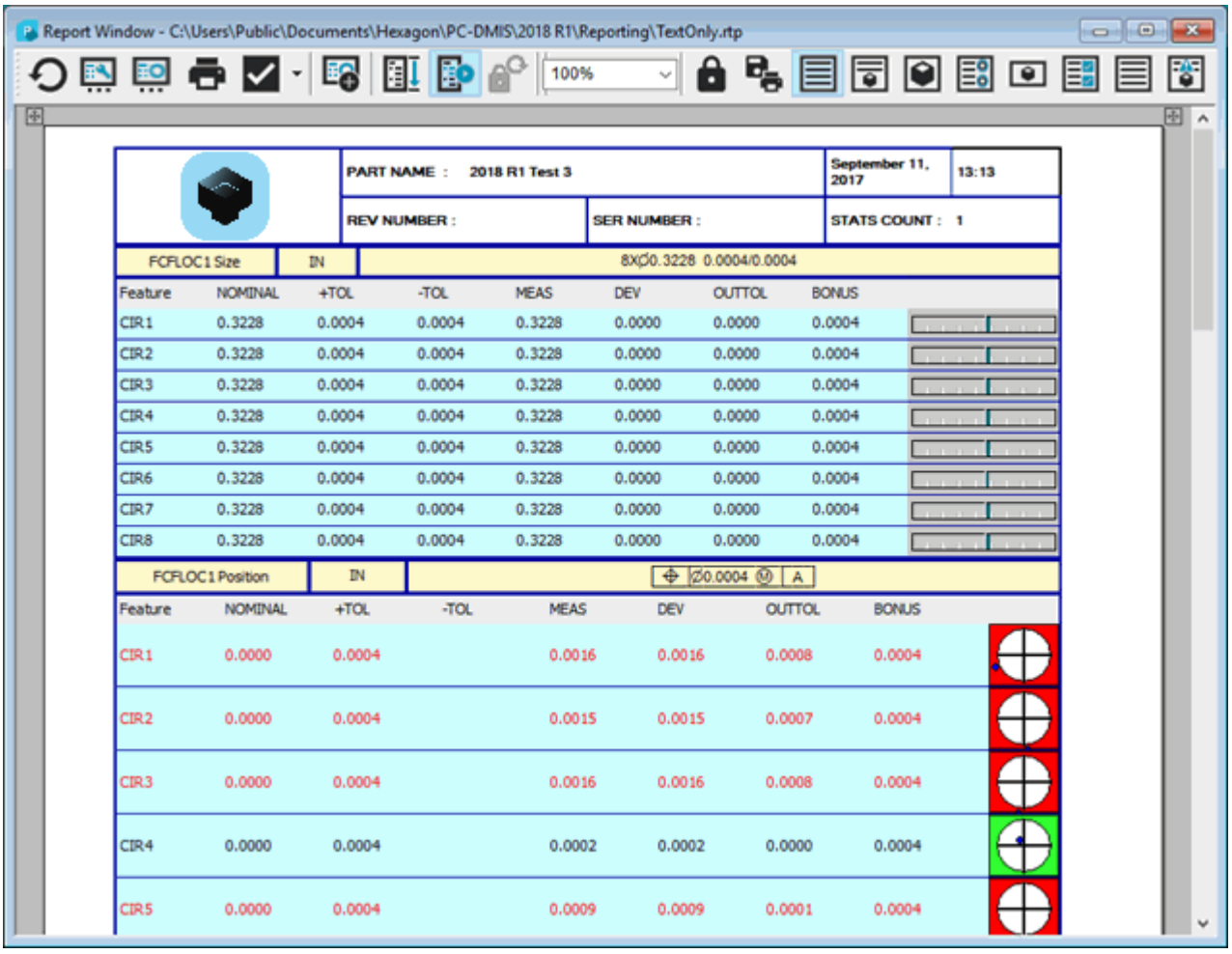
*A - **Measurement Routine Executes a Command***
B - Data from Command Goes to Report Template
C - Command Defined in Rule Tree Editor (RTE)?
D - Nothing Happens
E - RTE Calls Label Template or Text Display
E1 - Label
E2 - Text
F - Textual Message Displayed Instead of a Label Template Call
G - Label Template Formats and Displays Passed Data
H - Report Window Displays Information for Command
I - More Commands to Execute?
J - End

About the Report Window

To display the Report window, select **View | Report Window**. This window displays your measurement results. The Report window acts as any other window in the PC-DMIS application and is subject to the operations on the **Window** menu as well. The Report window's title bar displays the pathway and filename of the current loaded report template. The Report window also has its own **Reporting** toolbar.

With Report Templates

If you use report templates, this window, after measurement routine execution, displays your measurement results and automatically configures the output according to a default report template. You can set any report template as the default report template, but initially PC-DMIS uses the "TEXTONLY.RTP" template.



Report Window showing a standard, text-based report from TEXTONLY.RPT

The Report window contains static content based on the selected report template and the current measurement routine. See "Creating Templates".

With Custom Reports

For a Custom Report, this window does not use a report template to configure or display its data. Instead, it loads the information that you've already defined in the stored custom report. See "Creating Custom Reports".

Reporting Toolbar

Reporting Toolbar

The Report window's **Reporting** toolbar performs these functions:

1. **Redraw the Report** - Redraws and updates the report with newly added changes. If you modify the current report template, label template, or custom report, or your measurement routine changes, click this icon to redraw and update your report to use the newly-modified template or new data from your measurement routine.

2. **Preview Pane** - Opens and closes a pane of thumbnail images so that you can preview the report. Each image on the pane corresponds to a page on the report. For more information, see "Working with the Preview Pane" in this chapter.

3. **Template Selection** - Displays a dialog box that allows you to manage your templates and select a template from which to generate your report output. For more information, see "Applying or Removing a Report Template".

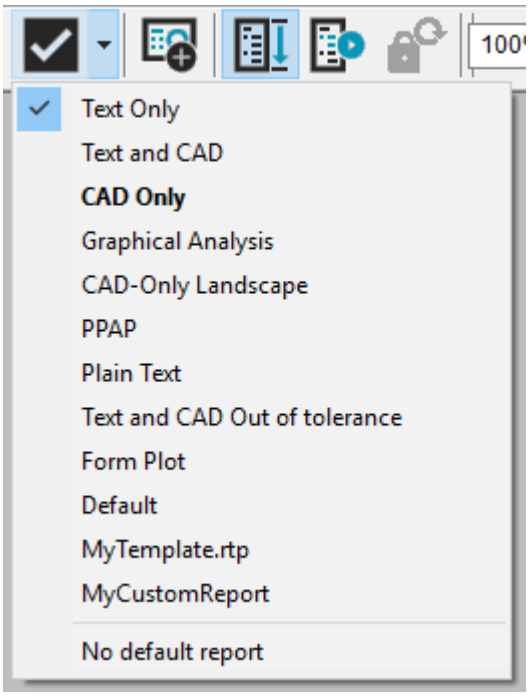
4. **Custom Report Selection** - Displays a dialog box that allows you to create a new blank custom report or open an existing custom report. For more information, see "Creating Custom Reports".

5. **Print Report** - Prints the report based on the output settings defined for the Report window. For information on defining your output, see "Setting Output and Printer Options for the Report Window" in the "Using Basic File Options" chapter.

6. **Set this report as the default report** - Sets the current template on the screen as the new default template for your current measurement routine. PC-DMIS automatically uses the defined default report template whenever you open your measurement routine at a later date. Initially, this default report is set to TEXTONLY.RTP.

The drop-down menu next to this icon lets you set a default report without having to load it first. It displays all the saved report layouts.

To set the default report, select the item from this list. PC-DMIS places the check mark next to it to indicate it as the new default report. The currently loaded report is indicated by a **bold** font.



For example, in the above image, the check mark indicates TextOnly.rtp as the default report and CADOnly.rtp as the current loaded report.

If you select **No default report** then nothing appears in your Report window the next time you load the measurement routine and access the Report window.

Setting a Default Report for All Measurement Routines

If you want to set a default report for all measurement routines, use the PC-DMIS Settings Editor, access the **FileMan** section, and set the `DefaultReportTemplate` entry to the name of the report template you want to use. For information on entries, see the "Modifying Setting Entries" appendix.

7. **Add Template to Toolbar** - Saves the current report template or custom report as a stored report, and a new stored report icon appears on the toolbar with the same name as your template or custom report to the right of the standard report icons. When you click the added icon, the Report window automatically uses the saved template or custom report to display your report.

8. **View Report Mode** - Generates the report from the first command to the last command in the measurement routine, regardless of how PC-DMIS last executed the measurement routine.

9. **Last Execution Report Mode** - Shows only those items executed in the most recent measurement routine execution. It also shows the commands in the same order in which they were executed. If commands are executed multiple times due to a loop, it shows those multiple executions as well.

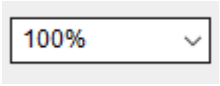
   The `SaveExecuteList` entry in the **Reporting** section of the PC-DMIS Settings Editor determines whether or not this information is retained when you later re-open your measurement routine. By default, this functionality is enabled.

   - If set to 1, PC-DMIS stores the items reported during your last execution within your measurement routine, so you can easily view them later by selecting the **Last Execution Report Mode** icon.
   - If set to 0, PC-DMIS only stores the information temporarily. If you close your measurement routine, PC-DMIS clears out this information.

   Partial Executions and Last Execution Report Mode

   If you perform partial executions, PC-DMIS appends the reported items to the execute list, potentially causing items to be reported that you don't need reported. You can clear out any partially-executed items from the stored execute list by performing a full execution.

10. **View Last Saved Report** - Restores the saved report data to what it was at the time it was last locked. This includes the execute list, temporary modifications made, and the mode that was last used (either Report Mode or Last Execution Report Mode).

11. 

Selecting an item from this list sets the report's magnification in the Report window. The zoom level doesn't affect how the report prints out, but you may find these options helpful when previewing the report or determining what template to use.

- For small percentages, the pages are laid out side by side horizontally first and then vertically. You can also type in a specific magnification. Using a small percentage makes it easier to rearrange pages. For more information, see "Rearranging Report Pages".
- For large percentages, where you are zoomed in on one portion of the report, you can pan the report by either using the scroll bars or by holding Ctrl and dragging on the main page of the report (dragging on a label moves that label). PC-DMIS drags the image in the direction that you pull the mouse.

12. **Lock the Report** - Locks the Report window so that modifications can't be made to any reports in the window. In addition, whenever you click this icon, PC-DMIS saves the following:

    - Temporary modifications you made
    - The execute list
    - The type of report mode used (**Report Mode** or **Last Execution Report Mode**)

    You can restore this saved information by clicking the **View Last Saved Report** icon.

    PC-DMIS shows the icon as "pressed" to indicate it is locked:

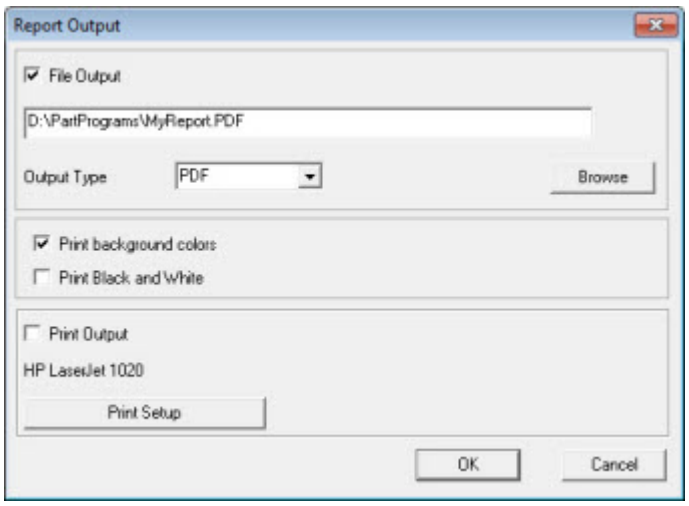
    Also, with the Report window locked, the mouse pointer changes to this symbol whenever you move the mouse over the window to indicate that the report is in a locked state and cannot be modified:

    You can still scroll, print, and view items in the Report window, but you cannot make any modifications to any report until you unlock the window by clicking the icon again.

13. **Save** - Saves the edited custom report with the same name. PC-DMIS enables this button if you enter edit mode with **Edit report** below.

14. **Save As** - Saves the custom report with a new name. PC-DMIS enables this button if you enter edit mode with **Edit report** below.

15. **Save/Print the Report** - Displays a **Report Output** dialog box where you can save the report as a .pdf, .rtf, or .txt file or send it to the default printer.



File Output - Determines whether or not the software sends the report to a file when you click **OK**. The box beneath this check box defines the pathway and file name for saving it as a file. Make sure the filename in this box lists a valid extension.

Output Type - Defines the output type to generate when saving the report as a file. If you type a different extension, the report is not saved to the file.

PDF - Generates a .pdf file using the current report template for the output.

RTF - Generates an .rtf file using the current report template for the output.

TXT - Generates a simple plain-text report using the default.rtp template. It doesn't have any formatting.

3D PDF - Generates a .pdf file as 3D PDF. In this type of .pdf file, you can manipulate the CAD model directly in the .pdf. For more information, see "Working with PDF 3D Files".

Print background colors and **Print Black and White** - Determines whether or not to generate the report with background colors or as a black and white report. For information, see where these are already documented in the "Setting Output and Printer Options for the Report Window" topic in the "Using Basic File Options" chapter.

Print Output - Sends the report displayed in the Report window to the selected printer when you click **OK**. Initially, this is set to your default printer.

Print Setup - Displays the standard Windows **Print Setup** dialog box so you can choose a different printer or change printer options.

16. **Edit report** - If your report is a custom report, this opens your report in edit mode. If your report is one of these supported template reports (Text Only, Text and CAD, CAD Only, or CAD-Only Landscape), the software converts it into a custom report first, and then the software enters edit mode.

> Template conversion does not support data from loops.

For information on editing custom reports, see "Editing or Deleting Custom Reports". For information on custom reports and how to create them, see "Creating Custom Reports".

17. **Text Only**
18. **Text and CAD**
19. **CAD Only**
20. **Graphical Analysis**
21. **CAD-Only Landscape**
22. **PPAP**
23. **Plain Text**
24. **Text and CAD Out of Tolerance**
25. **Form Plot**
26. **Default**

For descriptions and examples of the standard reports (items 17-26 in this list), see "Using Standard Reports" below.

27. This style of icon, if found to the right of the **Default** icon, depicts a modified report template that has been saved to the toolbar. You can use the Report Template Editor to modify existing templates or create new ones. For more information, see "Understanding the Report Template Editor".

28. This style of icon, if found to the right of the **Default** icon, depicts a custom report that has been saved to the toolbar. For information on custom reports, see "Creating Custom Reports".

Customizing and Restoring the Toolbar

You can add your own templates or custom reports to the toolbar by using the **Add Template to Toolbar** icon discussed above. When you add a report to the toolbar, a new icon for that report or template appears on the right side of the toolbar. Each new icon you add represents a stored custom report or custom report template.

- To remove any report icons from the toolbar, hold down Shift and then drag the icon off of the toolbar. This only removes the template or report from the toolbar; it does not delete the item from the system.
- To restore the toolbar to its original toolbar icons, you can do the following:

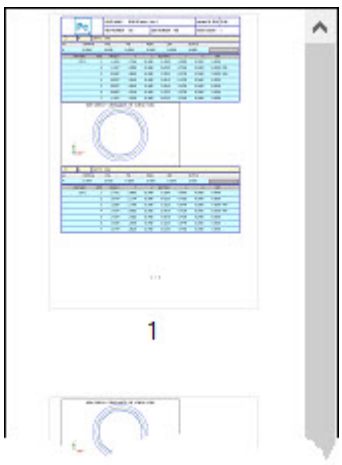
  1. Close PC-DMIS.
  2. Navigate to the C:\Users\<username>\AppData\Local\Hexagon\PC-DMIS\<version>, where <username> is your Windows user name and <version> is the version of PC-DMIS you are using.
  3. Delete reportingtoolbar.dat.
  4. Restart PC-DMIS.

> **!** The **Reporting** toolbar only resides within the Report window. You need to access the Report window to see it. You cannot select it from the toolbar area of the main PC-DMIS interface.

Working with the Preview Pane

The **Preview Pane** button ( ) on the Report window's **Reporting** toolbar opens and closes a pane of thumbnail images so that you can preview the report.

Each thumbnail image on the pane corresponds to a page on the report.



Preview Pane

When the pane is open, you can click on a thumbnail image to go to that report page.

Entering Edit Mode for Custom Reports

To perform certain operations with the preview pane, you need to place your custom report into edit mode. This accesses a slimmed down version of the Custom Report Editor. This lets you edit a report without leaving the usual PC-DMIS menu structure. You can do this through either of these options:

- Click **Custom Report selection dialog** ( ) and then click **Blank Report** and then **Open**.
- Click **Edit report** ( ).

For more complex edits, you can still use the Custom Report Editor.

Adding and Deleting Pages in Custom Reports

In edit mode, you can add or delete pages on your custom report.

To add a page, click the large plus button ( ) after the last thumbnail image; or right-click on any thumbnail image and select **Add Page**.

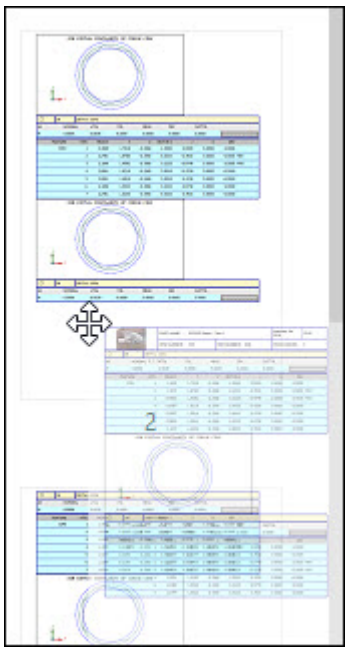
To delete a page, either select a thumbnail image and press Delete on your keyboard; or right-click on the thumbnail image and select **Delete Page**. You must always have at least one page in your report.

For more information on custom reports, see "Creating Custom Reports" in this chapter.

Ordering Pages in Reports

To define the order of pages in any report:

1. Locate the thumbnail image of the page you want to move.
2. Drag that page to the desired location up or down in the preview pane.
3. Release the mouse button when you see the crosshair cursor ( ) to insert the page at that location.



Moving Objects to Other Pages in Custom Reports

In edit mode, you can move one or more objects from one page to another page through the thumbnail images.

1. Select the objects that you want to move until you see the green handles around them. (You can press Ctrl to select multiple objects.)
2. Drag the objects onto the thumbnail image of the page where you want to place the objects.
3. Release the mouse button.
4. Click on that thumbnail image to go to the page, and position the object.

Another way to move objects to another page is to cut the objects (Ctrl + X), click on the thumbnail image of the page, and then paste the objects (Ctrl + V).

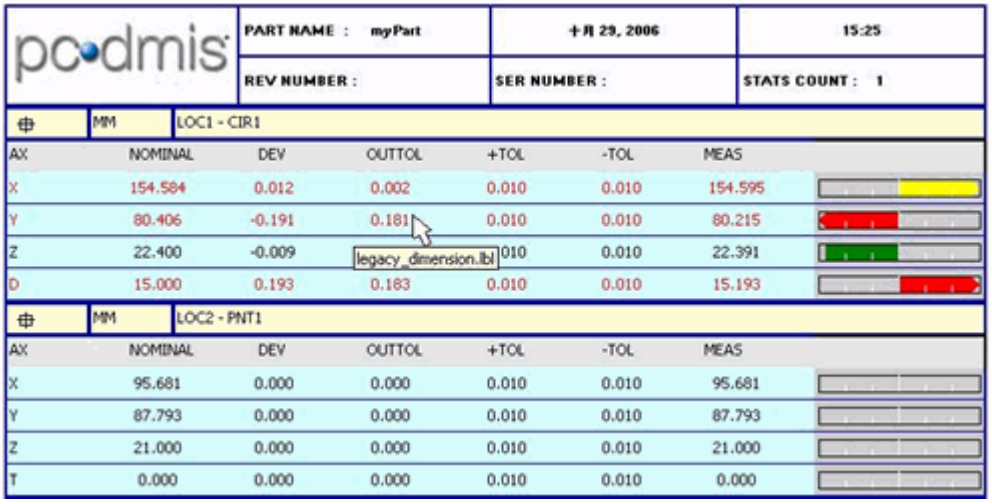
If you select a CADReportObject, and move that to a new page, any labels associated with it also move with it. You cannot move labels to that object by themselves.

Sizing the Preview Pane

You can size the pane with your mouse pointer. To do that, drag the vertical bar that separates the pane from the report. You can size the pane to a maximum of 30% of the report's width and a minimum of 10% of the report's width.

PC-DMIS saves the visibility state of the pane and the pane's width to the current measurement routine. PC-DMIS uses the last used visibility state and width for new measurement routines.

Using Tooltips in the Report Window



Example report showing a tooltip with the name of the label being used

By simply moving your mouse over different parts of the Report window, you can get information in the form of a tooltip about what label is being used, or if there is no label, the tooltip will display either the name of the object or the current page number.

- If the object is a label, the tooltip displays the label template file name.
- If the object is not a label, for example, if it's a **TextReportObject** or a **CadReportObject**, the tooltip displays the object name that is defined in the report template.
- If you hold the mouse over an empty space on the page, (technically the **Page** object), the tooltip displays the page number.

These tooltips provide you with a simple way to find out what objects are being used in the report.

For information on objects, see "The Object Bar".

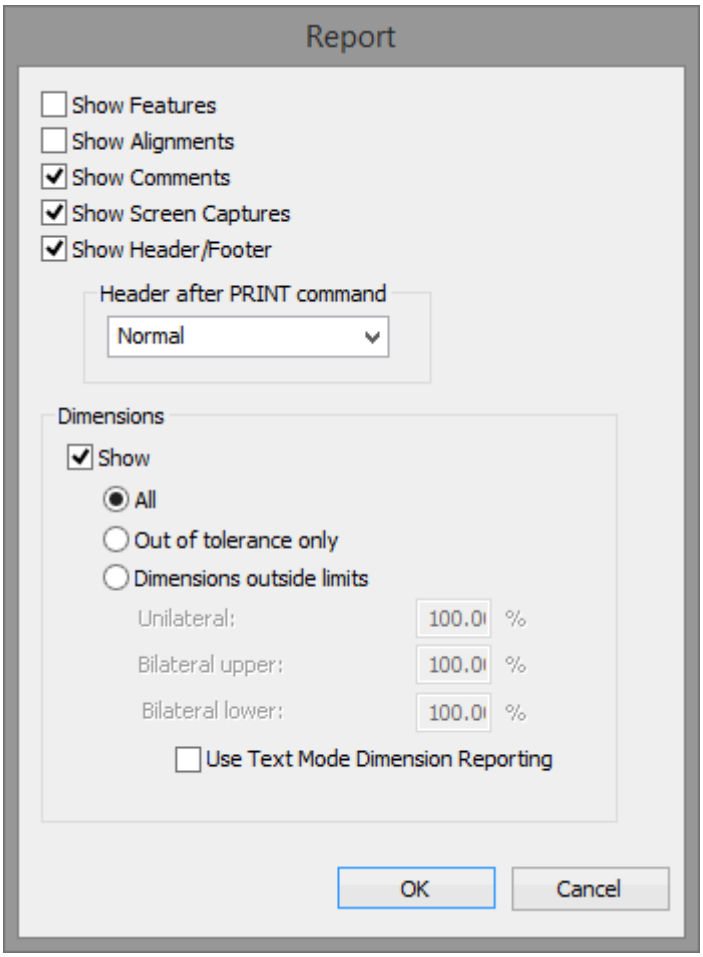
Changing the Report Window's Contents

See the "Shortcut Menus in the Report Window" topic for a list of functions you access to perform some of the modifications discussed in this topic and its subtopics.

While you can permanently change how and what information appears in your Report window by modifying the underlying templates and the rules used, you can also perform modifications to the objects on a Report window directly within the Report window by using the information provided in the topics below.

Please note that custom reports, templates, label templates, and other reporting modifications are NOT backwards compatible.

Edit Text Reporting

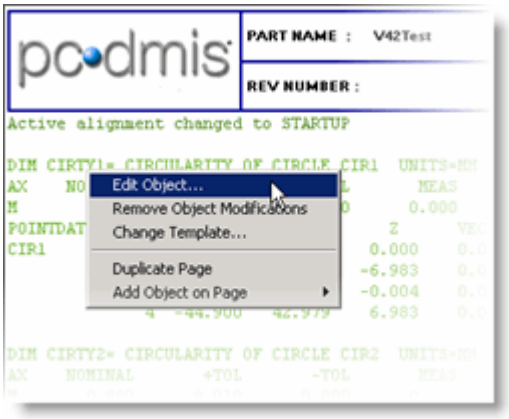


Report Dialog Box

The **Report** dialog box (available with template reporting) lets you determine the overall information that PC-DMIS includes in the Report window and how it displays that information.

You can access this dialog box in these ways:

- Right-click on any text-only reporting text or white space at the end of the Report window and then select the **Edit Object** menu item when the shortcut menu appears.



For additional information on this shortcut menu, see the "Shortcut Menus in the Report Window" topic from the "Using Shortcut Keys and Shortcut Menus: Introduction" appendix.

- Press Shift and then right-click on any label object in the Report window, and then select the **Edit Object** menu item.

The following check boxes let you show or hide various items:

Show Features

This option displays all feature measurements in your measurement routine.

If you are using the default report template, textonly.rtp, and your measurement routine measures a feature with more than the minimum number of points, it also displays a form plot of the feature.

Show Alignments

This option shows alignment changes as they occur during the measurement routine. It will display all alignment changes that occur in the dimension or feature lists.

Show Comments

This option shows any comments that have been added to the measurement routine. (For information on comments, see "Inserting Programmer Comments" in the "Inserting Report Commands" chapter.)

Show Header / Footer

This option displays a header and footer in your report by applying the label template defined in the **File Header** rule of the **Rule Tree Editor**. By default, this is the FILE_HEADER.LBL label template that ships with PC-DMIS.

Show Screen Captures

This option displays any screen capture graphics related to DISPLAY/METAFILE and ANALYSISVIEW commands in the report. (For information on screen captures, see "Using Screen Captures of the Graphic Display Window" in the "Editing the CAD Display" chapter.)

The **Header after Print command** area is enabled if you mark the **Show Header / Footer** check box. This area determines how PC-DMIS handles a report header *after* a PRINT/REPORT command. The list in that area contains these options:

Normal

PC-DMIS generates only one report header. It does not generate a new report header even if you execute a measurement routine with a PRINT/REPORT command in any kind of loop. (For information on the different kinds of loops, see the "Branching by Using Flow Control" chapter.)

Always

PC-DMIS always generates a new report header for any next report output that follows the PRINT/REPORT command.

If File Header executed

PC-DMIS generates a new report header only if you executed the File Header command block before the PRINT/REPORT command. The File Header is the command block in the Edit window that has the part name, revision number, serial number, and so on. (This shows as "File Header" in Summary Mode.) By default, the File Header uses the FILE_HEADER.LBL template during report generation. While you cannot delete the File Header command block, you can skip over it with certain partial execution options, such as **Execute from cursor**. (For more information on partial execution, see "Executing Measurement Routines" in the "Using Advanced File Options" chapter.)

For more information on the PRINT/REPORT command, see "Inserting a Print Command" in the "Inserting Report Commands: Introduction" chapter.

The **Dimensions** area allows you to control the display of dimensions in your reports. To display dimensions, select the **Show** check box. Once selected, the other items in this area become available for selection. These include:

All

Selecting this option acts just as if you had selected both the **Out of Tolerance Only** and **Dimensions Outside Limits** options.

Out of Tolerance Only

If this option is marked, PC-DMIS will only show the out of tolerance dimensions. ("Show Dimensions" must be ON.)

This option is only available when you select **Show Dimensions** and *deselect* **Show Dimensions Outside Limits**.

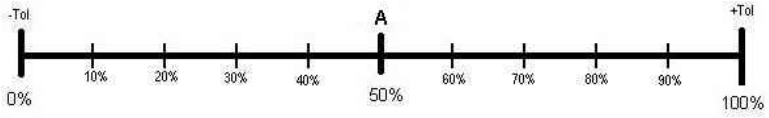
Dimensions Outside Limits

If you select this option, PC-DMIS will only show the dimensions that lie outside the percentage of the tolerance zone.

When you select the **Show Dimensions Outside Limits** check box, the tolerance percentages will become editable, allowing one percentage for unilateral tolerances and an upper and lower percentage for bilateral tolerances.

The tolerance zone can be considered as a range between 0 and 100 percent (similar to how dimension colors are applied to tolerance zone regions or how the colored bar graphs are drawn at the end of dimensions). The lower limit (nominal - minus tol) corresponds to 0% and the upper limit (nominal + plus tol) corresponds to 100%. Consider the following chart where A, on the center of the chart, represents the nominal value:

Chart



Dimensions that have deviations outside the tolerance zone will either be less than zero or greater than one hundred percent.

For a *unilateral* dimension (like roundness, which only has a plus tolerance), the zero deviations are at zero percent, and out of tolerance is anything larger than one hundred percent.

Unilateral Example

Nominal: 0.0000

Measured: 0.0028

Deviation: 0.0028

Plus Tol: 0.0050

Minus Tol: 0.0000

Because this deviation is 56% of the tolerance range, it will be displayed if Unilateral Percentage is less than 56%.

For a *bilateral* dimension (like distance, which has both a plus and minus tolerance), the zero deviations are somewhere in the middle. If the tolerances are equal, the zero deviations will be at the fifty percent position.

Bilateral Upper Example

Nominal: 3.0000

Measured: 3.0075

Deviation: 0.0075

Plus Tol: 0.0100

Minus Tol: 0.0100

The percentage is calculated using the Lever Rule:

(measured value – lower limit) / (upper limit – lower limit) * 100.

With this data, it would be:

*(3.0075-2.9900) / (3.0100-2.9900) * 100. = 87.5%*

This dimension will be displayed in the inspection report if the Bilateral Upper Percentage is *greater* than 87.5%.

Bilateral Lower Example

Nominal: 3.0000

Measured: 2.9925

Deviation: 0.0075

Plus Tol: 0.0100

Minus Tol: 0.0100

The percentage is calculated using the Lever Rule:

(measured value – lower limit) / (upper limit – lower limit) * 100.

With this data, it would be:

*(2.9925-2.9900) / (3.0100-2.9900) * 100. = 12.5.%*

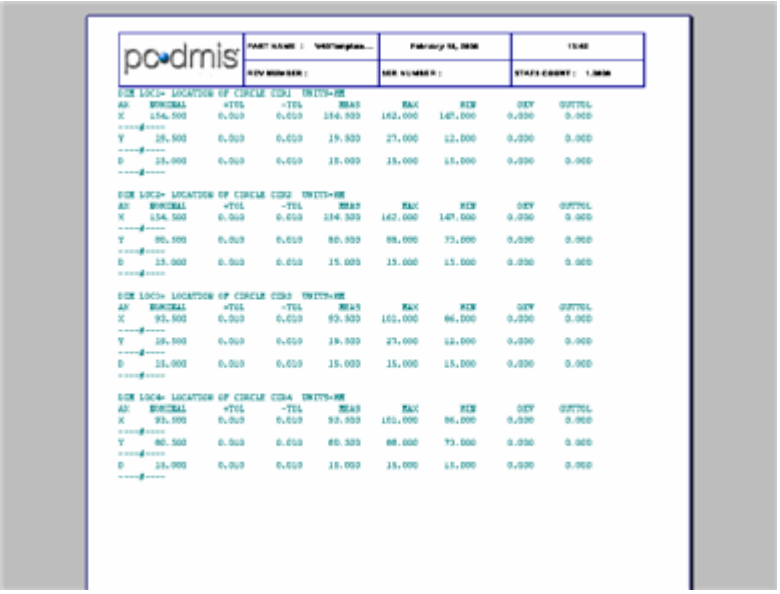
This dimension will be displayed in the inspection report if the Bilateral Lower Percentage is *less* than 12.5%.

Use Text Mode Dimension Reporting

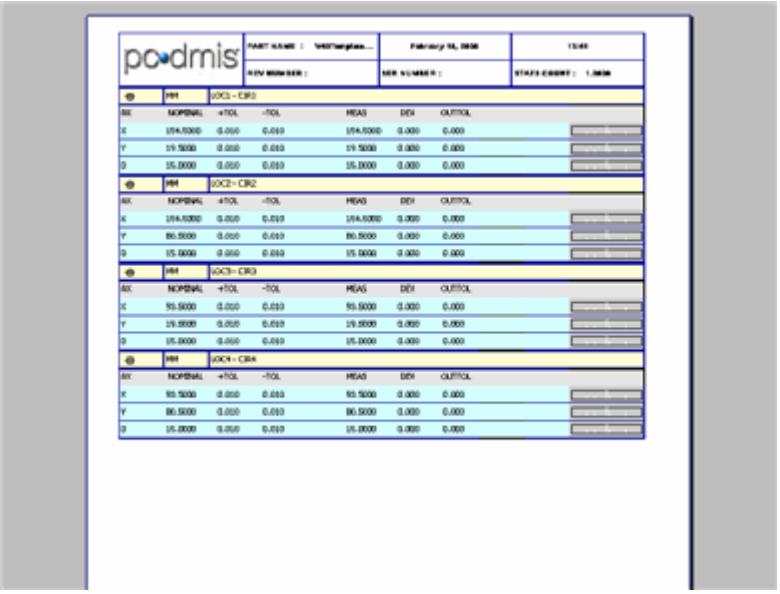
This check box determines whether PC-DMIS uses formatted text (selected) or uses a graphics table (not selected) for displaying the dimensions.

For this report type, you must use only Legacy dimensions in your measurement routine.

Sample Report with "Use Text Mode Dimension Reporting" Selected:



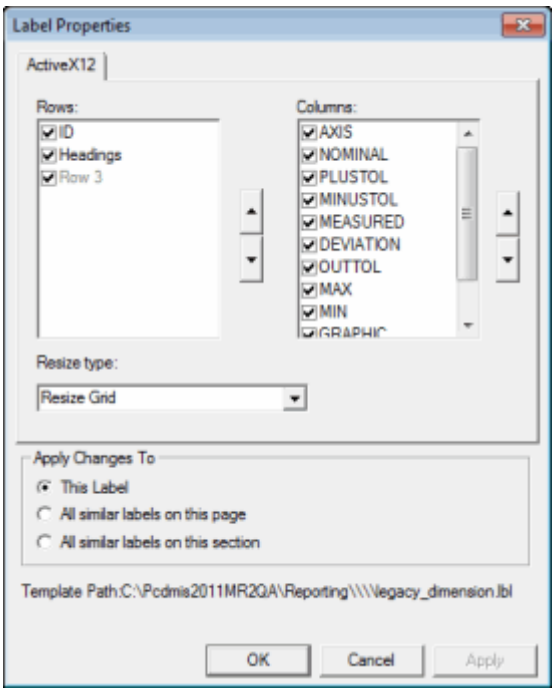
Sample Report with "Use Text Mode Dimension Reporting" Not Selected:



If PC-DMIS cannot find a specified report template, it will use a template named "default.rtp". This generic template provides you with a basic text-based report. Since default.rtp only supports text-based reporting, clearing this check box will do nothing.

Edit Label Reporting

If you right-click on a label and click **Edit Object**, the **Label Properties** dialog box appears.



Label Properties Dialog Box

With this dialog box you can change the order of columns and rows, or you can control the visibility state of individual columns and rows.

> **To change the order** - Select a row or column, then click the black up or down arrow.
>
> **To hide or show an item** - Clear the check box next to the item to hide it. Select the check box to display it.

Once you have made the desired modifications to the label, select one of the option buttons at the bottom to apply the changes to the current label, all similar labels on the current page, or all similar labels in the entire section.

Determine what the label will do whenever you hide or show columns, by selecting an item from the **Resize Type** list. For a label associated with a CadReportObject, **Resize Grid** is the only option. For other label contexts, these three items are available:

> **Resize Grid** - This retains existing column sizes and resizes the label to fit the new width.
>
> **Resize Columns to Fit** - This retains the label's existing width and resizes each column equally to fit the existing width.
>
> **Hide Text** - This doesn't resize the label or the columns. It simply hides the text.

Click **OK** and PC-DMIS will change the display.

Edit Label Positioning and Size

If your label has leader lines or it is a free label (it isn't part of a TextReportObject), if you press CTRL and then click on a label (or if you drag the mouse to box-select a label), you will see that it becomes selected. Once selected, you can drag it to a new location or re-size it by clicking and then dragging the black square handles that surround the selected label.

Edit CAD Model Reporting

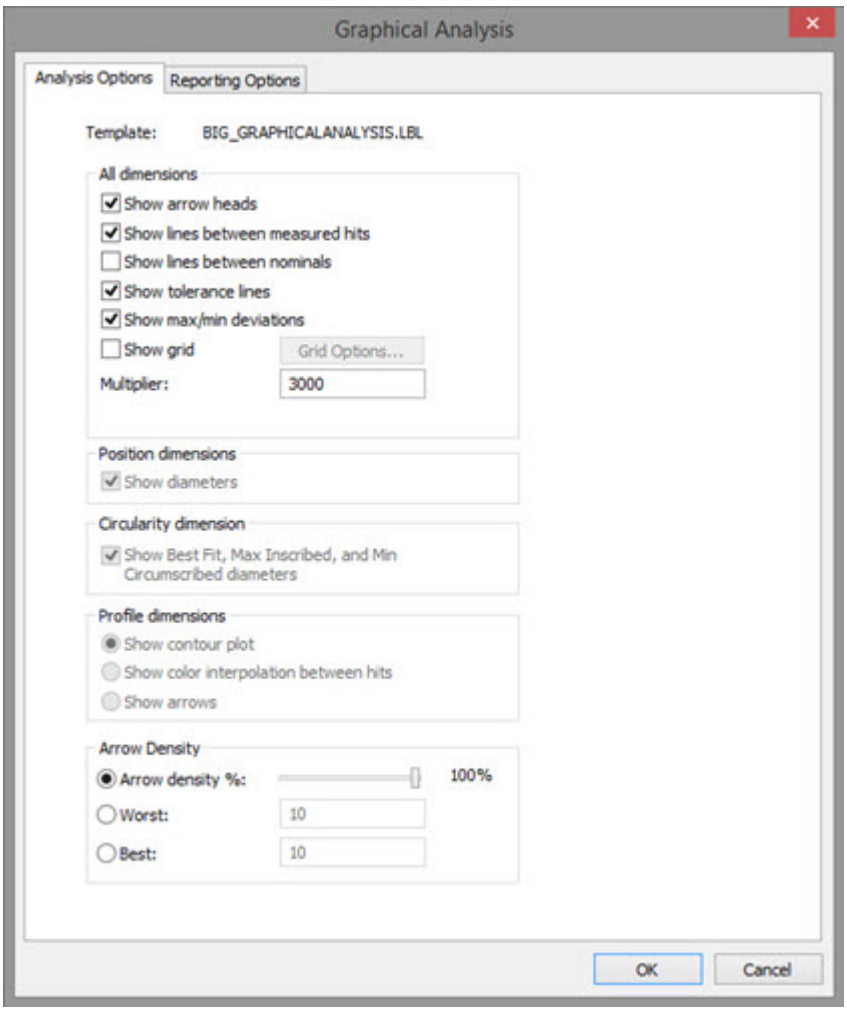
If you double-click on any CadReportObject, it becomes "active". This means you can rotate or zoom the CAD model to how you like it, just as you can inside the Graphic Display window. Click outside of the CadReportObject to use the new orientation and zoom level. See "CadReportObject" for information on this.

If you right-click on a CadReportObject in the Report window, and select **Edit Object**, PC-DMIS displays the **Label Layout Wizard**, allowing you to quickly modify label positions and the CAD model's location directly within the Report window. See "The Label Layout Wizard" topic under the "CadReportObject" for information on how to use this wizard.

You can size and move the CadReportObject as described in "Edit Label Positioning and Size".

Edit Graphical Analysis

If you right-click on an **AnalysisWindow** object in a Report window and choose **Edit Object**, a **Graphical Analysis** dialog box appears:



For information on the options on these tabs, see the "Changing Properties" sub-topic in the "AnalysisWindow Object" topic of the current chapter.

The items that appear shaded are items that are not available for modification on the fly. However, you can modify them at the label template level. For information on editing label templates, see "About Labels and Label Templates" in this chapter.

Changing Label Templates on the Fly

PC-DMIS provides you with a way of changing on the fly a label template used in a report.

Changing a Single Label Template

From the Report window (**View | Report Window**), select the label template and then right-click on it and select the **Change Template...** menu item. You can then select the new template to use from the **Open** dialog box.

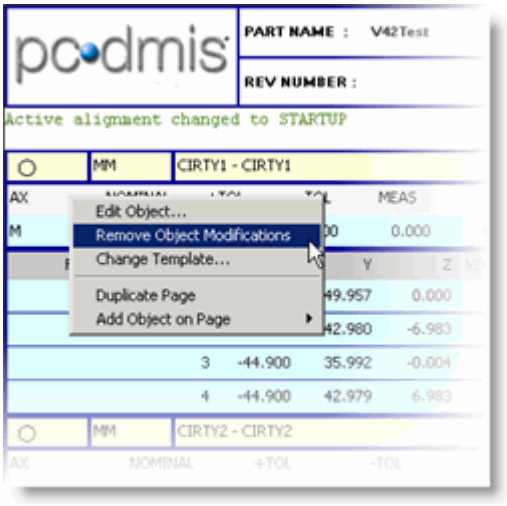
Changing Multiple Labels on the Same Page

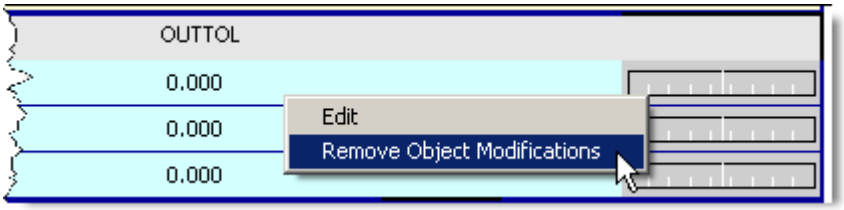
To select and replace multiple labels on the same page, hold down CTRL, select multiple templates, and then select the **Change Template...** menu item. Alternately, you can choose **Select Matching Labels** to choose all the labels in a CADReportObject that use that same label and are on the same page.

Removing Report Window Modifications

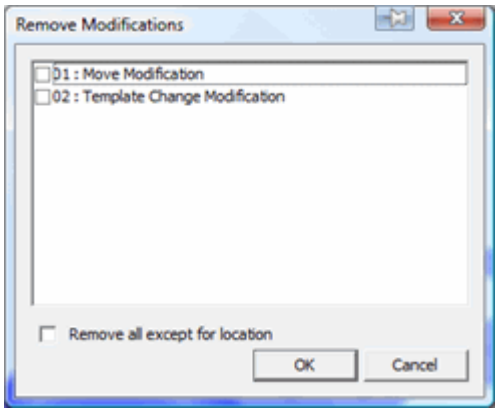
Removing Modifications to One Object

To quickly remove modifications that you have made to a report object (a TextReportObject, CADReportObject, or Label object), right-click on the desired object, and select the **Remove Object Modifications** menu item.





- If only one modification was made to an object, PC-DMIS will return the report object back to its default state.
- If more than one modification was made, PC-DMIS displays the **Remove Modifications** dialog box. This dialog box contains check boxes next to each type of modification made, allowing you to remove targeted modifications to the report object by selecting the modification to remove and clicking **OK**. The **Remove all except for location** check box removes all modifications made to that object except for repositioning changes.



A Sample Remove Modifications dialog box showing two different modifications on an object.

Removing All Modifications

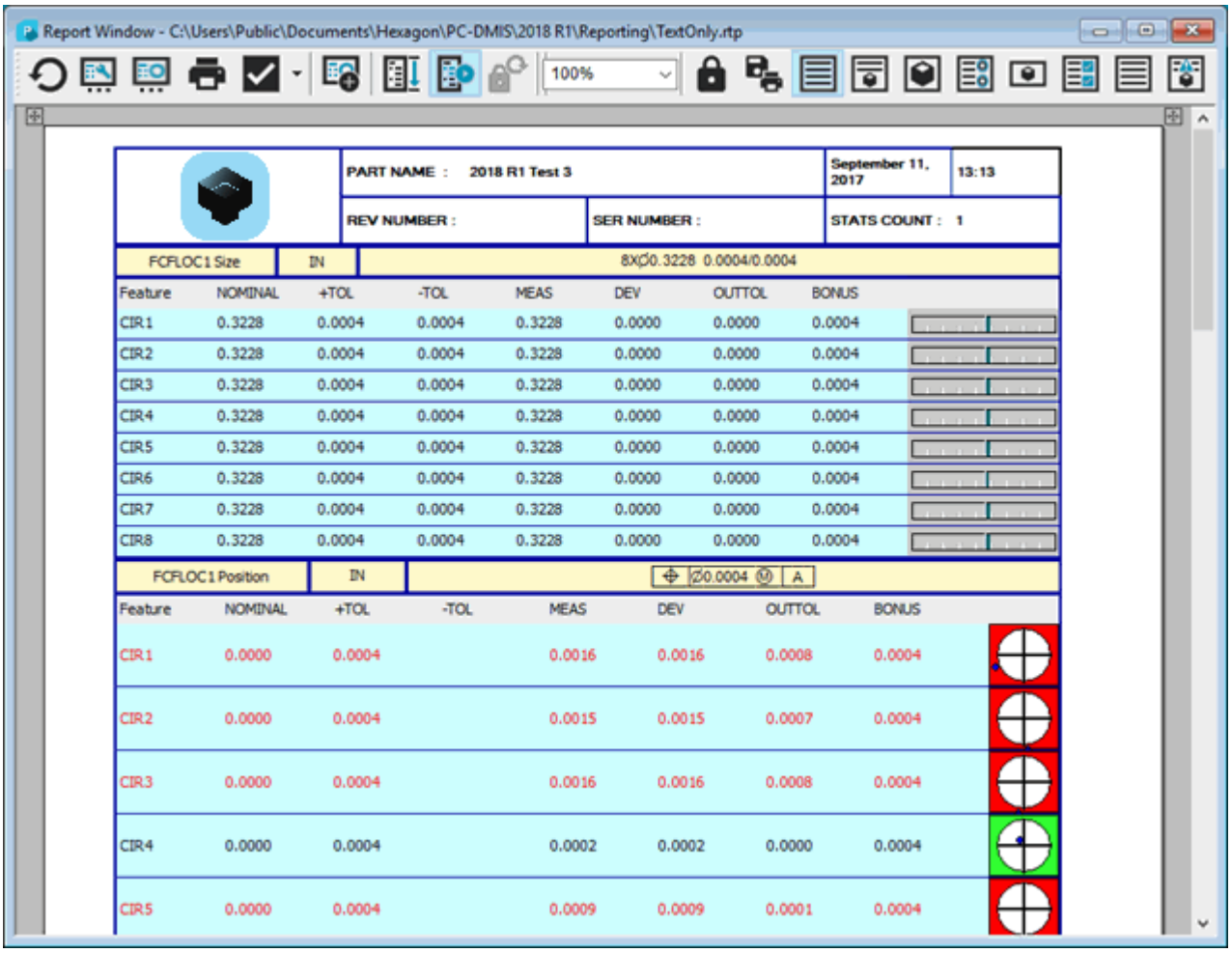
To quickly remove all object modifications on a report template, select the **File | Reporting | Clear Template Associated Data** menu item.

Removing Pan, Zoom, Rotate Modifications for a CAD Object

You can also remove just pan, zoom, and rotate modifications for the CADReportObject. To do this, right-click on the object in the Report window, and choose **Remove CAD Reporting Object Pan/Zoom/Rotate MOD**. Other mods (such as section cuts, various label definitions, solid or wire displays and so forth) are not affected.

Using Standard Reports

The default standard report template, TEXTONLY.RTP, is a simple template that plugs measurement data into a TextReportObject to show text-based information in a more graphical way.

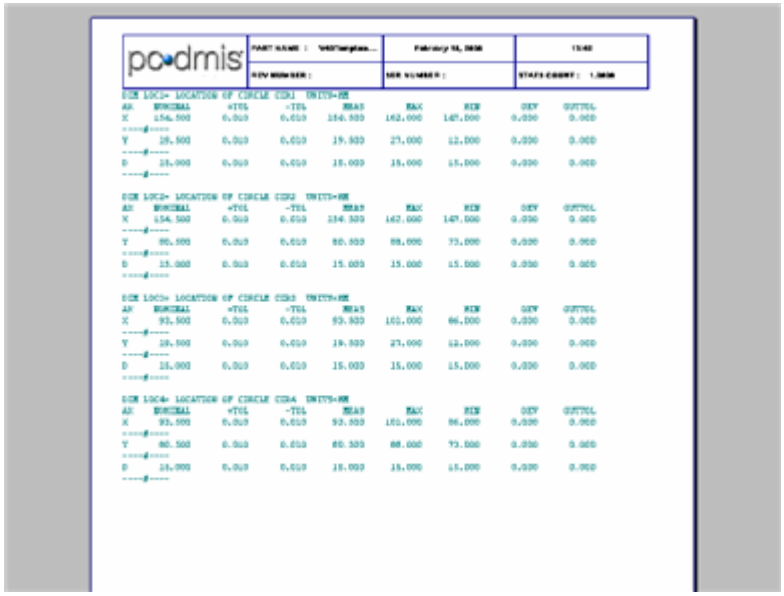


Report Window showing a standard, text-based report

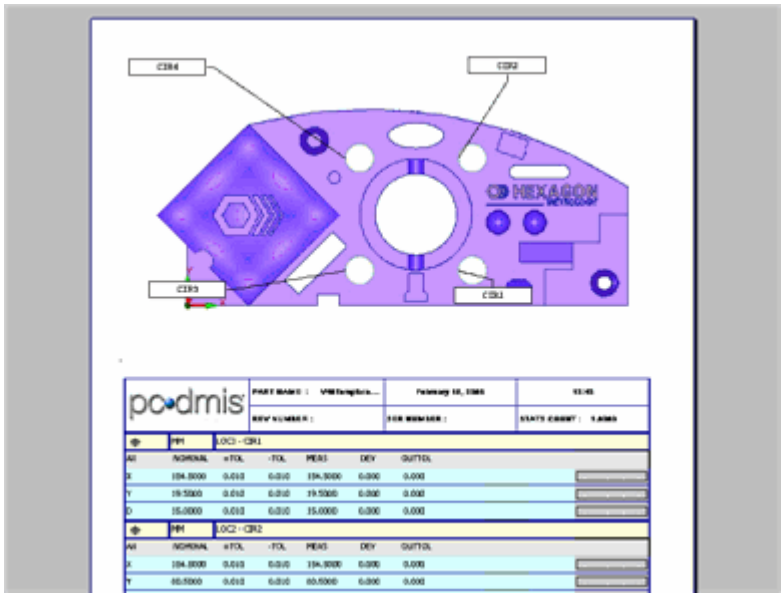
If the TEXTONLY.RTP standard report template doesn't show what you need, PC-DMIS also ships with additional standard report templates that you may find useful. You can

easily change the layout used for the current report by clicking the desired icon from the **Reporting** toolbar. These are the standard report templates:

- **Text Only** (TEXTONLY.RTP) - This template uses the TextReportObject with some standard labels included.



- **Text and CAD** (TEXTANDCAD.RTP) - This template uses the TextReportObject and the CadReportObject with some standard labels included.



- **CAD Only** (CADONLY.RTP) - This template uses the CadReportObject with some standard labels included in portrait layout. Since this template supports fewer labels than CADONLY_LANDSCAPE.RTP (up to 10), the labels are larger and more detailed.

- **Graphical Analysis** (GRAPHICALANALYSIS.RTP) - This template uses the Analysis Object to provide you with a graphical analysis of each form dimension (such as Flatness, Roundness, and so forth) sent to the report. It will not show an analysis for non-form dimensions. PC-DMIS graphically displays the deviation of each individual hit for the dimensions used. The report displays the error in the form of individual arrows for each hit. These arrows, with their colors and directions, indicate the relative size of the error, as well as its direction.

- **CAD-Only Landscape** (CADONLY_LANDSCAPE.RTP) - This is similar to **CAD Only**, but it displays in landscape layout. Since this template supports more labels than CADONLY.RTP (up to 30), the labels are smaller and less detailed.

- **PPAP** (PPAP.RTP) - This template produces a report that can be used in Production Part Approval Process (PPAP).

- **Plain Text** (PLAINTEXT.RTP) - This template produces a simple report without any formatting. The report retains its table-like organization.

- **Form Plot** (FORM PLOT.RTP) - This template produces a very similar report to the graphical analysis report, except it produces a larger output, displaying one form analysis per page. You must use a form dimension (circularity, cylindricity, flatness, straightness, line profile, or surface profile) to see data in this report.

- **Text and CAD Out of Tolerance** (TEXTANDCAD_OOT.RTP) - This template produces a text and CAD report showing the out-of-tolerance dimensions.

- **Default** (default.rtp) - This template produces a raw text (legacy) report. This is similar to reports used in older versions of PC-DMIS when the software didn't have a dedicated Report window. PC-DMIS defaults to this report if it cannot locate or display other report templates.

These templates are accessible from the **Report Templates** dialog box by default. For information on accessing and using this dialog box, see "Applying or Removing a Report Template". The actual template files are located in a public Reporting sub-directory.

You can find the Reporting folder at this location. This folder contains all the templates that the Report window uses:

C:\Users\Public\Documents\Hexagon\PC-DMIS\<version>\Reporting

<version> represents the PC-DMIS version. If you want to customize existing templates or create new ones, do so in that directory.

If you need to restore any modified template to its factory settings, you can copy it from the DefaultReportingTemplateBackup directory located within the PC-DMIS installation directory and paste it the above Reporting directory.

Do not modify the templates in the DefaultReportingTemplateBackup directory.

If you load a measurement routine that uses a report template that does not exist in the specified reporting directory nor in the directory where you installed PC-DMIS, then PC-DMIS generates a text-only report template on the fly called "default.rtp". This allows you to always see a report in the Report window even if you don't have the specified report template. Remember, default.rtp only displays information in a text-only format. This means you cannot deselect the **Use Text Mode Dimension Report** check box in the **Report** dialog box. For information on accessing and using the **Report** dialog box, see "Changing the Report Window's Contents".

Viewing a Legacy Text Report

```
DATE=6/26/2013                          TIME=12:38:09 PM
PART NAME  : 2013 MR1 QA - Width Construction
REV NUMBER : REV123
SER NUMBER : SER456
STATS COUNT : 3

Active alignment changed to STARTUP

PLN3=PLANE MEASURED FROM 3 HITS
PLN1=PLANE MEASURED FROM 3 HITS
PLN2=PLANE MEASURED FROM 3 HITS
CIR1=CIRCLE MEASURED FROM 4 HITS
CIR2=CIRCLE MEASURED FROM 4 HITS
CIR3=CIRCLE MEASURED FROM 4 HITS
CIR4=CIRCLE MEASURED FROM 4 HITS
PLN4 = PLANE BUILT FROM FEATURE PLN3
CIR5=CIRCLE MEASURED FROM 7 HITS
PLN5=PLANE MEASURED FROM 3 HITS
PLN6=PLANE MEASURED FROM 3 HITS
PLN7=PLANE MEASURED FROM 3 HITS
PLN8=PLANE MEASURED FROM 10 HITS
Active alignment changed to A1

LIN1=LINE MEASURED FROM 4 HITS
DIM ANGL1= 2D ANGLE FROM PLANE PLN6 TO PLANE PLN7
AX        MEAS     NOMINAL       +TOL       -TOL        DEV      OUTTOL
A      316.1003   316.1003     0.0100     0.0100     0.0000     0.0000 ----#----

DIM ANGL1= 2D ANGLE FROM PLANE PLN6 TO PLANE PLN7
AX        MEAS     NOMINAL       +TOL       -TOL        DEV      OUTTOL
A      316.1003   316.1003     0.0100     0.0100     0.0000     0.0000 ----#----
```

Report Window showing a legacy text report

This topic discusses how you can get a legacy (old style) text-only report to show up in your Report window. PC-DMIS version 3.7 and earlier used this report type.

For this report type, you must only use legacy dimensions in your measurement routine.

You can toggle between using legacy dimensions and the newer Geometric Tolerance commands from the **Insert | Dimension | Use Legacy Dimensions** menu.

For information on using the .DAT files with this template, see "Using .DAT Files in Generated Reports".

1. Select **View | Report Window** to open the Report window.

2. From the **Reporting** toolbar, click the **Default** icon .

3. *This step is optional.* If you intend to modify the finished report inside an RTF editor, you may want to change how RTF files are converted. In the PC-DMIS Settings Editor, expand the **Printing** section, and set `DoNotUseAmyUniRTF` to **2**. Click **Save Setting** and then click **OK**. This makes the text for the report print to RTF files like it used to in 3.7 and earlier. If this is set to 0 (the default), characters show up inside of text boxes.

Reported Tolerance Zones for Form Dimensions

PC-DMIS displays the tolerance zone for form dimensions as a linear graphic. For text mode reporting, this zone is comprised of characters with the number sign (#) indicating the amount of deviation. For non-text mode reports, this is shown as a bar graph with the bar growing larger from left to right as the deviation increases. For the non-text mode dimensions the color of the bar graph changes as deviations move towards out of tolerance.

For form dimensions (Straightness, Flatness, Circularity, and Cylindricity) there is no negative tolerance, so the linear graphic represents the deviation from perfect form (0 tolerance) where the extreme left of the graphic is 0 tolerance and the extreme right of the graphic is the maximum allowed tolerance.

Example showing text mode and non-text mode tolerance zones.

In the graphical example given above, the tolerance zone is indicated in both text-mode and non-text-mode with a deviation of 0.186 and a maximum tolerance of 0.3.

Here are some additional examples showing how a Circularity dimension's tolerance zone appears inside of PC-DMIS as a text-mode and non-text mode report respectively:

```
FCFCIRTY1 = CIRCULARITY OF CIR1   UNITS=MM
AX     NOMINAL       +TOL       -TOL       MEAS        DEV      OUTTOL
M        0.000       0.030      0.000      0.027       0.027      0.000 -------#
```

A sample text mode tolerance zone.

FCFCIRTY1	MM	◯ 0.03				
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CIR1	0.000	0.030		0.027	0.027	0.000

A sample non-text mode tolerance zone.

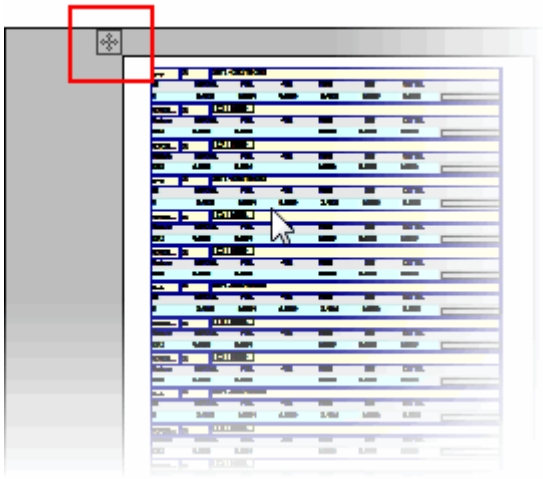
Rearranging Report Pages

PC-DMIS provides you with a drag-and-drop approach to rearranging pages in the Report window (**View | Report Window**).

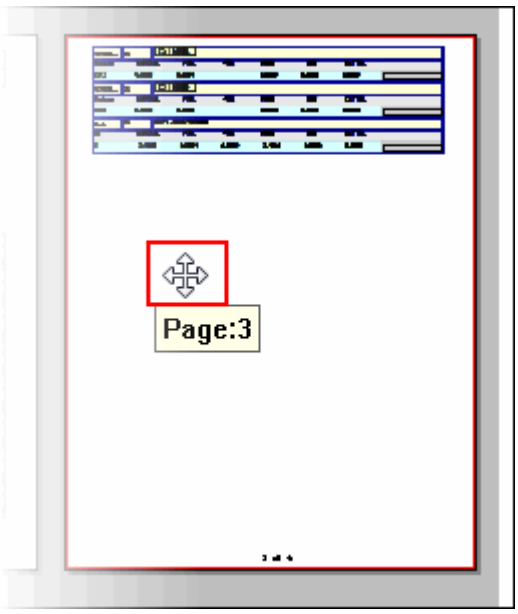
You can also rearrange pages with the Preview Pane. For more information, see "Working with the Preview Pane".

To rearrange pages, use this procedure:

1. Size the report magnification to where you can view the pages you want to rearrange, probably somewhere around 25%.
2. Hover your pointer over any report page. Notice that at the top left of the page, a small cross-hair icon appears. Click that icon.



3. Drag your pointer to drag the current page.
4. Drag the page on top of an existing page. The target page receives a red highlight and the pointer cursor changes.



5. Release the mouse button. PC-DMIS arranges the ordering of the report pages according to the direction in which you dragged the report page.

    - If you drag and drop the page to a page earlier in the existing page sequence, the software inserts the page *before* the page on which you dropped it.
    - If you drag and drop the page to a page later in the existing page sequence, the software inserts the page *after* the page on which you dropped it.

    Suppose you have a sequence of seven pages in your report: p1, p2, p3, p4, p5, p6, p7. If you drag p4 and drop it at p2 (earlier in the sequence), the order will be p1, p4, p2, p3, p5, p6, p7. If you then drag p4 and drop it at p5 (later in the sequence), the order will be: p1, p2, p3, p5, p4, p6, p7

6. Continue rearranging pages as needed.

Modifying Standard Templates

You may want to modify the standard reporting and label templates that ship with PC-DMIS.

- To change your report's header, see "Modifying the Report's Header".
- To change properties in a standard template report, open the template in the template editor, right-click on the object, and modify its properties in the **Properties** dialog box. For information, see "About Object Properties".

If you want to create entirely new templates, see "Creating Templates".

About the Reporting Directory

You can find the Reporting folder at this location. This folder contains all the templates that the Report window uses:

C:\Users\Public\Documents\Hexagon\PC-DMIS\<version>\Reporting

<version> represents the PC-DMIS version. If you want to customize existing templates or create new ones, do so in that directory.

If you need to restore any modified template to its factory settings, you can copy it from the DefaultReportingTemplateBackup directory located within the PC-DMIS installation directory and paste it the above Reporting directory.

Do not modify the templates in the DefaultReportingTemplateBackup directory.

Using .DAT Files in Generated Reports

PC-DMIS uses the LOGO.DAT, HEADER.DAT, and ELOGO.DAT data files to a limited extent in a generated report. For information on these data files, see "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window" chapter.

- LOGO.DAT populates the beginning of the report.
- HEADER.DAT populates the header of each subsequent page.
- ELOGO.DAT populates the ending text on the last page of the report.

Conditions for this to Work

- You must use a report template that does not contain rules, such as default.rtp.
- The `UseLogoElogoHeaderDat` entry in the **Reporting** section of the Settings Editor must be set to 1, the default value.

Limitations of Using .DAT Files

- Bitmaps are placed at the top of the report, regardless of where in the .DAT files they're defined.
- The last page will not contain the HEADER.DAT information if the only item on the last page is ELOGO.DAT.

Modifying the Report's Header

Many users want to change the header in some way. Usually, they want to modify the default image that appears in the default report's header to something more company-specific. You can change the header image by following one of these procedures below.

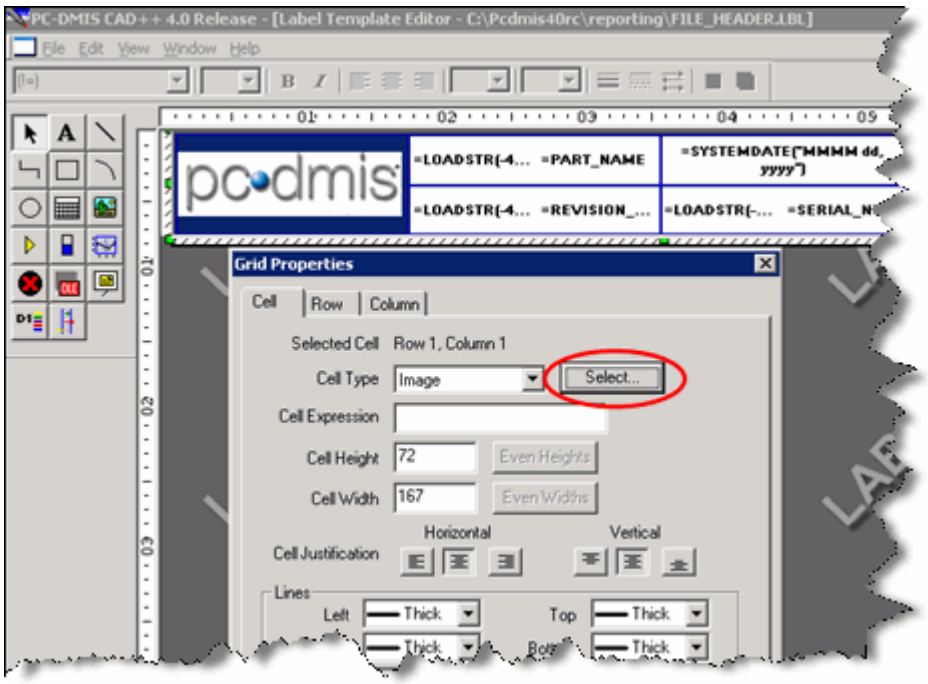
To Change Only a Header's Image:

If you only need to change the image but not other elements in the header, we recommend this:

1. Access the report in the Report window or load a custom report inside a Custom Report Editor.
2. Right-click on the header (such as FILE_HEADER.LBL).
3. Choose **Change image** to show the **Open** dialog box.
4. From the **Open** dialog box, locate the desired image, and then click **Open** to update the report with the image. This action also updates the underlying label template (.lbl) with the new image.

To Modify an Entire Header in the Label Template Editor:

1. Open the label template in the Label Template editor.
    - Select **File | Reporting | Edit | Label Template**. An **Open** dialog box appears.
    - Select the FILE_HEADER.LBL template and click **Open**.
    - In the Label Template Editor you should now see the edit area filled with a GridControlObject. This object controls all the header data that you see in the report.
2. Change the header's image.
    - Double-click on the GridControlObject. You will see a hash border around the object. This tells you that you can edit the code or items inside of the header.
    - Right-click on the PC-DMIS graphic, and a **Grid Properties** dialog box appears.
    - In the **Cell** tab, click **Select**. An **Open** dialog box appears.



Example showing the button to use to change the report header's graphic

- Navigate to the new graphic image you want in the header, select it, and click **Open**. The dialog box closes.
- Click **Apply**. The new graphic appears in the cell.

3. Make other modifications to the header as desired.
4. Save and test the modified label template.
   - Once you have configured things as desired, click **OK**, and then select **File | Save** to save your changed Label Template.
   - Select **File | Close** to close the Label template editor, returning to your measurement routine.
   - Select **View | Report window**. If your change doesn't immediately appear, select the **Redraw the report** icon from the **Reporting** toolbar.

Here are some other options to consider:

Using LOGO.DAT, HEADER.DAT, and ELOGO.DAT in Generated Reports

To a limited extent, you can use the # keywords contained in these .DAT files to define the information that appears in your generated report. See "Using .DAT Files in Generated Reports" for more information.

See "Modifying the Edit Window's Headers and Footers" in the "Using the Edit Window: Introduction" chapter for information on the # keywords.

Converting LOGO.DAT, HEADER.DAT, and ELOGO.DAT Keywords

You can also use PC-DMIS's own Report Expressions to accomplish the same thing as the # keywords contained in the .DAT files.

The following table details the equivalent Report Expression or report object to use and where to use it.

.DAT # Keyword	Functionality	Equivalent Command in Report Template
#DATE	Inserts the current date.	=SYSTEMDATE("MMMM dd, yyyy")
#TIME	Inserts the current time.	=SYSTEMTIME("HH:mm")
#PAGE	Inserts the current page number.	=Page()
#TRn	Inserts the value of tracefield n, where n is the tracefield number.	=TRACEFIELD(n)
#PARTN	Inserts the measurement routine name.	=PARTNAME( )
#DRWN	Inserts the revision number.	=REVNUM()
#SERIALN	Inserts the serial number.	=SERNUM( )
#SEQUENCE	Inserts the sequence number.	(Not applicable)
#SHRINK	Inserts the scale factor.	=MEASSCALE()
#NMEAS	Inserts the total number of dimensions.	=NUMMEAS()
#NOUT	Inserts the total number of dimensions that are out of tolerance.	=NUMOUTTOL()

#ELAPSTIM	Inserts the time elapsed between start and end of execution.	=ELAPSEDTIME()
#BMP=bitmappath	Inserts a bitmap of the specified full path and name.	Use Bitmap object

For information on using other expressions in your report templates, see "About Report Expressions".

For information on limitations on using variables inside report headers, see "Displaying a Variable's Value".

Changing the Report's Text Colors

At times you may want to modify the default text colors used in your report. To change these text colors, access and modify the **Color Editor** dialog box for your report template's TextOnlyObject:

1. Select **File | Reporting | Edit | Report Template**.
2. Select the report template for which you want to change the colors.
3. In the Report Template editor, select the **TextReportObject**, and right-click on it to access the **Properties** dialog box for that object.
4. Select the **Colors property** from the **Properties** dialog box.
5. The **Color Editor** appears. This is the same editor as used for the Edit window, but in this case it modifies the TextOnlyObject used in your report template. For information on how to use the Color Editor, see "Defining Edit Window Colors" in the "Setting Your Preferences" chapter.
6. Make your changes, and then save the template.
7. Load the template into the Report window.

Adding Notes to a Report

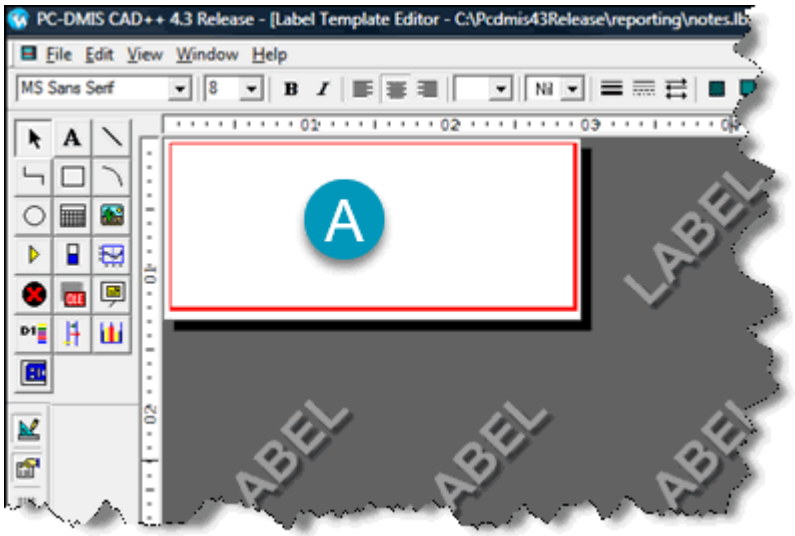
It's often useful to add your own notes to a report after it has executed. One way to add notes is to use PC-DMIS's standard Comments functionality and allow PC-DMIS to display the comments in the Report window. But this approach has some limitations. Displaying comments only works on report templates that use the TextReportObject. Besides, by default, standard comments aren't very visible, and they only appear within the

TextReportObject at the location they were executed. Also, you aren't able to easily edit them after they're on the report. For information on comments, see "Comments" in the "Using the Edit Window" chapter.

However, if you want more flexibility, consider the option of adding a note box into which you could type your notes directly onto the final Report. The following steps provide you with a procedure on how to do this:

Step 1: Create the Note Box

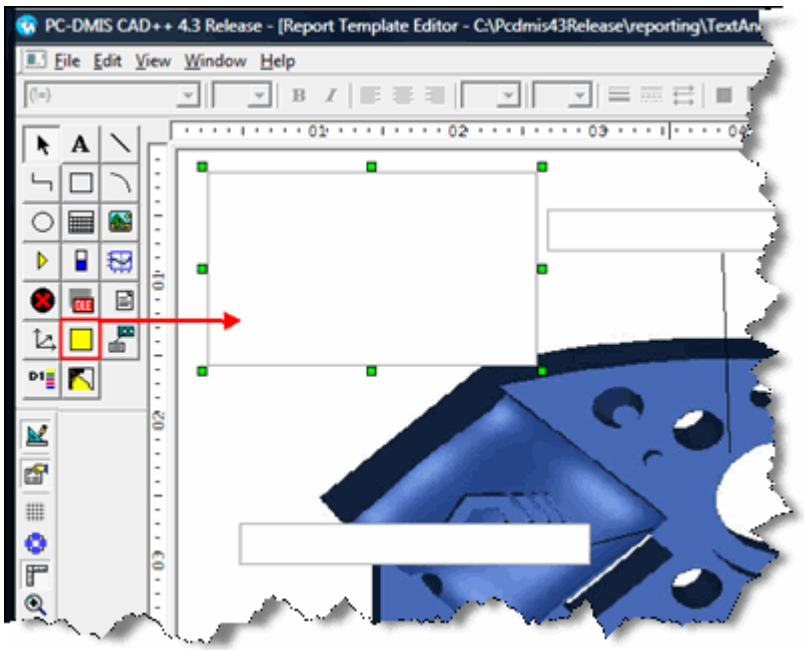
To create a note box, create a new label template inside the Label Template Editor, and drag a **Text** object onto the editing area. Format the object with font size, font color, background color, and borders as desired. If you want your note box to be transparent, select the **NIL** background color. Set the object's **Text** property to hold this value: "Right-click here and select Edit Object to modify this note text." Save your label template.



A new label template named notes.lbl
(A) - Right-click here and select Edit Object to modify this note text.

Step 2: Tie the Note Box onto the Report Template

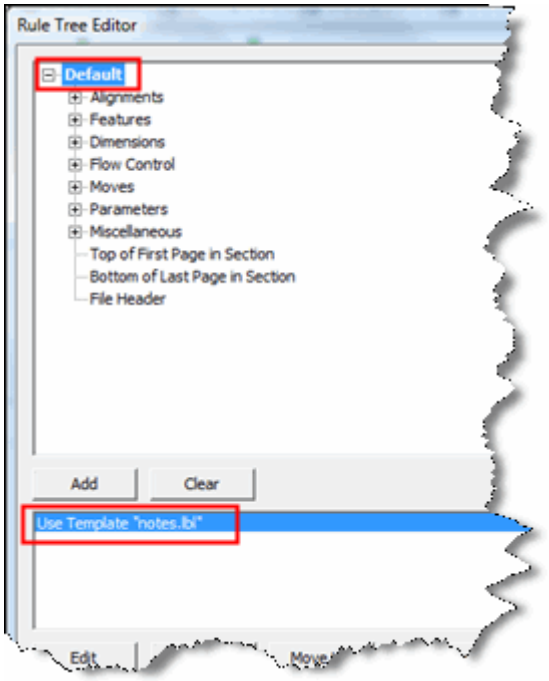
Open up a report template inside the Report Template Editor that you can edit. Drag a **Label** object onto the editing area, and position it more or less where you want the note box to appear. You can change its final position in the Report window.



A label object added onto a TextAndCad Report

Step 3: Tell PC-DMIS When to Display the Note Box

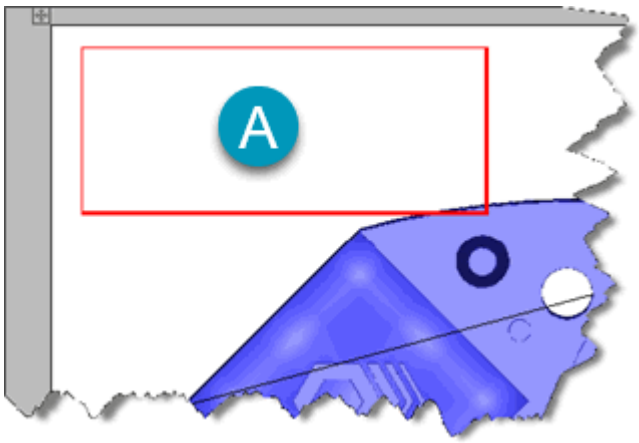
Right-click on the label object to access the **Properties** dialog box, and then click **Rules** to bring up the **Rules Tree Editor**. The new label object does not yet have any rules. Add a rule for the **Default** item at the top of the tree, and have the rule pull in the label template you saved in Step 1. This action causes the label template containing your note box to show up on every page of the report. Save your changes.



A rule specifying to load notes.lbl assigned to the Default item

Step 4: Test the Report

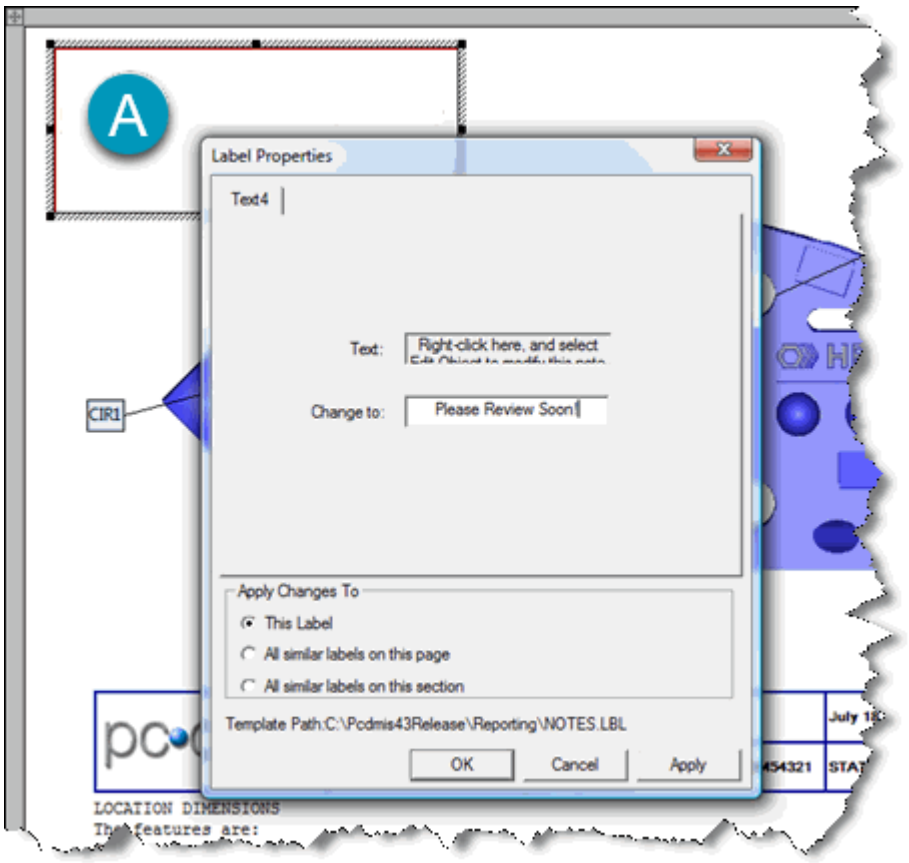
Access the Report window and load in your modified report template. Notice that your note box appears with the instructions on how to change its default text. Double-click to activate the object and drag it to a new location if desired.



A note box with default text
(A) - Right-click here and select Edit Object to modify this note text.

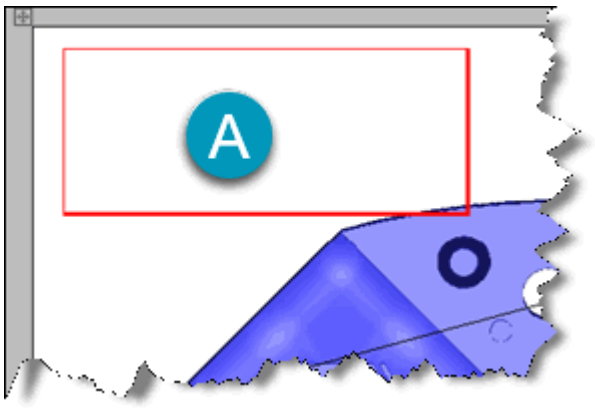
Step 5: Modify the Note Box Text

To change the text in the note box, right-click the object and select **Edit Object**. A **Label Properties** dialog box appears.



The Label Properties dialog box
(A) - Right-click here and select Edit Object to modify this note text.

Type your new note text in the **Change to:** box and click **OK**. PC-DMIS updates your text.



A note box with updated text
(A) - Please Review Soon

And that's all there is to it. Of course, you can modify the note box to better suit your needs. For example, if you wanted a table-like structure in your note box, on Step 1, instead of a single text object in the label template, you could add in a **GridControlObject** into the label template and then overlay the cells with **Text** objects.

If you wanted to show only the note box based on certain conditions, you could change the rules used in the **Rules Tree Editor** in Step 3. For example, if you only wanted the note box on the first page of your report, you would use the **File Header** item instead.

Adding Watermarks to Certified Measurement Routine Reports

You can add a watermark to generated reports for certified measurement routines. The UseWaterMarkText Settings Editor entry controls whether the watermark appears, and WaterMarkText Settings Editor entry specifies the watermark text.

About the Reporting and Form Editors

The Report Template Editor, Label Template Editor, Custom Report Editor, and Form Editor modify existing or create new, templates, custom reports, and forms. These editors share many of the same common user interface elements: the menu bar, the toolbars, the object bar, and so on. While the individual icons or items on these menus may change between editors, the look and feel are essentially the same.

The user interface is discussed in the following topics.

Menu Bar for the Form and Reporting Editors

File menu

File | New - Creates a blank template or form.

File | Open - Opens a previously stored report template file or form.

File | Close - Closes the template editor or form.

File | Save As - Saves the current template or form with a new file name. See "Saving a Template or Form to an Earlier Version".

File | Edit / File | Run - Lets you toggle between the editor's two modes: *Edit Mode* and *Run Mode*. Run mode lets you "run" or test a form. Edit mode lets you modify a template or form. (Run mode only works with the Form Editor; it does exist in the Report or Label template editors.)

File | Exit - Exits PC-DMIS. If you have unsaved changes, PC-DMIS will display a message box asking if you would like to save them.

Edit menu

Edit | Layout | Align Objects, Space Evenly, Center in View, Make Same Size - These submenus lets you layout, align, and space objects in the editor. See the "Layout Bar" for additional information.

Edit | Layout | Properties - Displays the property sheet for the currently selected object. If no object is selected, this displays the property sheet for the editing area (called a "Section" or "The Frame/View"). See "Property Sheet" and "About Sections".

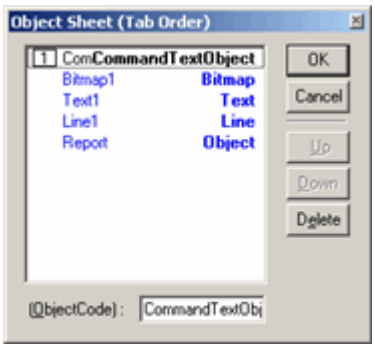
Edit | Layout | Check Mnemonics - This menu item ensures that any shortcut keys using the ALT key plus a letter are unique on form or template items that have a text display (such as buttons).

To define a shortcut key for a form or dialog box control, type an ampersand symbol (&) before the character. For Example:

A button with the text label of "&Continue", looks like "Continue" on the form and is accessible with ALT + C.

Edit | Layout | Grid Settings - Displays the **Grid Settings** dialog box. This dialog box lets you display or hide a grid of points on your layout's background. See "Working with the Grid".

Edit | Layout | Objects - Displays the **Object Sheet** dialog box. Use this dialog box to view or select the objects in the Form and to set their tab order. See "Object Sheet". (Only available in the Form Editor.)



Object Sheet Dialog Box

Edit | Order - This submenu lets you move objects behind or in front of other objects that overlap each other.

Edit | Undo - Lets you undo the last action taken in the editor.

Edit | Redo - Lets you redo the last undone action.

Edit | Cut - Cuts the object and stores it for pasting.

Edit | Copy - Copies the object and stores it for pasting.

Edit | Delete - Deletes the selected objects.

Edit | Paste - Pastes the cut or copied object. If from a copied object this often this pastes the object right on top of the copied object, so you will need to drag it to a new location or else it will overlap the object from which it was copied.

Edit | Paste Special - This differs from the standard **Paste** menu item since it allows you to maintain a link between the pasted item and the source from which it was copied so that if the source changes, the pasted item will also get updated. For Example:

Suppose you want to place a Microsoft Excel chart object in your Hyper Report, and you want to keep the chart's data in synch with the data from the Excel file. To do this:

1. In the Microsoft Excel file, copy the chart object.
2. Access your template or form.
3. Select **Edit | Paste Special**. The **Paste Special** dialog box appears.
4. Select the **Past Link** option.
5. Click **OK**. The chart object appears in your template or form.

Now, if you make changes to the data for your chart object in your Excel file, PC-DMIS will update the chart object in the report or form.

Edit | Set Tab Order - This lets you determine the objects that get selected when a user presses TAB to cycle through objects in Run Mode. (This only works in the Forms Editor.)

Edit | User Assigned Properties - Accesses the **User Assigned Properties** dialog box, allowing you to manage all your user assigned properties at once. See "Working with User Definable Properties". (Only available in the Report Template Editor.)

View menu

View | Graphic Display Window, Edit Window, Preview Window, Form Editor, Marked Sets Window, Basic Script Editor, Inspection Report, Probe Readouts, Probe Toolbox - Shows or hides these PC-DMIS windows or editors. Most of the time these do not serve any purpose when working in the editor, and you can hide them.

View | Object Bar - Displays the editor's Object Bar. See "Object Bar".

View | Layout Bar - Displays the editor's **Layout Bar**. See "Layout Bar".

View | Font Bar - Displays the editor's **Font Bar**. See "Font Bar".

View | Ruler Bars - Displays the rulers on the top and left sides of the editor. See "Toggle Ruler".

View | Snap Points - Displays the "snap points" for any of the objects in the editor. See "View Snap Points".

View | Route - This only functions with snap points. On complex pages with a lot of objects, you may find it useful to use the this menu option to see the chain of objects connected to the currently selected object via snap points. This menu option selects all objects connected in some way by snap points to the currently selected object. (Only available in the Form Editor.)

Windows menu

This menu lets you perform standard window manipulations with any of the windows in PC-DMIS. See the "Navigating and Displaying Multiple Windows" chapter.

Help menu

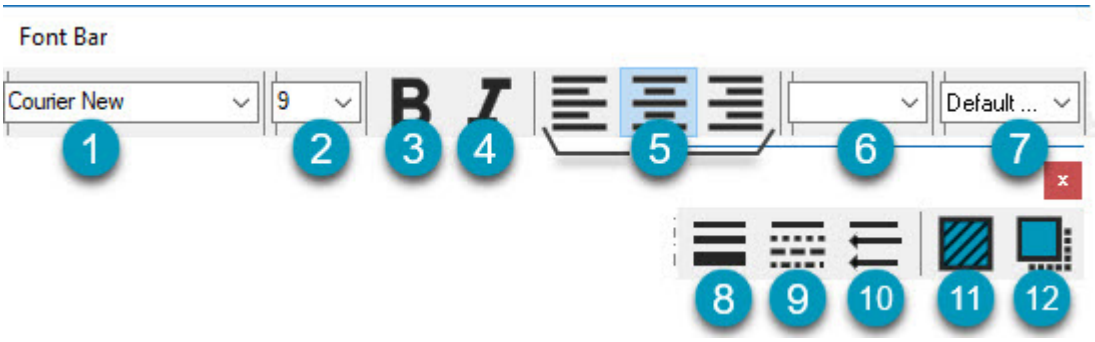
Help | Index - Accesses the PC-DMIS Help file.

Help | Using Help - Displays a topic about using the Help file.

Help | About PC-DMIS - Displays the **About PC-DMIS** dialog box, showing information about your version of PC-DMIS.

The Font Bar

The editor's **Font Bar** lets you modify frequently modified font, color, background, line, and other properties for various objects.



The following options are found on the **Font Bar**:

1. This list defines the object's font if it supports text. You can also set this in the **Font** property.
2. This value defines the object's font size if it supports text. You can also set this in the **Font** property.

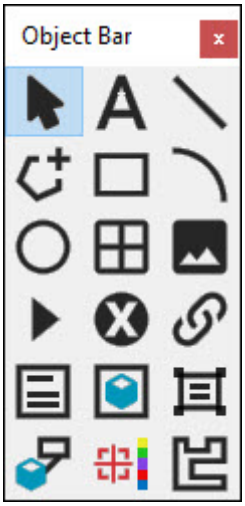
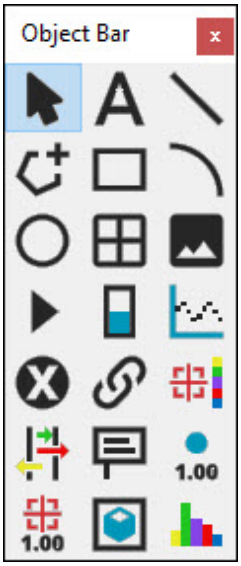
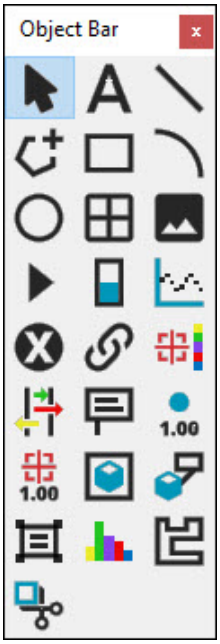
3. **Bold** - This button displays your object's text in boldface. You can also set this in the **Font** property.
4. **Italic** - This button displays your object's text in italics. You can also set this in the **Font** property.
5. **Left**, **Center**, and **Right** - These buttons align your text to the left side, center, or right side, of the object. You can set also this in the **Alignment** property.
6. This list defines the background color of your object. You can also set this in the **BackColor** property.
7. This list defines the foreground color of your object. Foreground sets any text and border colors. You can also set this in the **ForeColor** property.
8. **PenWidth** - This button sets the border or line's width. Clicking this multiple times switches between various options. You can also set this in the **LineWidth** property.
9. **PenStyle** - This button sets the style of a line (does not apply to borders). Clicking this multiple times switches the line style between a solid line and various dotted lines. You can also set this in the **LineStyle** property.
10. **Arrowhead** - This button places arrowheads at the end or ends of a **Line** object. Clicking this multiple times switches the arrow to different locations on either or both ends of the line. You can also set this in the **Arrowhead** property.
11. **Hatch Style** - This button sets a background pattern, called a "hatch style", on an object that supports hatch styles. Clicking this multiple times switches between the available hatch styles. You can also set this in the **HatchStyle** property.
12. **Shadow Style** - This button sets a shadow under the object if the object supports shadows. Clicking this multiple times switches between the available shadow styles. You can also set this in the **ShadowStyle** property.

The Object Bar

The **Object Bar** is a toolbar that lets you insert various objects into a template or objects and controls into a form. You can insert text, pictures, multi-media, dialog box controls, special PC-DMIS specific objects, and other items to help you create custom labels, reports, and forms.

This bar appears whenever you enter an editing environment such as the Report Template Editor, the Label Template Editor, the Custom Report Editor, or the Form Editor.

The **Object Bar** comes in one of these types, one for each editing environment:

Report Template Editor	Label Template Editor	Custom Report Editor	Form Editor
			

Adding Objects:

To add an object to a template, custom report, or form:

1. Inside the appropriate editor, access the **Object Bar**.
2. Click on the object you want to insert into your report.
3. Next, hold down the left mouse button and drag a rectangle on your current section.
4. Finally, release the mouse button.
5. Upon creation, the object is selected, as shown by small green squares, called handles, at each corner of the object.

A sample Text object showing handles

Selecting and Manipulating Objects:

To manipulate an object you have to first select it. To select an object, click on the object so that the green handles are showing.

To drag an object to a new location - Select the object. Click and hold the mouse on top of the object, then drag it to the new location. Release the mouse.

To resize an object - Select the object, move your mouse over one of the green handles until it changes to a resize cursor, a line with two arrows. Then click on the handle and drag the mouse to a new location. Release the mouse. The object expands or shrinks.

To change an object's properties - Select the object, right-click on the object so that the **Properties** dialog box appears. Select properties and change them as desired.

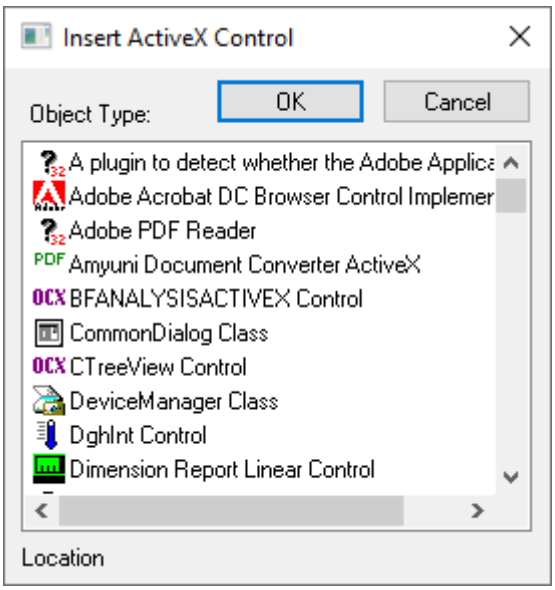
To align, group, ungroup, and perform other operations - Use the **Layout Bar** or the **Edit | Layout** and **Edit | Order** submenus.

Coding Objects to Change Dynamically

For some objects, you can use BASIC scripting to dynamically change certain properties of those objects. For example, you could use code to automatically populate plotted points on a Graph object, or you might want to use the Gauge object like a bar graph of some sort. For an example of how to do this, see the topic, "Example of Using BASIC Scripting to Dynamically Change the Gauge's Value".

ActiveX Object

The **ActiveX** Object inserts an ActiveX control into your template or form. Once it inserts the control, PC-DMIS displays the **Insert ActiveX Control** dialog box.



Insert ActiveX Control Dialog Box

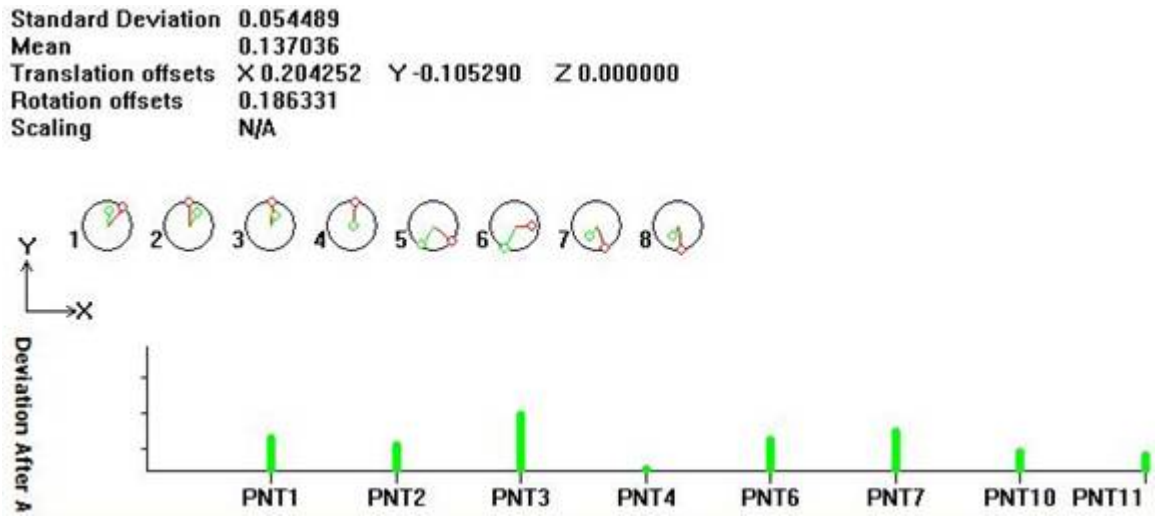
This dialog box allows you to select the type of control you want to insert from a list of known controls on your computer. The controls in the **Object Type** list are unique to each computer, based on the programs installed on your computer. ActiveX controls expand the possibilities of what you can do in a form or template. For example, you could insert a chart control and then with Visual BASIC code, pass measurement routine data to dynamically create the chart.

PC-DMIS ActiveX Controls

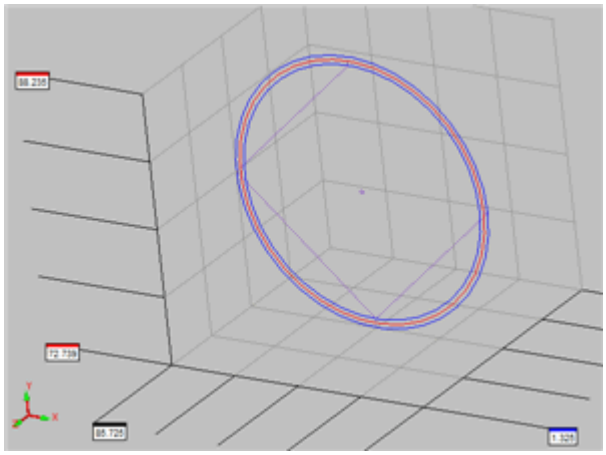
PC-DMIS ships with ActiveX controls, most of which are related to feature analysis. The feature analysis controls appear in certain cells of the **GridControlObject** on some of the standard labels that come with PC-DMIS. In general, the software uses these controls to display graphical analysis information. You can insert these into your own templates or forms from the **Insert ActiveX Control** dialog box.

The PC-DMIS ActiveX controls related to feature analysis are shown here with graphical examples:

BFANALYSISACTIVEX:



DIMANALYSISACTIVEX:

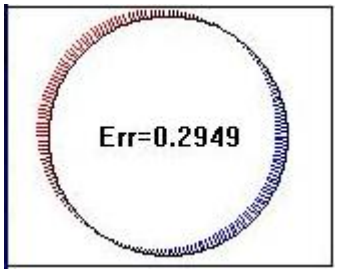


Dimension Report Linear:

Dimension Report Linear2 (Surfer NT Style Control):

Dimension Report Radial:

FeatureAnalysisActiveX:



The PC-DMIS ActiveX control that is not related to feature analysis is:

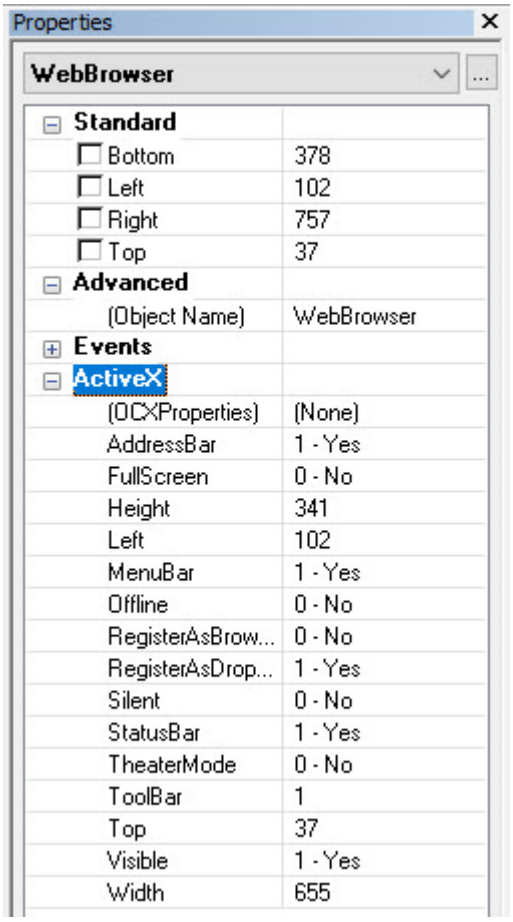
DataFileFormatControl
This control allows the interpretation of .dat files (logo.dat, elogo.dat, and header.dat) in your template-based reports. See "Some PC-DMIS ActiveX Controls" and "Using .DAT Files in Generated Reports".

As with any other object, the PC-DMIS ActiveX object also contains properties, both standard and unique, that you can modify using the **Properties** dialog box. The topic below contains information on accessing those properties.

Accessing an ActiveX Object's Methods and Properties from Basic

If you add a control, you can access its events and properties through the property sheet just like any other object.

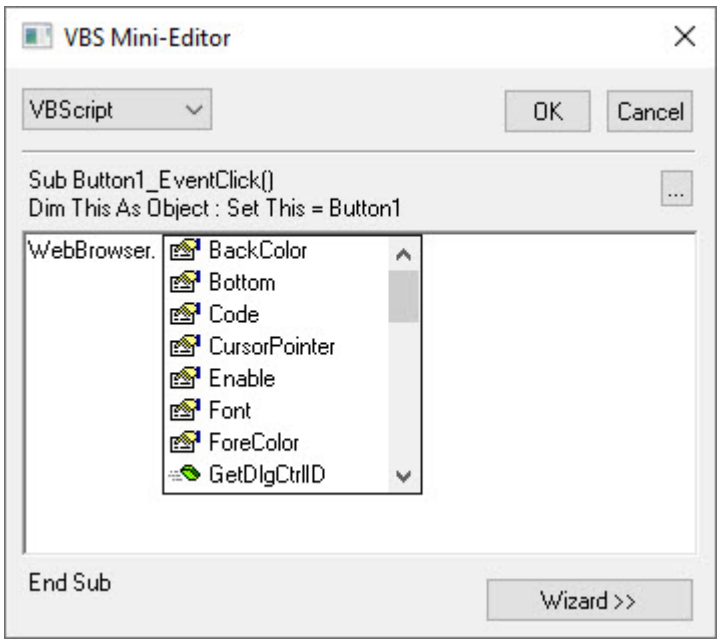
For example, suppose you add a **Microsoft Web Browser** control and set its `(Object Name)` property to `WebBrowser`. Because it is an ActiveX object, a new section of properties is visible, called **ActiveX**. That section contains a user-accessible list of variables and properties with their default values.



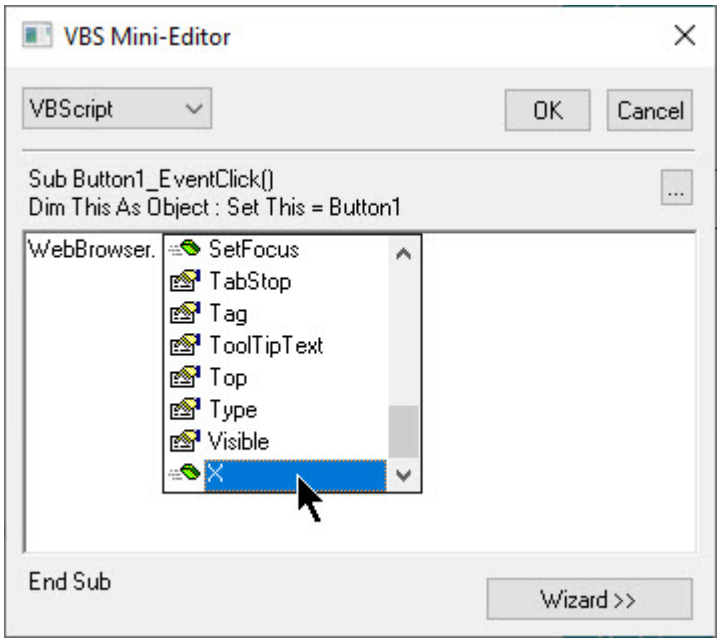
Microsoft Web Browser Control Active X Properties

To set properties for the ActiveX control inside the **VBS Mini-Editor** for an event handler

1. Type the name of the ActiveX control. For example, if you used the example above, you would type **WebBrowser**.
2. Then type a period after it. A window appears so that you can access the variables and methods.



3. Select the **X** from the list. PC-DMIS opens another popup window that lists a set of internal editable variables for the ActiveX control.



In this way you can access an Active X object's methods, properties, and events.

For information on the properties of the different PC-DMIS ActiveX controls, see the "Using PC-DMIS ActiveX Controls" topic in the PC-DMIS Core documentation.

Arc Object

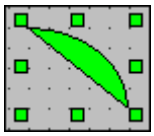
The **Arc** object inserts an elliptical arc into the template or form. A default arc doesn't have a fill color and has a beginning angle of 0 degrees and an ending angle of -90 degrees.

Along with the ability to resize the object, add a fill color, add a border color, and change other attributes, the following editable properties are included for arc objects:

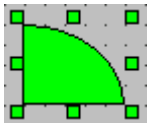
Angle1
This number sets the start angle for the arc. The default is 0 degrees.

Angle2
This number sets the end angle for the arc. The default is -90 degrees.

IsWedge
If set to NO (the default), PC-DMIS draws a filled arc without a wedge, like this:

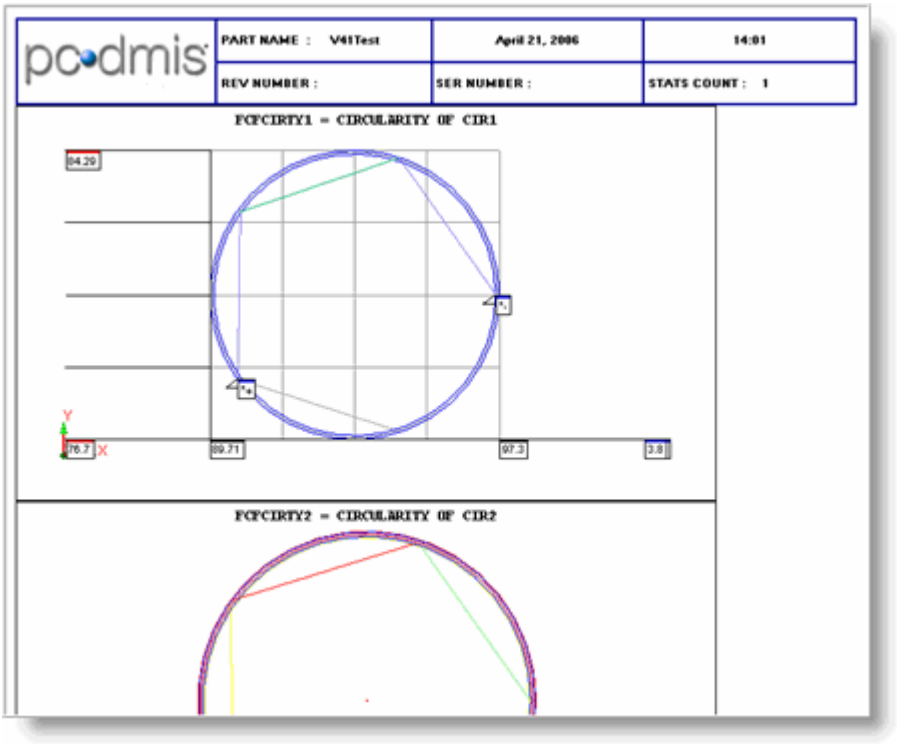


If set to YES, PC-DMIS draws a filled arc with a wedge like this:



AnalysisWindow Object

The **AnalysisWindow** object is used to insert a PC-DMIS Graphical Analysis window into a label template or into a custom report. Then, in the Report window, PC-DMIS displays a graphical analysis of dimensions that use a report template, using the label template with an analysis object in it.



Report Window showing the graphical analysis of the dimension.

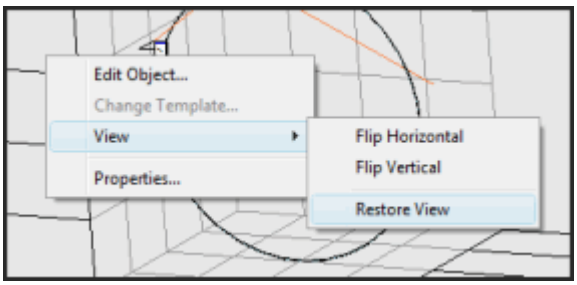
For information on the Analysis window, see "Inserting Commands Related to the Analysis Window" in the "Inserting Report Commands" chapter.

> The AnalysisWindow object functions only with Feature Control Frame dimensions. It does not function with Legacy dimensions.

Manipulating the Object

You can double-click on this object to *activate* it in the Report window. This lets you manipulate its display in the Report window on the fly. Once activated, you can use your mouse wheel to zoom in or out on the object's graphical display and change its rotation. Double-clicking outside of the object *deactivates* it. You can also right-click on the object in the Report window to access the **Graphical Analysis Options** dialog box described below.

If you need to revert a rotated graphical display in the object back to its original orientation, right-click on the object, and select **Restore View** from the **View** menu on the shortcut menu. This restores the orientation of the displayed graphical information in the object to its original vector.



Restore View Menu Item

The other menu items in the **View** menu, **Flip Horizontal** and **Flip Vertical**, flip the horizontal or vertical orientation of the object.

Using with Point Info Boxes in Custom Reports

When adding an **AnalysisWindow** object to a custom report, you can use the following two properties to control what point information is displayed and how it should look by defining the label template to use:

- **PointInfoLabel** - This property displays the **Browse Files** dialog box, showing the label templates in your Reporting subdirectory. You can use this dialog box to select the label template you would like PC-DMIS to use when creating Point Info boxes inside of the **AnalysisWindow** object. Once selected, the property will display the label name and any Point Info boxes that exist within the **AnalysisWindow** object will be immediately updated.

  PC-DMIS remembers the selected template and uses it for subsequent **AnalysisWindow** objects that are added into the Custom Report editor. The defined template is also stored in the `LastUsedPointInfoLabel` entry under the **Reporting** section of the PC-DMIS Settings Editor.

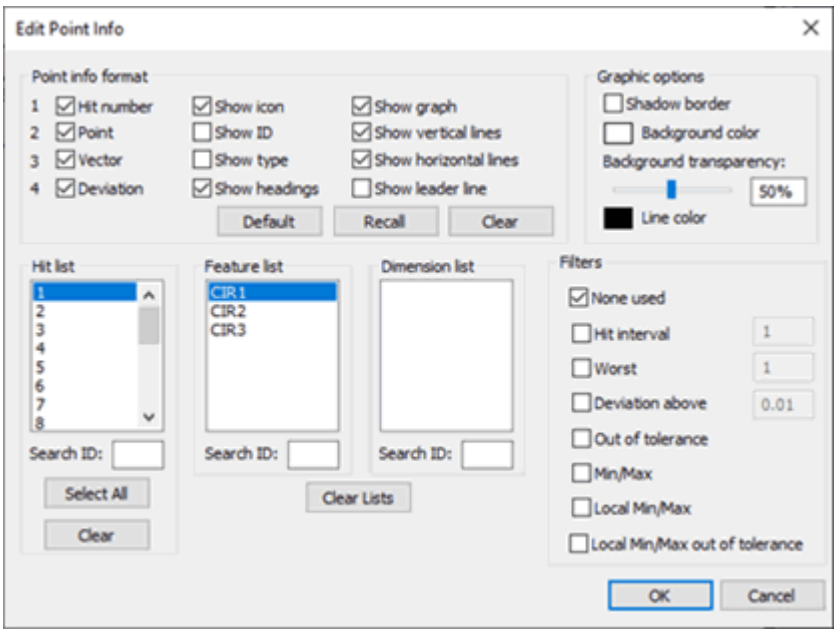
  If you do not define any label template in this property, any Point Info boxes used are generated the same as if you had dragged a **PointInfo** object onto the Custom Report editor. In that case, it does not use a label template, but is created internally.

  Some label templates to consider using are: ga_point_info.lbl, ga_point_info2.lbl, and ga_point_info_devonly.

  To clear this property, open the **Browse Files** dialog box, remove the filename listed in the **File name** box, and click **Cancel**.

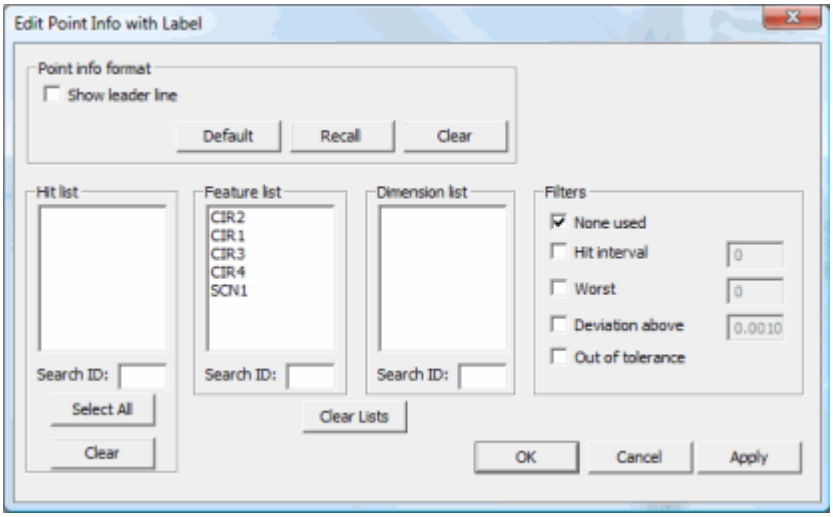
- **SelectPointInfo** - This property displays the **Edit Point Info** dialog box. That dialog box allows you to create Point Info boxes inside the **AnalysisWindow** object's window. This dialog box comes in two different styles depending on whether or not you have a label template defined in the `PointInfoLabel` property.

  If you do not have one defined, then the standard **Edit Point Info** dialog box appears:



Edit Point Info Dialog Box

If you do have one defined, then a slightly different **Edit Point Info with Label** dialog box appears:

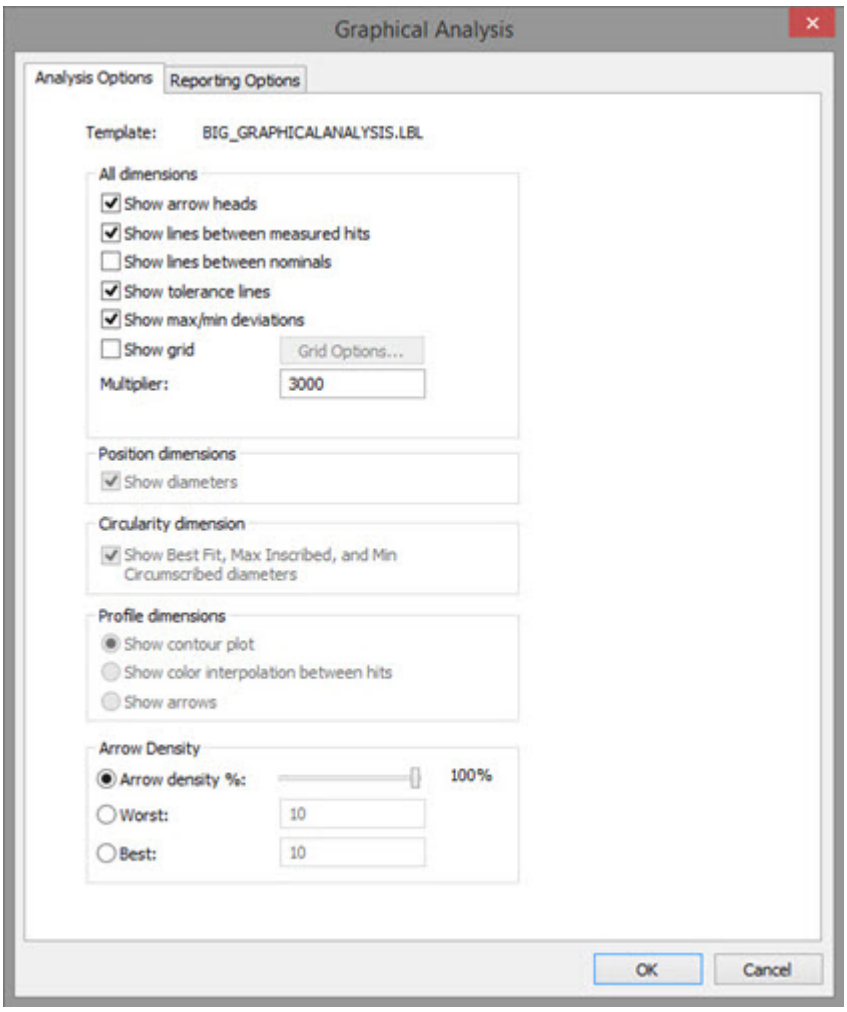


Edit Point Info with Label Dialog Box

These dialog boxes are similar, but if you have defined a label, you can only choose to hide or display leader lines using the **Point info format** area. This is because the other items do not apply if you are using a customized label template. For information on the available items in either of these dialog boxes, see the "Inserting Point Info Boxes" topic in the "Inserting Report Commands" chapter.

Changing Properties

To determine what this object displays, you can modify its many properties. You can also use the `(Settings Dialog)` property to access the **Graphical Analysis** dialog box inside the template editor. This dialog box lets you set many of the **AnalysisWindow** object's properties. There are two tabs:



Graphical Analysis Dialog Box

Note that when you open the **Graphical Analysis** dialog box from an analysis view, it shows the tab names as **Analysis Options** and **Display Options**. When you open the dialog box from within a report, it shows the tab names as **Analysis Options** and **Reporting Options**.

Analysis Options tab

For a description of the options on this tab, see "Dimension Options" in the "Inserting Report Commands" chapter. (The **Template** name near the top of the tab appears only if you edit an **AnalysisWindow** object within the Report window. For information on how to edit directly within the Report window, see "Edit Graphical Analysis".)

Reporting Options tab

For a description of the options on this tab, see the corresponding properties under the "Available Properties" subtopic below.

Here are the properties that are tied to the options on this tab:

Line color (measured) - see "MeasuredLineColor"

Line style (measured) - see "MeasuredLineStyle"

Line color (nominal) - see "NominalLineColor"

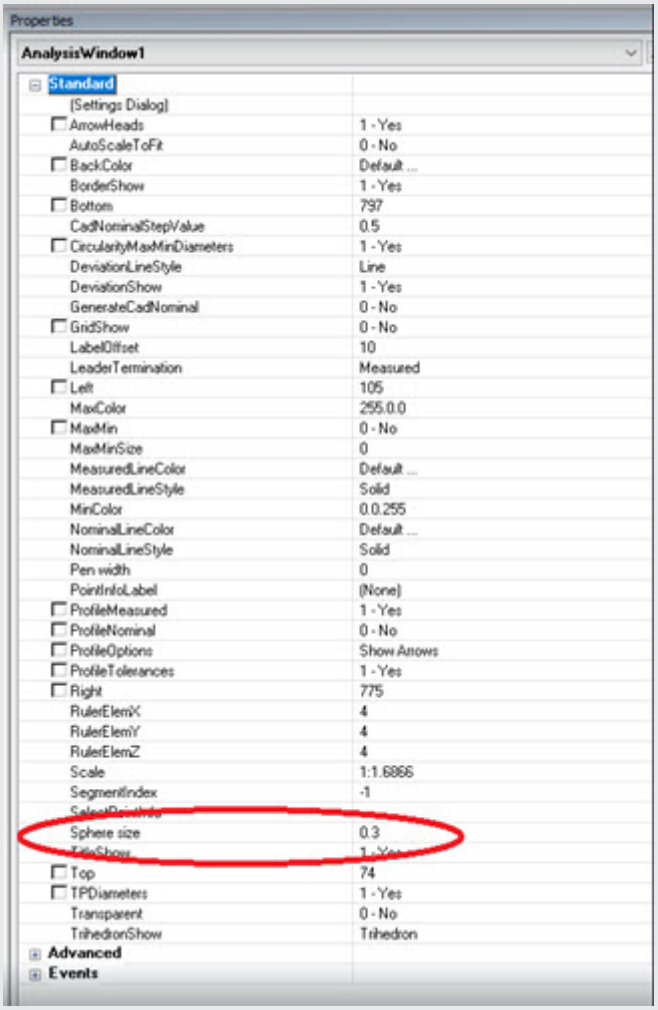
Line style (nominal) - see "NominalLineStyle"

Line style (deviation) - see "DeviationLineStyle"

Sphere size - PC-DMIS only enables this option when you are in **Line** mode. This value defines the diameter of the line's tip in mm. If you run PC-DMIS in inch units, the software converts the value you enter to mm for display purposes only.

You can set this value in the Settings Editor with the `AnalysisViewLollipopSphereSizeInMM` setting. For details on this setting, see "AnalysisViewLollipopSphereSizeInMM" in the PC-DMIS Settings Editor documentation.

You can view this and other properties of the analysis object on the report. For example, select **File | Reporting| New | Custom Report** and then create the analysis object on the report. Then right-click the object and select **Properties**. PC-DMIS lists all the properties related to the analysis object report:



Example of the analysis object report properties

Show trihedron - see "TrihedronShow"

Min/Max ball size - see "MaxMinSize"

Minimum ball color - see "MinColor"

Maximum ball color - see "MaxColor"

Background color - see "BackColor"

Line Size - see "PenWidth"

Leaderline termination - see "LeaderTermination"

Show Deviation - see "DeviationShow"

Show Title - see "TitleShow"

Show Border - see "BorderShow"

Available Properties

The available properties for this object are listed below:

All properties are used in Custom Reports unless otherwise stated. Those properties followed by an asterisk (*) are also used in Report Template Builder.

(Settings Dialog) *
Displays the **Graphical Analysis Options** dialog box for the selected AnalysisWindow object.

Arrow Best Value
Defines the number of best deviations for which to display arrows.

Arrow Density Options
Determines how to handle arrow density.

`0 - by Percent` - This uses the percentage defined in the **Dimension Analysis Options** dialog box's **Arrow density** slider.

`1 - by Worst` - This uses the `Arrow Worst Value` property.

`2 - by Best` - This uses the `Arrow Best Value` property.

Arrow Worst Value
Defines the number of worst deviations for which to display arrows.

ArrowHeads *
Same as the **Show Arrow Heads** check box in the dialog box.

AutoScaleToFit
Scales the information being analyzed so it fits within the boundaries of the object.

BackColor *
Common Properties

BorderShow *
Shows or hides the AnalysisWindow object's border.

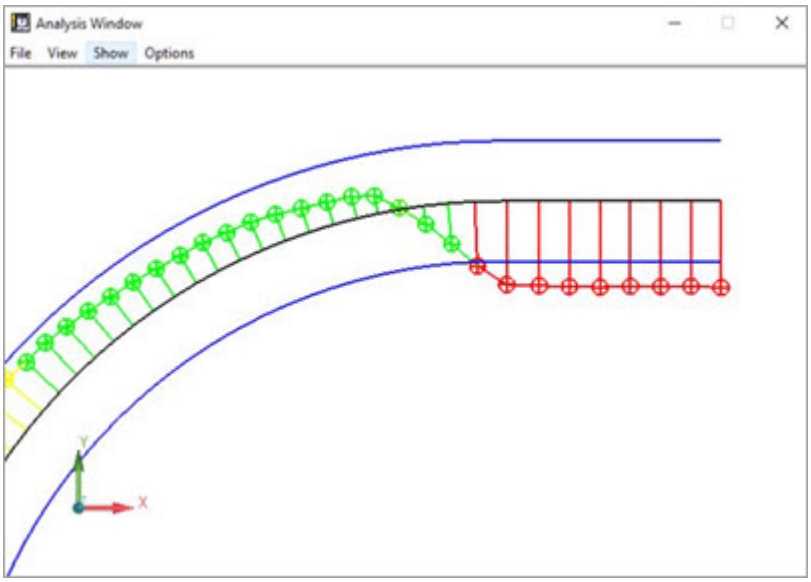
Bottom *
Common Properties

CadNominalStepValue
Defines the density of the points projected along the polyline to the CAD interpolation for the points inserted from the CAD model in the `GenerateCadNominal` property.

CircularityMaxMinDiameters *
Same as the **Show Best Fit, Max Inscribed, and Min Circumscribed Diameters** check box in the dialog box.

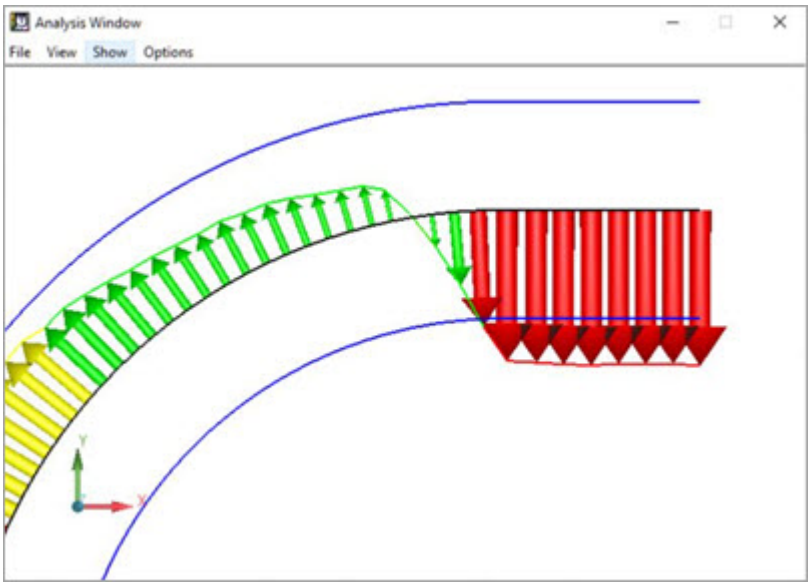
DeviationLineStyle
Defines the deviation line style.

`Line` - Deviation lines are drawn as simple lines that terminate with lollipop style heads.



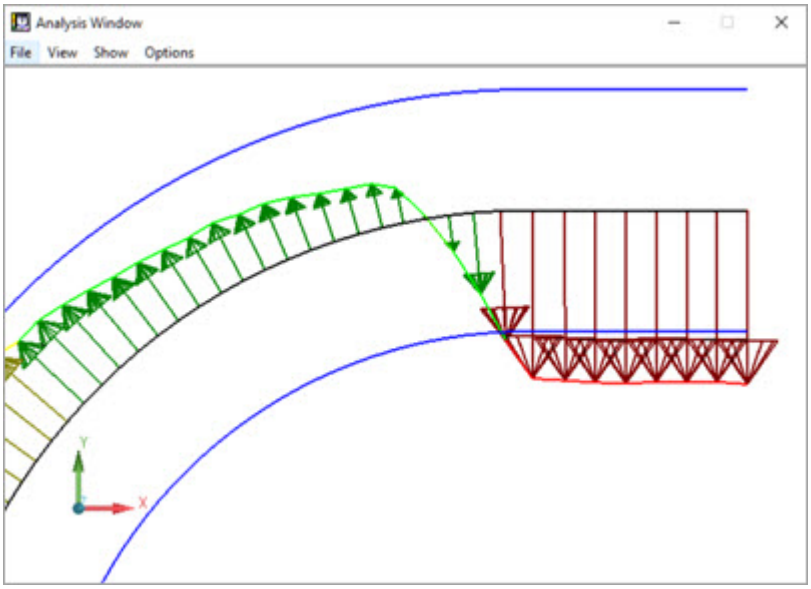
DeviationLineStyle - Line Option

`Cylinder` - Deviation lines are drawn as shaded cylinders that terminate with arrowheads.



DeviationLineStyle - Cylinder Option

`Unshaded Cylinder` - Deviation lines are drawn as wireframe cylinders that terminate with arrowheads. This option is only available from the **Display Options** tab of the **Graphical Analysis** dialog box.



DeviationLineStyle - Unshaded Cylinder Option

DeviationShow
Shows or hides the deviation lines. By default these are drawn as arrows.

Enable *
Common Properties

GenerateCadNominal
Generates a smoother nominal polyline by projecting additional points onto the CAD model as needed in order to see a line that more closely matches the CAD model. The density of these additional points can be increased or decreased using the `CadNominalStepValue` property.

GridShow *
Displays the 3D grid-like backdrop for the Analysis window.

LabelOffset
Positions a user-defined Point Info label at the specified offset distance from the hit location. The label to use is defined in the `PointInfoLabel` property.

LeaderTermination
Sets the termination location for any leader lines between Point Info boxes and their associated point locations:

`End of Arrow` - The leader lines point to the end-of-arrow locations.

`Nominal` - The leader lines point to the nominal point locations.

`Measured` - The leader lines point to the measured point locations.

Left *
Common Properties

MaxColor
Determines the sphere point color for maximum deviations. The default is red `(0.0.225)`.

MaxMin *
Same as the **Show Max/Min Deviations** check box in the dialog box.

MaxMinSize
Determines the diameter of the sphere point that marks the Max/Min points in the **AnalysisWindow** object. The default is `0`.

MeasuredLineColor
When lines are displayed between the measured hits, this determines the line color used.

MeasuredLineStyle
When lines are displayed between the measured hits, this determines the style used. You can choose `Solid` or `Dashed`.

MinColor
Determines the sphere point color for minimum deviations. The default is blue `(255.0.0)`.

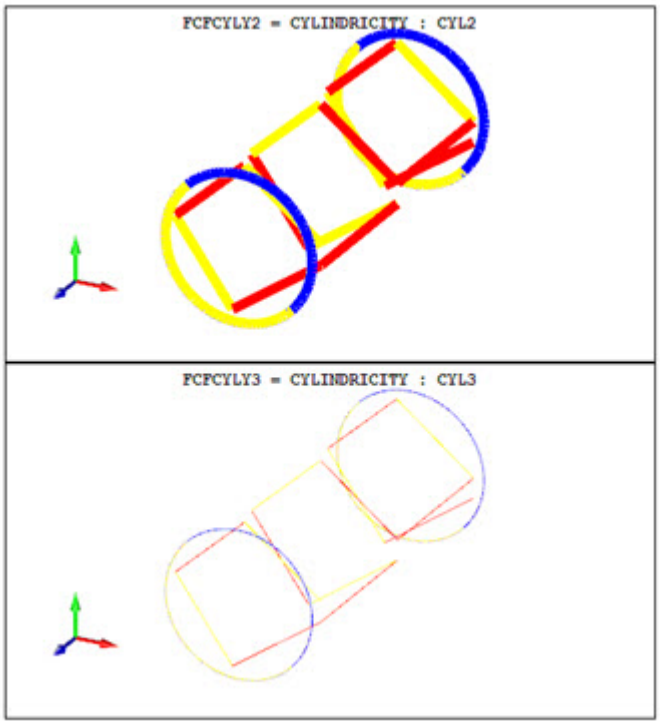
NominalLineColor
Determines the color of the nominal graphical line.

NominalLineStyle
Determines the style of the nominal graphical line. You can choose `Solid` or `Dashed`.

PC-DMIS Command Objects
Common Properties

PC-DMIS Reference
Common Properties

PenWidth
Sets the line size for the lines in the analysis object. The default is 0. The maximum line width is determined by your graphics card and driver.



PenWidth of 8 (top) and 0 (bottom)

PointInfoLabel
Selects the label template to use for Point Info boxes tied to the **AnalysisWindow** object. Use-defined labels in this object cannot be moved except by using the `LabelOffset` property.

ProfileMeasured *
Same as the **Show Lines Between Measured Hits** check box in the dialog box.

ProfileNominal *
Same as the **Show Lines Between Nominals** check box in the dialog box.

ProfileOptions *
Same as selecting one of the option buttons from the **Profile Dimensions** area in the dialog box.

ProfileTolerances *
Same as the **Show Tolerance Lines** check box in the dialog box.

Right *
Common Properties

RulerElemX
Same as **Number of elements** XYZ boxes discussed in the **Analysis Grid Options** dialog box.

RulerElemY
Same as **Number of elements** XYZ boxes discussed in the **Analysis Grid Options** dialog box.

RulerElemZ
Same as **Number of elements** XYZ boxes discussed in the **Analysis Grid Options** dialog box.

Scale
Scales the Analysis window by a specific ratio. The format is n1:n2 where n1 and n2 are two numbers. For example, a value of `1:3` would display the info in the Analysis window at three times its normal size, and a value of `2:1` would display the info at half its normal size. The property also displays the current zoom ratio if you zoom using the mouse buttons.

> The `AutoScaleToFit` property must be set to `No` for the `Scale` property to work.

SelectPointInfo
Displays a dialog box so that you can select the point information to display in Point Info boxes in the **AnalysisWindow** object.

TitleShow *
Shows or hides the text of the items being dimensioned at the top of the **AnalysisWindow** Object.

Top *
Common Properties

TPDiameters *
Same as the **Show Diameters** check box in the dialog box.

Transparent
Disables any color defined in the `BackColor` property and makes the object transparent, showing other objects behind it.

TrihedronShow
Shows or hides the trihedron symbol. Alternately, you can select **Ruler** to show a ruler along each axis.

Visible *
Common Properties

Border Object

The **Border** object inserts a rectangular border into the template or form. Along with the ability to resize the object, add a fill color, add a border color, and change other attributes, the following editable properties are included for border objects:

HighlightColor
This sets the color used for the `BorderStyle` property.

BorderStyle
This sets the 3D highlight style of the object's border. Options include `Normal`, `3D`, `Sunken` and `Raised`.

Image Object

The **Image** object allows you to insert a pre-created bitmap graphic into your template or form. When you click this icon and drag a box in the editor, a dialog box opens.

Bitmap Dialog Box

- The **OK** button inserts the bitmap loaded into the dialog box into the form or template.
- The **Cancel** button closes the dialog box without inserting any bitmap.
- The **Copy** button copies a bitmap to the Clipboard that you have already loaded into this dialog box through the **Load** button.
- The **Paste** button pastes a copied bitmap graphic from the Clipboard into the dialog box. After you paste a graphic you can select the **RLE** check box to compress the graphic when it's saved with the report.
- The **Load** button loads a graphic image (bitmap or JPEG file) into the dialog box. After you load the graphic, you can select the **Link** check box to link the bitmap to the template or form by its directory path. This means if the graphic gets updated, it will appear in its updated form inside the template or form.
- The **Clear** button clears the loaded bitmap out of the dialog box.
- The **Transparent** list specifies one color within the bitmap to make transparent.

Along with the ability to resize the object and change other attributes, the following editable properties are included for bitmap objects:

Bitmap
This contains the file name for the bitmap.

Layout
`Center`, `Left`, or `Right` is the alignment of the bitmap within the rectangle.

`Stretch` expands or shrinks the graphic to fit the size of the box.

`Size to Fit` automatically expands the rectangle size to fit the size of the graphic.

Button Object

The **Button** object inserts a button control into a form. You can choose the action the form will take when the button is clicked by modifying the `ButtonType` property.

Along with the ability to resize the object, change colors, and change other attributes, you can further customize button objects by using these properties:

Default
This determines how buttons will respond when you press the Enter key.

`YES` - The button responds to a pressed Enter key even if another button has the focus.

`NO` - If Enter is pressed only the button that has the focus will respond.

ButtonShape
This determines the look of the button. Options include:

`0 - Normal`: Changes shape to a standard button.

`1 - Property tab Inactive`: Changes shape to an unselected property sheet tab.

`2 - Property tab Active`: Changes shape to a selected property sheet tab.

Text
This determines the text that appears on the button.

Bitmap
This places a bitmap of your choice on top of the button face. It uses the same dialog box and properties as described in the "Bitmap Object".

Layout
This determines the position of the bitmap or text on the button's face. The values include:

`0 - Center`

`1 - Left`

`2 - Right`

`3 - Top`

`4 - Bottom`

ButtonType
This determines the action a button will take when it's clicked.

`0 - Cancel`: Closes the form and does nothing.

`1 - EventClick`: Executes a C++ or VBScript action.

`2 - Goto`: Goes to the Multi Document Interface (MDI) associated file.

`3 - Help`: Invokes the WinHelp() function using the HelpContextID property to open the proper topic.

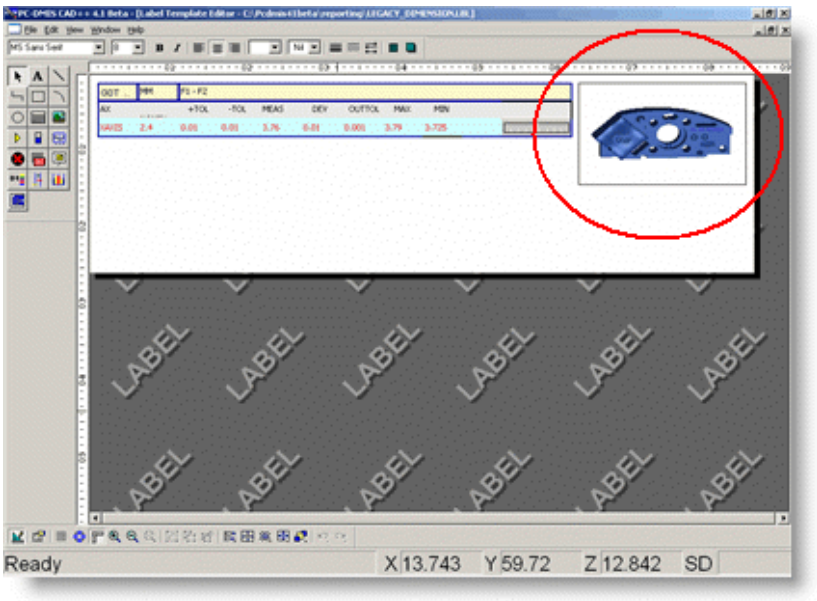
`4 - OK`: Records changes and closes the form.

`5 - Record`: Records changes and keeps the form open.

GotoPath
This determines the MDI associated file the form will go to when the button is clicked. This is mostly used to implement property sheet-like controls.

CadImageObject

The **CadImageObject** lets you display the CAD image inside of a label template or a form. When you select this item and drag a box in the editing environment, PC-DMIS inserts a **CadImageObject** into the editor, and the object displays a dummy CAD model image.



Label Template Editor showing an inserted CadImageObject

How you size the object in the editing environment determines the size of the object when actually displayed in the Report window or executed form.

The **CadImageObject** contains these properties:

Standard

Bottom
Defines the bottom location for the object in pixels from the top of the editor.

Left
Defines the left location for the object in pixels from the left of the editor.

Right
Defines the right location for the object in pixels from the left of the editor.

Show Border
Shows or hides the object's border.

- `ON` - Displays a border around the object.
- `OFF` - Does not draw a border.

Top
Defines the top location for the object in pixels from the top of the editor.

GraphicsOptions
Determines whether to hide or show labels on the CAD image.

- `0 - None` - Doesn't show labels.
- `1 - Labels` - Shows labels.

Hide all measured features
Shows or hides features on the CAD image.

`0 - No` - Shows features.

`1 - Yes` - Hides features.

`2 - Use entry setting` - (Default) Uses the entry default. This can override the `HideAllMeasuredFeaturesOnReport` entry located in the **Reporting** section of the PC-DMIS Settings Editor.

Hide all points
Shows or hides point features on the CAD image.

`0 - No` - Shows point features.

`1 - Yes` - (default) Hides point features.

`2-Use entry setting` - Uses the entry default. This can override the `HideAllPointsOnReport` entry located in the **Reporting** section of the PC-DMIS Settings Editor.

TrihedronShow
Shows or hides the trihedron on the CAD image.

`0 - No` - (default) Hides the trihedron.

`1 - Yes` - Shows the trihedron.

Advanced

(Object Code)
Defines the object's unique name.

Enable
Determines whether or not this object is enabled for modification in the form or template.

`YES` - Enables the object for modification.

`NO` - Does not enable the object for modification.

Visible
Determines whether or not this object is visible in the form or template.

`YES` - Shows the object.

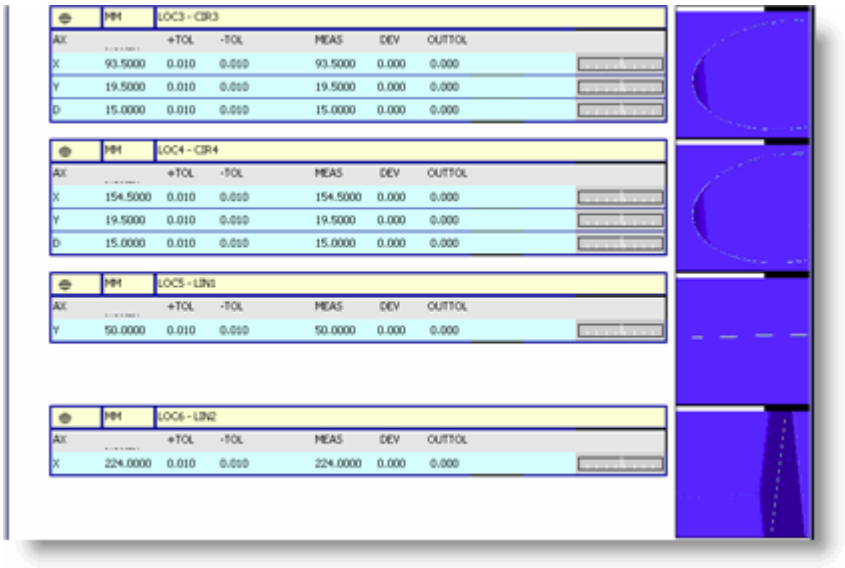
`NO` - Hides the object.

Events

EventReportData - This is the event that pulls report data from PC-DMIS into the template or form.

CadImageObject on Labels

You can use the **Rules Tree Editor** dialog box to display the label template in a report template for a particular dimension or feature. In such a case, the Report window displays the portion of the CAD model that contains that dimension or feature.

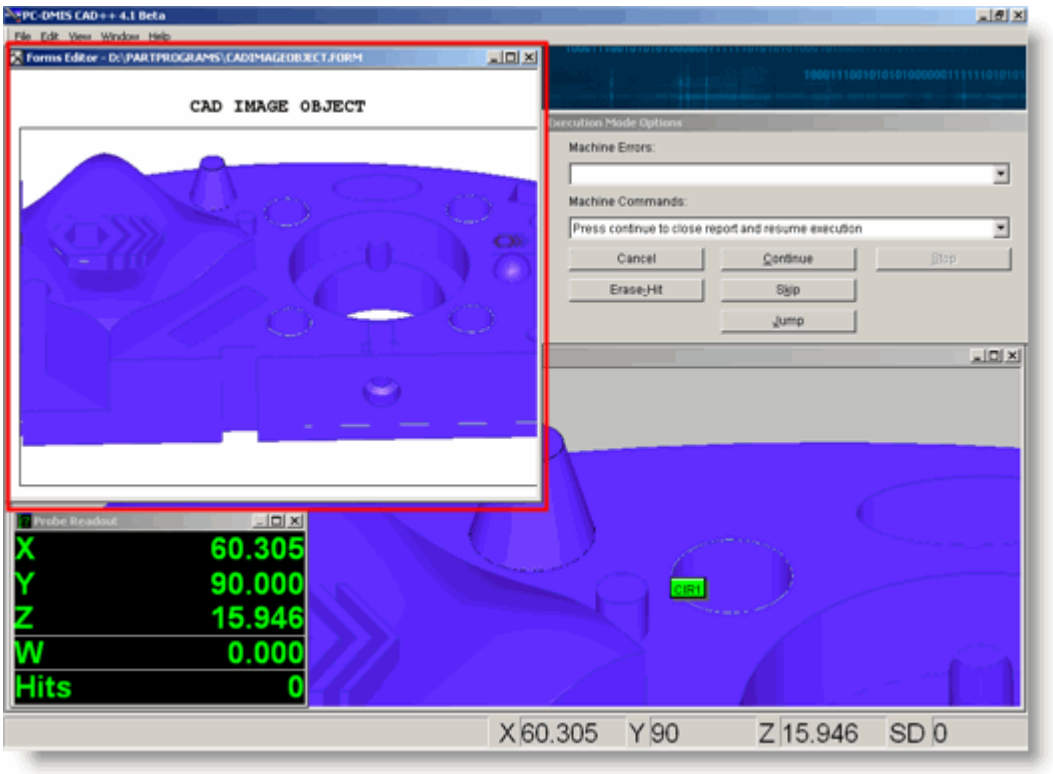


Report window showing a CadImageObject next to each displayed dimension

You can scale, scale to fit, rotate or translate this CAD image from the Report window. To do this, double-click on the CAD image to make it editable, and then manipulate the image just as you would the model in the Graphic Display window. To return to the Report window, storing your changes, double-click outside of the object.

CadImageObject on Forms

When it is used on a form, the CadImageObject is bound to the current viewset when PC-DMIS executes the form command. Once you execute the form (CTRL + E) or the measurement routine that calls the form, you cannot modify the CadImageObject.

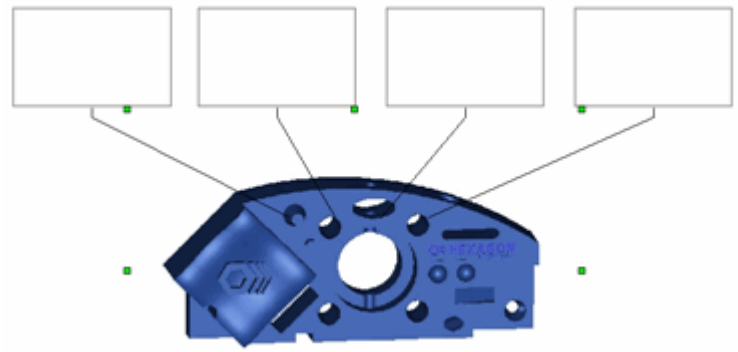


Form window showing a CadImageObject during measurement routine execution

For additional information on forms, see the "Creating Forms" topic in this chapter.

CadReportObject

The **CadReportObject** (CRO) lets you view your CAD drawing inside a finished report. When you initially drop the CRO into a report template, PC-DMIS automatically launches the **Label Layout Wizard**. After using this wizard to position the labels you want PC-DMIS to use, you will see a dummy graphic of the Hexagon test block, and if no rules are defined, dummy labels as well. It will look something like this:



A sample CRO inserted into a report template.

Remember, these dummy items are simply representations of what you want on your report. In the actual Report window, it shows the actual CAD model. The labels display a picture of the label template defined by the first rule in the rule tree of the CRO. If the label is not associated, and no rules apply, it draws the dummy label image.

How it Works in the Report Window:

In the Report window, your CAD display initially appears exactly as shown in your Graphic Display window when it finishes executing your measurement routine or at the time you redraw your report, except that it does not zoom in any closer. If you make modifications to a CRO in the Report window, PC-DMIS retains those changes to the CAD image on the report—even after subsequent measurement routine executions—until you right-click on the CAD view(s) inserted into your report and select **Remove Object Modifications** or until you select **File | Reporting | Clear Template Associated Data**.

While your Report window often shows your entire CAD drawing, it only shows labels and leader lines for those features you have specified in the **Rule Tree Editor**. For example, if your measurement routine has four measured circles and two measured lines, yet in the **Rule Tree Editor** your CRO displays labels only for the measured circles, then the report only shows label information for those circles, even though you may have measured the lines as well in your last execution.

Also, if a CRO is configured to display—using the **Label Count** list on the **Label Layout Wizard**—a fewer number of labels than the number of features it is specified to report on in the **Rule Tree Editor**, then the Report window displays additional instances of your CAD drawing on additional pages in the Report window. These additional images show leader lines and labels to any remaining features. This is especially helpful if your labels have a lot of information that may make your report look cluttered if you have more than one or two labels around a part drawing.

If you're using viewsets, your CAD display appears exactly as shown in your viewsets. PC-DMIS inserts a new CAD display on a new page of the report for each viewset encountered during execution. For information on viewsets, see "Working with Viewsets".

Rotating, Moving, and Zooming the CRO CAD Image

You can change the CRO CAD image orientation and zoom level in the Report window, or move it to a new location.

- To rotate the image, double-click on the object to "activate" it. Once activated, click and hold the wheel button. As you do this, drag the mouse. You can also press Ctrl and right-click while dragging the mouse.
- To zoom in or out of the image, double-click the object to "active" it. Once activated, rotate your mouse's wheel button to zoom in and out.
- To move the CRO, right-click the object to select it. Once selected, click the object and drag the mouse.

To cancel zoom or rotation changes, press the Esc key. The CRO is "deactivated" without applying your changes.

To save zoom or rotation changes, double-click outside of the CRO. The CRO is "deactivated", and the CAD image uses the new orientation and zoom level. If you are using the Feature-based label layout on the CRO, the entire report is reloaded with the new orientation and zoom level applied to the CAD image. If the CRO is from a `SNAPSHOT` command, PC-DMIS applies these same changes to your `SNAPSHOT` command in the Edit window.

To save movement changes, click outside of the selected object.

To remove pan, zoom, and rotate modifications for the CRO, right-click on the object in the Report window, and choose **Remove CAD Report Object Pan/Zoom/Rotate MOD**. Other mods (such as section cuts, various label definitions, solid or wire displays, and so forth) are not affected.

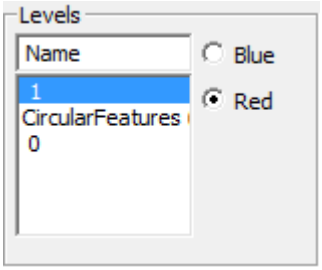
Switching Between Solid and Wireframe Modes

- In the Report window, you can do this by right-clicking the CRO and selecting the **Wireframe** menu item from the shortcut menu.
- In the Report Template Editor, you can do this by right-clicking the CRO, accessing the **Properties** dialog box and setting the **Wireframe** property to either YES or NO.

Additional Functionality in the Custom Report Editor

In the Custom Report Editor, if you activate a CRO, the **View Setup** icon on the **Graphic Modes** toolbar becomes available for selection. The **Edit | CAD Operation | View Setup** menu item also becomes available. Clicking this icon or selecting the menu item displays the **View Setup** dialog box. In this context, you can use this dialog box to only modify the display of the CRO just as you would manipulate the view of the Graphic Display window normally. See "Setting Up the Screen View" for information on using the **View Setup** dialog box to define views.

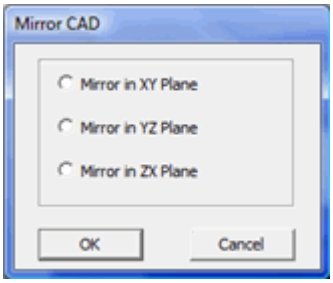
- **Choosing Wireframe or Solid:**
  You can display your CRO in Custom Report Editor as either a solid or wireframe by using the **View Setup** dialog box and marking or clearing the **Solid** check box. Alternately, with the object only selected but not activated, you can right-click on the CRO, select **Properties** from the shortcut menu and then set the **Wireframe** property to either YES or NO. PC-DMIS toggles between a solid or wireframe in your custom report.
- **Viewing Levels:**
  You can use the **View Setup** dialog box to select specific levels of CAD elements to be applied to the object in the Custom Report Editor. The **Levels** area lists any predefined CAD levels.



Levels area showing three levels

Select one or more levels and click **OK**, PC-DMIS displays the selected levels on the selected CRO in your custom report. For information on creating your own levels, see "Working with CAD Levels" in the "Editing the CAD Display" chapter.

- **Mirroring the CAD:**
  PC-DMIS lets you mirror the CAD image contained in that object in the Custom Report Editor. To do this, select the CRO, select **Edit | CAD Operation | Mirror**. The **Mirror CAD** dialog box appears:



Mirror CAD Dialog Box

This dialog box lets you create a mirror image of the part. Select the plane (axis) in which you want to mirror the part, and click **OK**. PC-DMIS displays the mirrored CAD model on the selected CRO. See "Transforming a CAD Model" for additional information on Mirroring CAD in the Graphic Display window.

See "Creating Custom Reports" for information on custom reports.

Using Image Caching

PC-DMIS lets you cache your part's image once and use the image cache later in reporting contexts instead of always using your measurement routine's associated .cad file. You may find this useful in situations where you have a very large .cad file that takes a long time to load into the Graphic Display window. Once you cache your images for a particular report, you won't need your .cad file to view that report.

To cache your image:

1. Close PC-DMIS and open the PC-DMIS Settings Editor.
2. Under the **Reporting** section, set the `UseImageCachingScheme` entry to **1**.
3. Close the PC-DMIS Settings Editor and restart PC-DMIS.
4. Load the measurement routine and the .cad file as usual.

5. Open the Report window and select a report template that uses the CRO.
6. Generate the report.
7. Make any desired modifications to the CRO items used in the report.
8. Print your report.

This process caches the images used in this particular report. Now if you delete (or rename) your .cad file to speed up the measurement routine opening process, this same report uses your cached images. Note that other reports that use the CRO are not cached however, and you need to load the .cad file as usual.

Properties:

Many of the properties in this table are covered in more detail in the "Label Layout Wizard" topic.

AutoZoom
Determines if the object automatically zooms and focuses attention on only that portion of the CAD model where label objects are currently displayed.

Draw wires in shaded mode
This determines whether or not PC-DMIS displays a wireframe of the CAD object on top of the shaded image when viewing the CAD model in a solid surface view.

Label Layout
This opens the **Label Layout Wizard**. This lets you quickly layout multiple labels around your CAD drawing. See "The Label Layout Wizard" for more information.

Leader Line ArrowHeader Height
Defines the height of the arrow on the leader line.

Leader Line ArrowHeader Style
Defines the style of the arrow on the leader line.

Leader Line Color
Defines the color of the leader line.

Leader Line ending at two features for angle
Determines if the leader line splits to point to both features.

Leader Line ending at two features for distance
Determines if the leader line splits to point to both features.

Leader Line Feature Mode
Determines whether the leader line points to the feature's centroid or to the usual drawn location.

Leader Line Style
Defines the style of the leader line.

Leader Line Termination
This draws the leader line from the label to either the **Measured** feature on the CAD or the **Nominal** feature on the CAD. See description in the Label Layout Wizard.

Leader Line Width
This defines the width of the leader line in pixels.

RuleTree
This opens the **Rules Tree Editor** dialog box. This dialog box lets you define conditions and actions for this object. See "About the Rule Tree Editor" for more information.

ShowBorder
This draws or hides a border around the CRO.

Single Click Rule Tree
This lets you browse and select a label template that PC-DMIS displays when you click on a label from the Report window.

TrihedronShow
Shows or hides the trihedron on the CAD image.

`0 - No` - (default) Hides the trihedron.

`1 - Yes` - Shows the trihedron.

Wireframe
This property lets you show the CRO in your report in either a wireframe (if set to `YES`) or solid surface view (if set to `NO`).

Hide all measured features
Shows or hides features on the CAD image.

`0 - No` - Shows features.

`1 - Yes` - Hides features.

`2 - Use entry setting` - (Default) Uses the entry default. This can override the `HideAllMeasuredFeaturesOnReport` entry located in the **Reporting** section of the PC-DMIS Settings Editor.

Hide all points
Shows or hides point features on the CAD image.

`0 - No` - Shows point features.

`1 - Yes` - (default) Hides point features.

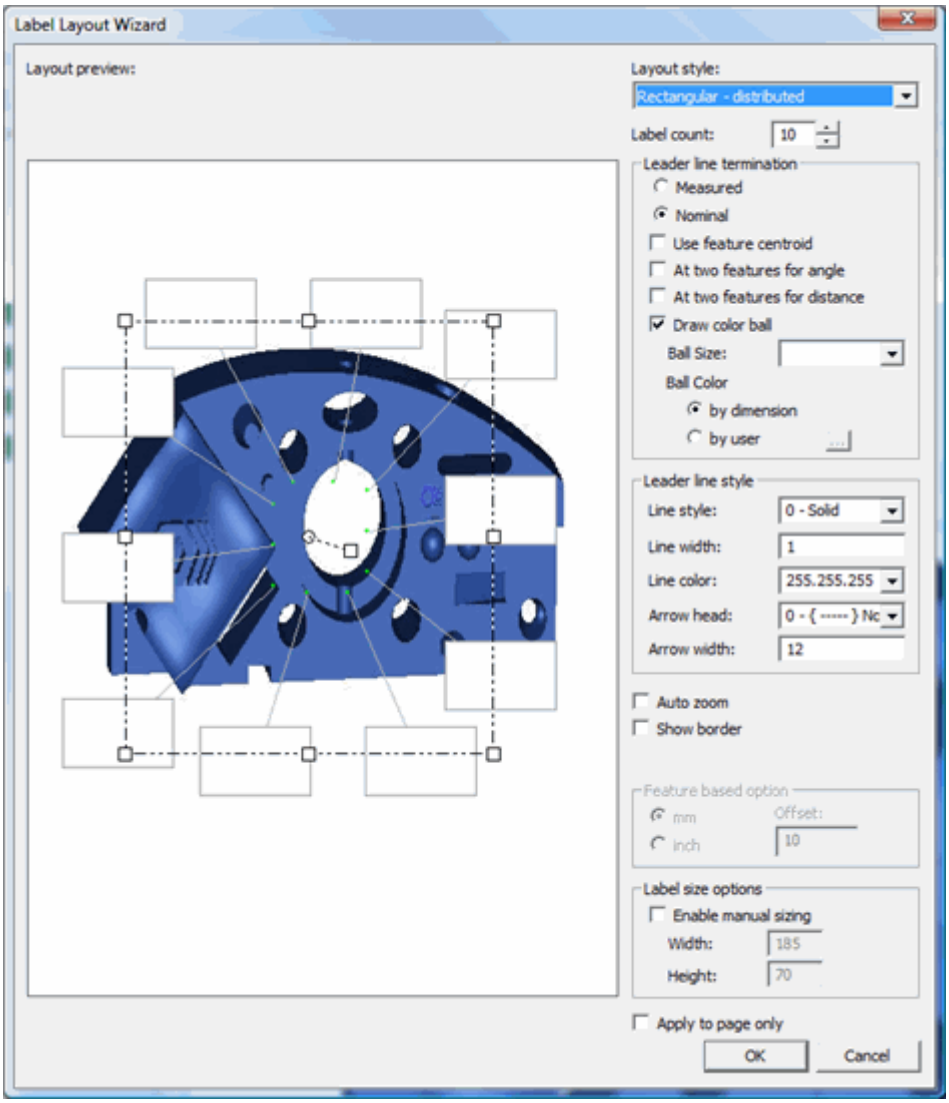
`2-Use entry setting` - Uses the entry default. This can override the `HideAllPointsOnReport` entry located in the **Reporting** section of the PC-DMIS Settings Editor.

Colormap is own page
Shows a colormap alone on its own page or with other objects. This property works with pointclouds (COP) Surface Colormaps; it does not affect Profile dimensions that have an embedded colormap.

`0 - No` - The colormap resides with other objects, and the report does not add a new page with only the colormap.

`1 - Yes` - (default) The report adds an additional page before the usual colormap page with only the colormap and its color key.

The Label Layout Wizard



Label Layout Wizard

The **Label Layout Wizard** is a tool that lets you quickly arrange multiple labels around your CAD drawing in the Report Template Editor and determine how the leader lines are drawn. In the Report Template Editor, this wizard appears automatically whenever you insert a **CadReportObject**. It also appears when you select the **(Settings Dialog)** property for a **CadReportObject**.

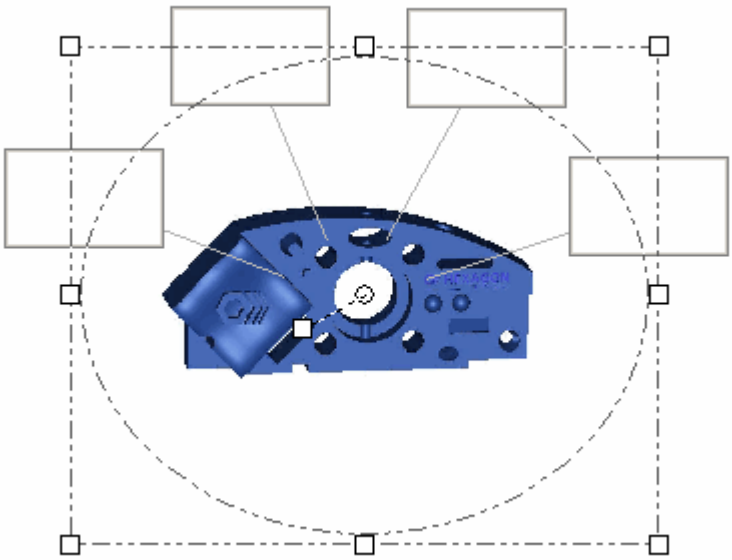
You can also utilize this wizard in the Report window. To access it, right-click on the **CadReportObject** in the Report window and select **Edit Object**. In the Report window, the wizard displays the real part model used with your measurement routine. In the Report Template Editor, however, it displays the Hexagon test block as a filler model.

Any changes you make inside this wizard will automatically change the appearance of the **CadReportObject**.

This wizard contains these two main parts:

- **Layout Preview** area
- **Settings** area

Layout Preview area



The **Layout Preview** area lets you preview how the layouts will appear around the **CadReportObject**.

Depending on how you want your labels to appear, you can rotate labels along either a rectangular or an elliptical path by clicking on the square, white handle, located at the center of the rectangle or ellipse and dragging it to a new location. You can click and drag any of the outer white handles to size this rotation path.

If you want to relocate your part drawing, click on it and drag it to a new location.

The following options are available:

Layout Style list
The drop-down list lets you control the spacing and rotate paths for the labels in the **Layout Preview** area. You can choose between these options:

Rectangular - Distributed - This creates a rectangular path that labels rotate around. The labels are distributed equally around the rectangle.

Rectangular - Packed - This creates a rectangular path that labels rotate around. The labels are close together along one side of the rectangle.

Elliptical - Distributed - This creates an elliptical path that labels rotate around. The labels are distributed equally around the ellipse.

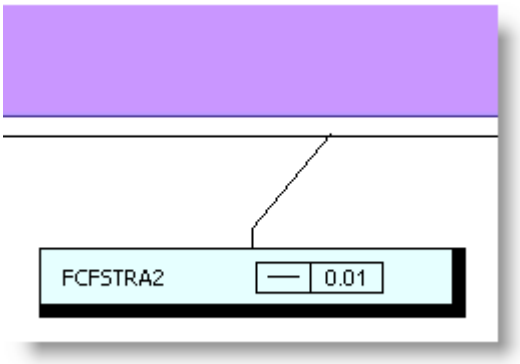
Elliptical - Packed - This creates an elliptical path that labels rotate around. The labels are close together along the ellipse.

Feature Based - This setting automatically places in the Report window all the feature labels around the part model in locations close to each feature's location. This is useful when you have, for example, a large amount of edge points and you want to display all their labels in one CAD drawing. Selecting this option disables the Label Count box and enables the Feature Based Option area where you can define the minimum leader line length.

Feature Based - Restricted - This acts the same as Feature Based except that it enables the Label Count box, allowing you to restrict the number of labels that will appear in the object in the Report window. If the object needs to display more labels than you have defined, the extra labels will appear in additional instances of the CAD drawing on new pages.

Leader Line Termination area
This area lets you choose where the leader lines from the labels will terminate on the CAD drawing.

Measured - This option terminates the leader line at the measured value's position.



Nominal - This option terminates the leader line at the nominal value's position.

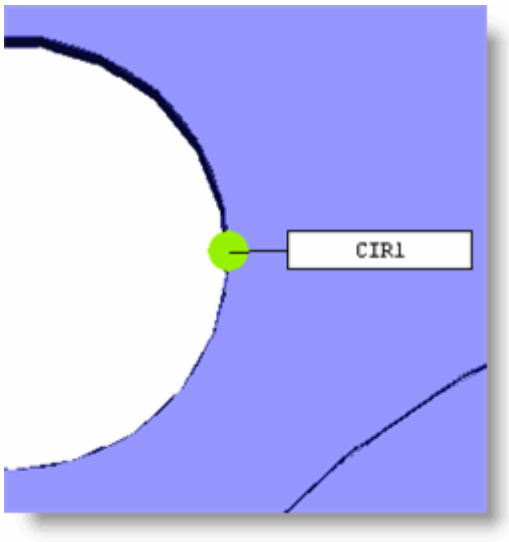
Use Feature Centroid - This check box terminates the leader lines to the centroid of the feature.

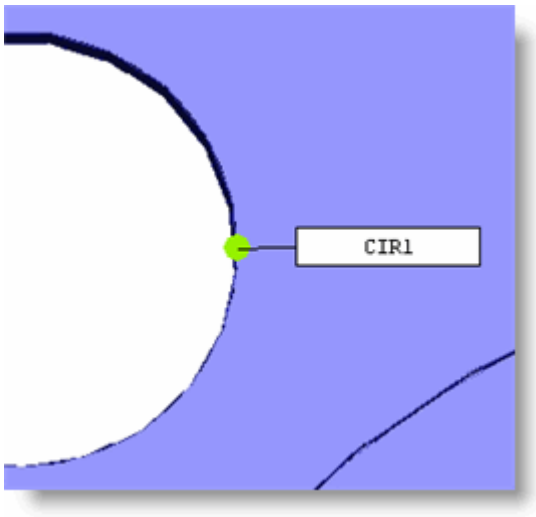
At two features for angle - This check box draws a single leader line from the label that then splits into two lines pointing to the two features making up an Angle dimension.

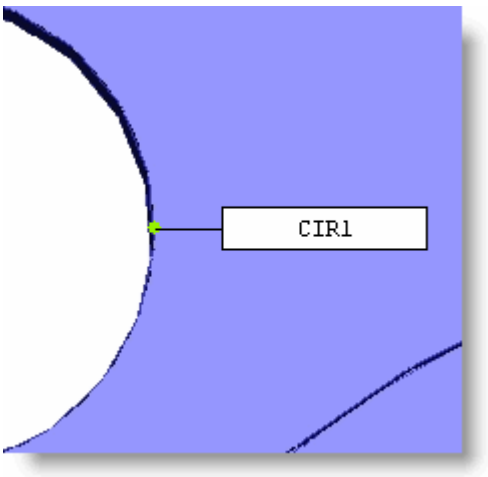
At two features for distance - This check box draws a single line from the label that then splits into two lines pointing to the two features making up a Distance dimension.

Draw Color Ball - This check box draws a colored ball where the leader line terminates. The color of the ball matches the tolerance color band for that feature's value.

Ball Size - This list determines the size of the colored ball drawn where the leader line terminates. It becomes enabled for selection when the **Draw Color Ball** check box is selected. You can choose between small, medium, and large. In addition, to define a more specific size, you can type a number in this box. This reflects the diameter of the color ball in pixels.







Ball Color by Dimension - This option colors leader line termination balls with the dimension's deviation color for the worst reported axis.

Ball Color by User - This option colors leader line termination balls with a single color that you select. To set the color, click on the **...** button and specify the color using the **Color** dialog box.

Leader Line Style area
This area lets you define what leader lines look like in the report.

Line Style - Defines the line style:

- *Solid*
- *Dash*
- *Dot*
- *Dash-Dot*
- *Dash-Dot-Dot*

Line styles only work at the default line width of 1. Wider lines will only use the Solid line style.

Line Width - Sets the line width in pixels.

Line Color - Defines the color of the line by setting RGB (Red, Green, Blue) values, consisting by three numbers separated by periods. You can either type in specific values for the colors, or you can select the **...** item from the list to access a standard **Color** dialog box and select your line color from that.

Arrow Head - Defines whether or not the leader lines have Arrow Heads.

- **None** - Lines do not have arrow heads.

- **Both** - Lines have an arrow head at each end.

Arrow Width - Defines the width of the arrow head in pixels

Show Border check box
This check box draws or removes a border around the entire CADReportObject inside the template editor or Report window.

Auto Zoom check box
This check box will automatically zoom in on the CAD display in the Report window in such a way that it focuses the attention on only that portion of the CAD model for which label objects are currently displayed.

Feature Based Option area
This area becomes enabled when you select **Feature Based** or **Feature Based-Restricted**. It contains two option buttons, **mm** and **inch**, and an **Offset** box. You can use the **Offset** box to define the minimum leader line length used with the labels. Depending on available room in the CadReportObject, the leader line length may grow, but it will never go below this minimum value.

Label Count box
This changes the maximum number of labels a report's page will display. It becomes disabled when you select **Feature Based**.

Label Size Options area
This area lets you switch between automatic or manual sizing of label objects. To make your label objects a specific size, click on **Enable Manual Sizing** and type in values in the **Width** and **Height** boxes.

Changing the labels sizes affects design purposes only. It does not affect label sizes in the actual Report window.

This area only appears when using the wizard in the Report Template Editor.

Apply to page only check box
Applies the changes made in the wizard to just the current page.

Colormaps and the CadReportObject

Laser scanners can scan and store thousands of points. PC-DMIS can then create colormap commands from these points. These colormaps can show deviations in a colored overlay. In the report, it does this with the CADReportObject (CRO).

For information on colormap commands, see the "Surface Colormap" and "Point Colormap" topics in the "PC-DMIS Laser" documentation.

For Standard Reports

The Report window shows a CRO on a new page for each marked colormap command. These CROs show the cad image with the colormap overlay. The colormap command defines what colormap appears.

For Custom Reports

You can add your own CRO and then drag a specific colormap command onto it.

In this way you can still view the colormaps but keep measured features and points hidden if **Hide all measured features** and **Hide all points** properties are set to **Yes**.

- For surface colormaps, if you associate more than one surface colormap to a CRO, it uses the last associated surface colormap.
- For point colormaps, you can combine point colormaps with other colormaps (point or surface) and show them all.
- If you drag and drop the same colormap onto the CRO where it's already been dropped, PC-DMIS removes that colormap from the CRO.
- If you do not associate any colormap with a CRO, PC-DMIS still attempts to show a colormap for that CRO. When it runs the `REPORT/CUSTOM` command it searches up from the insertion point. It uses the first colormap above the `REPORT/CUSTOM` command for the CRO. For information on the `REPORT/CUSTOM` command, see "The REPORT Command" in the "Inserting Report Commands" chapter.

For an example of dragging and dropping onto a CRO in a Custom Report, see "Dragging and Dropping Information into a Custom Report".

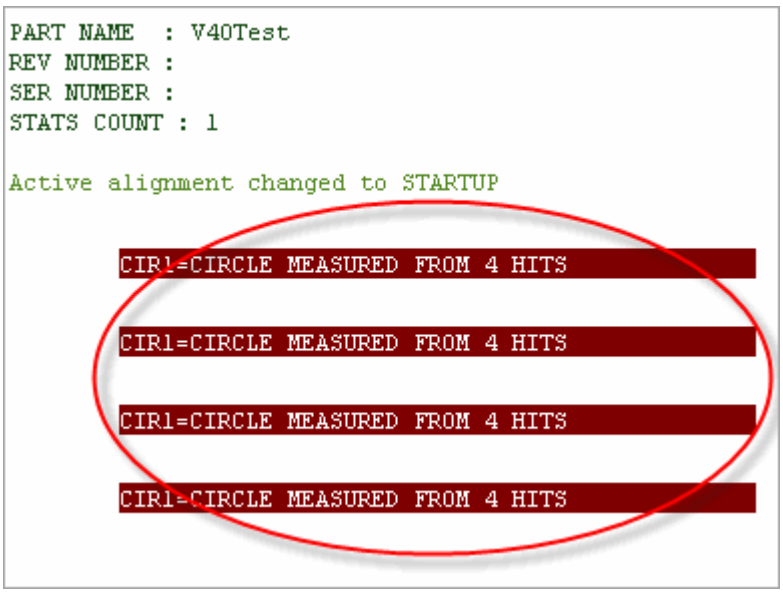
For information on Custom Reports in general, see "Creating Custom Reports".

CommandTextObject

The **CommandTextObject** merely acts as a container for a simple string of text that gives an overview of a feature or dimension.

When you use a template with this object with actual report data, PC-DMIS will display a string of text that shows, for a feature, the feature's text label, the feature's type, and how many hits were used to measure the feature. For a dimension, it shows the dimension's name and type and what units of measurement are used in the dimension.

A label that has only the **CommandTextObject** with actual report data might look something like this when displayed in the Report window:



Report showing the circle features generated from a label that uses a CommandTextObject.

Colors
Lets you change the colors of the text displayed. By default, PC-DMIS causes this object to use the same colors scheme as used in the Edit window's Command Mode, so if you click this property PC-DMIS displays a message box asking if you want to create an independent color set. Clicking **Yes** accesses the **Color Editor** dialog box. You can then use this editor to define a new color scheme for the selected CommandTextObject.

See "Defining Edit Window Colors" in "Setting Your Preferences" for information on how to use the **Color Editor**.

Empty Lines In Text Mode
This determines the number of blank lines to add between dimensions when dimensions are displayed as text.

Checkbutton Object

The **Checkbutton** object inserts a checkbutton object, or check box, into the form. Check boxes, unlike option buttons, are never mutually exclusive. You can select as many check boxes as desired (provided they are enabled for selection).

Along with the ability to resize the object, change colors, and change other attributes, you can further customize check boxes by using these properties:

AlignTextLeft
If set to YES, this moves the text to the left of the check box like this:

Text Here [ ]

If set to `NO`, this moves the text to the right of the check box like this:

[ ] Text Here

Bitmap
This allows you to specify a bitmap to use if `OwnerDrawn` is set to `TRUE` and if the check box is selected.

Additionally, the specified bitmap must be the same shape as a standard check box.

BitmapOffState
This allows you to specify a bitmap to use if `OwnerDrawn` is set to `TRUE` and if the check box is *not* selected.

Additionally, the specified bitmap must be the same shape as a standard check box.

HelpHotButton
Selecting `YES` shows a question mark help button next to the check box.

Clicking this button in run mode launches the PC-DMIS Online Help.

OwnerDrawn
This specifies how the form should draw the check box.

If set to `TRUE`, the check box gets drawn from a bitmap specified in the `Bitmap` and `Bitmap (Off)` properties.

If set to `FALSE`, the check box is drawn normally.

TriState
Enables (if set to `1 - YES`) or disables (if set to `0 - NO`) a check box's third state. Instead of just the checked or cleared states, this can add an additional third state to allow for cases when *a portion* of a selected option is true.

Text
The text specified here gets displayed on the check box control.

ComboBox Object

The **ComboBox** object inserts a combo list into your form. Using this list you can display options to select when you click the list in run mode.

Along with the ability to resize the object, change colors, and change other attributes, you can further customize this object by using these properties:

ComboType
This specifies the type of combo list. You can select either `Droplist` or `Dropdown`.

FlexHorizontal or **FlexVertical**
When the view is resized when viewing the form in run mode, you can elect to shift, expand, or keep objects proportional.

Selecting `Shift` moves the object to the right or down.

Selecting `Expand` enlarges the object right or down.

Selecting `Proportional` centers the object.

HelpHotButton
Selecting `YES` shows a question mark help button next to the check box.

Clicking this help button in run mode launches the PC-DMIS Online Help.

ListItems
This defines the list of items and the associated values (see the description of the **List Choices** dialog box described in the "Radiobutton Object").

NumDropped
The number specified determines how many list items the form displays when you click on the list in run mode.

Sort
Selecting `YES` sorts the list items in alphabetic order.

UseColors
This determines whether or not this control should use colors specified in `BackColor` and `TextColor` properties.

Dimension Color Key Object

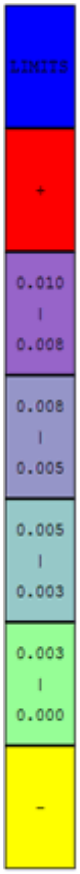
The **Dimension Color Key** object is used to drop a color key directly into the template. This color key depicts the colors that are used to represent the dimensional tolerance zones for analysis and histogram objects.

To see how these tolerance zones are defined or to change the colors representing each tolerance zone, see "Editing Dimension Colors" in the "Editing the CAD Display" chapter. Note that the **Decimal Places** value discussed in that topic does not affect the decimal places in the **Dimension Color Key** object here. That value only applies to the color bar in

the Graphic Display window. The decimal places for the **Dimension Color Key** object is hard coded to three decimal places.

Sample Horizontal Dimension Color Key Object

Sample Vertical Dimension Color Key Object



The following properties can be set:

BorderLines
This shows or hides the border line around the object.

LabelText
This shows or hides the overlay text on the color bar.

LimitsColor
This shows or hides the LIMITS color from the bar. This is the color used to draw the tolerance limits (tolerance lines).

ShadowStyle
This sets the location of the 3D shadow on the resulting object. Available locations are:

`Top/Left`, `Bottom/Right`, or `None`

DimensionInfo Object

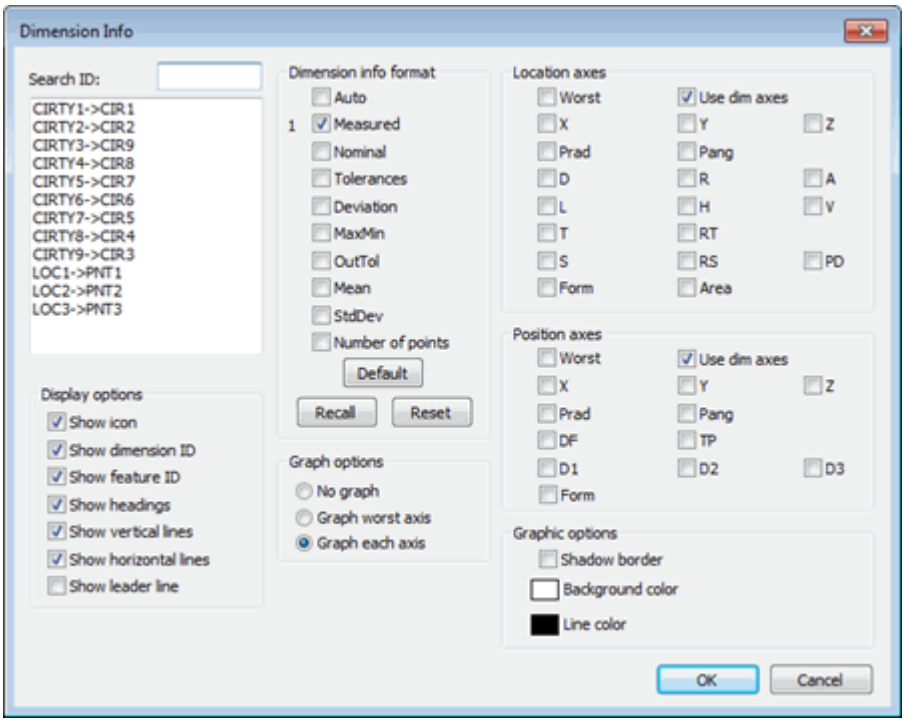
The **DimensionInfo** object acts as a container to hold individual dimension information in your report.

How to Use in a Label Template

From the Label Template editor's **Object Bar**, you can add and size a **DimensionInfo** object. Once you save the label template, you can use the **Rule Tree Editor** dialog box for a new or an existing report template to specify the dimensions for which PC-DMIS calls and displays that label template. For information on how to use the **Rule Tree Editor** dialog box, see the "About the Rule Tree Editor" topic in this chapter.

How to Use in a Custom Report

From the Custom Report editor's **Object Bar**, you can drag and drop a **DimensionInfo** object into the editor's canvas. The **Dimension Info** dialog box opens so that you can choose a dimension to associate with the object.



Dimension Info Dialog Box

From the dialog box, you can choose the dimension from the dimension list. Once you click **OK**, PC-DMIS displays a table on your report that contains the dimension information.

For complete information on how to use this dialog box to display dimension information, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

Below are the properties specific to this object:

For most properties, you can set them from the **Dimension Info** dialog box. You can open this dialog box at any time with the **(Settings Dialog)** property.

DimID
Defines the feature or dimension ID containing the hits that you want to display.

GraphOption
Defines if the graph is displayed and what it shows. The value can be one of the following numbers:

0 - Hides the graph.
1 - Graphs the worst axes.
2 - Graphs each axis.

LocOrderA
A value greater than 0 displays the A location axis and orders it according to the value given.

LocOrderAR
As above, except the AR (Area) axis is displayed and ordered.

LocOrderD
As above, except the D axis is displayed and ordered.

LocOrderForm
As above, except the Form axis is displayed and ordered.

LocOrderH
As above, except the H axis is displayed and ordered.

LocOrderL
As above, except the L axis is displayed and ordered.

LocOrderPAng
As above, except the Polar Angle axis is displayed and ordered.

LocOrderPD
As above, except the PD axis is displayed and ordered.

LocOrderPRad
As above, except the Polar Radius axis is displayed and ordered.

LocOrderR
As above, except the R axis is displayed and ordered.

LocOrderRS
As above, except the RS axis is displayed and ordered.

LocOrderRT
As above, except the RT axis is displayed and ordered.

LocOrderS
As above, except the S axis is displayed and ordered.

LocOrderT
As above, except the T axis is displayed and ordered.

LocOrderUseDimAxes
As above, except only the dimension axes for the location are displayed.

LocOrderV
As above, except the V axis is displayed and ordered.

LocOrderWorst
As above, except the Worst axis of the location is displayed.

LocOrderX
As above except the X axis is displayed and ordered.

LocOrderY
As above, except the Y axis is displayed and ordered.

LocOrderZ
As above, except the Z axis is displayed and ordered.

OrderAuto
A value greater than 0 displays and orders the default information automatically.

OrderDeviation
As above, except the deviation value is displayed and ordered.

OrderMaxMin
As above, except the maximum and minimum values are displayed and ordered.

OrderMean
As above, except the average of all the deviations is displayed and ordered.

OrderMeasured
As above, except the actual measured dimensions are displayed and ordered.

OrderNominal
As above, except the nominal values are displayed and ordered.

OrderNumberPoints
As above, except the number of points used to measure the feature or dimension are displayed and ordered.

OrderOutTol
As above, except the out of tolerance value is displayed and ordered.

OrderStdDev
As above, except the standard deviation is displayed and ordered.

OrderTolerances
As above, except the tolerances are displayed and ordered.

ShowDimensionID
A value of 1 displays the dimension ID. A value of 0 hides it.

ShowFeatureID
A value of 1 displays the feature ID. A value of 0 hides it.

ShowHeadings
A value of 1 displays the row and column headings. A value of 0 hides it.

TPOrderD1
A value greater than 0 displays and orders the diameter/width value of the first datum.

TPOrderD2
As above, except the second datum is displayed and ordered.

TPOrderD3
As above, except the third datum is displayed and ordered.

TPOrderDF
As above, except the diameter of the feature is displayed and ordered.

TPOrderForm
As above, except the integrated form dimension is displayed and ordered.

TPOrderPA
As above, except the Polar Angle of the position is displayed and ordered.

TPOrderPR
As above, except the Polar Radius of the position is displayed and ordered.

TPOrderTP
As above, except the position tolerance and its associated deviation are displayed and ordered.

TPOrderUseDimAxes
As above, except only the position dimension axes are displayed.

TPOrderWorst
As above, except only the worst axis of the position is displayed.

TPOrderX
As above, except the X axis of the position is displayed and ordered.

TPOrderY
As above, except the Y axis of the position is displayed and ordered.

TPOrderZ
As above, except the Z axis of the position is displayed and ordered.

EditBox Object

The **EditBox** object inserts an editable box into the form. You can use this object with Visual BASIC scripting to collect information from those running the form.

Along with the ability to resize the object, change colors, and change other attributes, you can further customize this object by using these properties:

Alignment
This aligns the text within the object. Available alignments are: `Left`, `Center`, and `Right`.

ReadOnly
Setting this to `YES` makes the text read-only and therefore not editable when the form is in run mode.

BorderDrawn
Setting this to `YES` draws a border around the control.

CaseOrPassword
This sets the style of the text in the box. You can select one of the following:

`0 - None:` doesn't change the display of the text.
`1 - Lower Case:` displays the case of the text in the box to all lower case.
`2 - Upper Case:` displays the case of the text in the box to all upper case.
`3 - Password:` masks each character of the text with an asterisk.

UseColors
If set to `YES` then the colors specified in `BackColor` and `TextColor` are displayed.

HelpHotButton
Selecting `YES` shows a question-mark help button next to the check box.

Clicking this help button in run mode launches the PC-DMIS online Help.

EditDblBox Object

The **EditDblBox** object inserts an edit box in the form that accepts variables of double types. This has all the properties of the **EditBox** object (see "EditBox Object") with the addition of the `MinimumEq` and `MaximumEq` properties:

MinimumEq
Minimum equation (this value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

MaximumEq
Maximum equation (this value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

EditLongBox Object

The **EditLongBox** object inserts an edit box in the form that only accepts Long numerical values. This has all the properties of the **EditBox** object (see "EditBox Object") with the addition of the `MinimumEq` and `MaximumEq` properties:

MinimumEq
Minimum equation (this value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

MaximumEq
Maximum equation (this value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

Ellipse Object

The **Ellipse** object inserts an ellipse into the form or template. The default ellipse is a circle with the same distance from the centroid of the circle to all points along the circumference; however, you can easily stretch the circle to form an ellipse.

Along with the ability to resize the object, add a fill color, add a border color, and change other attributes, the following editable property is included for ellipse objects:

HatchStyle
This property sets the type of pattern (or hash) to use to fill the resulting polygon. The available patterns are:

Horizontal

Vertical

Diagonal

Reverse Diagonal

Cross

Diagonal Cross

Frame Object

The **Frame** object inserts a frame into the form. You'll generally use this object to set off groups of similar options when designing your own dialog boxes inside your form.

Along with the ability to resize the object, add a fill color, add a border color, and change other attributes, the following editable properties are included for frame objects:

BorderColor
This sets the color of the border.

HiliteColor
This sets the color used for the `BorderStyle` property when you use `Raised` or `Sunken`.

BorderStyle
This sets the 3D highlight style of the object's border. Options include `Normal`, `3D`, `Sunken`, and `Raised`.

BorderWidth
This sets the width of the object's border in pixels.

Text
Text typed here determines the name used for the frame's title.

Alignment
This aligns the text within the object. Available alignments are `Left`, `Center`, and `Right`.

Gauge Object

The **Gauge** object inserts a gauge object into your form, label template, or custom report. This object is not valid in the Report Template Editor.

Default Gauge Object

- If the object's width is larger than the height, the percentile bar moves horizontally.

- If the object's height is larger than the width, the percentile bar moves vertically.

Along with the ability to resize the object, change colors, and change other attributes, you can cause the object's percentile bar to move by using these properties in conjunction with PC-DMIS commands or Visual BASIC code:

ValueEq
Value equation.

This is the current value of the gauge object. Once it is evaluated against the minimum and maximum ranges set in MinimumEQ and MaximumEQ, it displays a percentage number on the gauge.

It can be a set constant number, a value evaluated from a simple equation, or a value returned from a VBScript.

-$, $, CONST

MinimumEq
Minimum equation.

This defines what the 0% value is. If a value of 5 represents 0 percent on the gauge, you would set this property to 5.

This value can be a set constant number, a value evaluated from a simple equation, or a value returned from a VBScript.

-$, $, CONST

MaximumEq
Maximum equation.

This defines what the 100% value is. If a value of 10 represents 100 percent, you would set this property to 10.

This value can be a set constant number, a value evaluated from a simple equation, or a value returned from a VBScript.

-$, $, CONST

ShowText
This shows or hides the percentage number in the gauge.

Example of Using PARAM to Change the Gauge's Value

You can also use the REPORT/CUSTOM command with PARAM statements to change a gauge's value. Suppose you want to change the gauge percentage based on an input comment in PC-DMIS.

1. Add the object into one of the editors and size it as needed.

2. Right-click on it, choose **Properties**, and make a note the **Object Name** property.

3. Inside PC-DMIS, add code similar to the following:

```
C1=COMMENT/INPUT,NO,FULL SCREEN=NO,
   Type a percentage:
CS1=REPORT/CUSTOM, FILENAME=MyCustomReport, Section=-1
   PARAM/GAUGE1.VALUE=C1.INPUT
   PARAM/=
   ENDCUSTOM/
```

When you execute the above code, PC-DMIS stores a number value from the input comment into the C1.INPUT variable. It then looks for a custom report named "MyCustomReport".

Notice the PARAM statement. It modifies an object named **Gauge1**. It then uses a **VALUE** property to set the gauge's value equal to the value stored in the C1.INPUT variable.

Example of Using BASIC Scripting to Dynamically Change the Gauge's Value

You can also use BASIC code in the **VBS Mini-Editor** to dynamically change the value of the gauge at a given time. Suppose you want to create a form that moves the gauge from 0 to 100 percent based on how close a circle's diameter approaches its minimum and maximum allowed tolerances. You can do this by following this procedure:

Step 1 - Create the Form

1. Create a new form and make sure it's in Edit mode. First, you need to create a variable named MYDIAMETER that takes the diameter value of the circle.

2. Right-click on the grid and make sure **The Frame / The View** is selected from the object list on the **Properties** dialog box.

3. Expand the **Advanced** heading in the **Properties** dialog box. Select the value for the **LocalVariables** property. The **Local Variables** dialog box appears.

4. Click the **Add** button to add a new variable. Select **Double** from the **Type** list. In the **Name** box, type **MYDIAMETER**. Leave the **Value** as 0.

Local Variables dialog box showing the MYDIAMETER variable.

5. Click **OK** to return to the **Properties** dialog box. The MYDIAMETER variable is now added to the form.
6. Next, insert a **Gauge** object into the form.
7. Select the **Gauge** object and place and size it as desired. The vertical distance is the total distance the percentile bar can travel.
8. From the **Properties** dialog box, ensure that the added **Gauge** object is selected.
9. Again, under the **Advanced** heading on the **Properties** dialog box, click the **MaximumEq** property. The **VBS Mini-Editor** appears; much of it is unavailable for selection.
10. Select **CONST** from the list. This value allows you to define the gauge's 100% value.
11. Type the maximum allowed value for the circle's diameter. This example measures a one inch diameter circle with an allowed tolerance of .010. So, the maximum allowed value would be **1.010**.
12. Click **OK** to return to the **Properties** dialog box.
13. Click on the **MinimumEq** property. The **VBS Mini-Editor** re-appears. This value allows you to define the gauge's 0% value.
14. Select **CONST** from the list.
15. Type the minimum allowed value for the circle's diameter. Again, since this example measures a one inch diameter circle with an allowed tolerance of .010, the minimum allowed value would be **0.990**.
16. Click **OK** to return to the **Properties** dialog box. Now you need to give the actual value of the gauge the diameter of the circle.
17. With the gauge object still selected from the **Properties** dialog box, under the **Advanced** heading, select the **ValueEq** property. The **VBS Mini-Editor** appears; much of it is unavailable for selection.

18. In the list on the left, select **+ EQ ($)**.
19. In the list on the right, select the **<MYDIAMETER>**.

Using the VBS Mini-Editor to set the value of the gauge object

20. Click **OK** to close the **VBS Mini-Editor**.
21. Save the form. This example gives the form the name of *gaugetest.form*.

Step 2 - Create a Generic Circle Feature

1. Now, you need to create a generic circle feature inside PC-DMIS. Select PC-DMIS's Edit window and place it in Command Mode.
2. Insert a generic feature by typing **GENERIC** and then press the Tab key. Initially, a generic point feature appears in the Edit window with the feature's ID selected.
3. Press F9 to access the dialog box for this generic feature.
4. In the **Feature Type** area, select **Circle**.
5. In the **Data Type** area, select **Nominal Values**.
6. Type a name for the circle in the **Feature Name** box. This example uses CIR1 for the feature name.
7. Type the XYZ and IJK values for CIR1.
8. Select the **Diameter** option and give CIR1 a nominal diameter value of 1.
9. Change any other options as desired and then click **OK** when finished. The code for the generic circle in your Edit window should look something like this:

```
CIR1    =GENERIC/CIRCLE,DEPENDENT,RECT,OUT,$
         NOM/XYZ,1,1,0.95,$
         MEAS/XYZ,1,1,1,$
         NOM/IJK,0,0,1,$
         MEAS/IJK,0,0,1,$
         DIAMETER/1,0
```

Step 3 - Insert an INPUT Comment and Modify Generic Circle

1. Move your cursor *before* the CIR1 feature and insert an input comment to take a measured diameter value (since this example is done in Offline mode, the "measured" values are inserted manually). For example:

```
C1    =COMMENT/INPUT,Please type CIR1's measured
diameter:
```

2. Now go to the command block for CIR1 and on the last line where it says DIAMETER/1,0 change the second parameter, the measured diameter of 0, to C1.INPUT. This assigns the value from the input comment to CIR1's measured diameter.

```
DIAMETER/1,C1.INPUT
```

3. Now, move the cursor *after* the CIR1 feature and embed the saved form by selecting the **Insert | Report Command | Form** menu option.

Step 4 - Insert a FORM/FILENAME Command and Execute

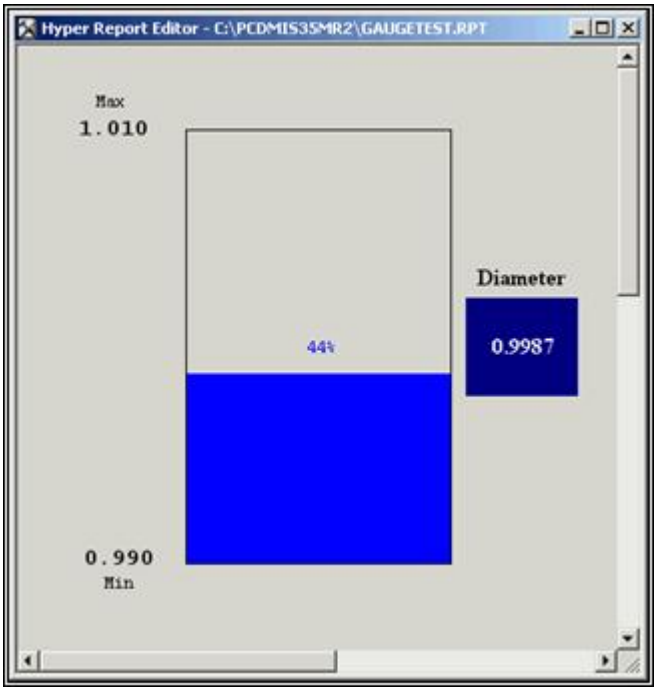
1. Type `FORM` and press the Tab key to insert a `FORM/FILENAME` command block into the Edit window.

2. Now, in the `FORM/FILENAME` command where it says `PARAM/=` , place your cursor immediately before the equals sign and type `MYDIAMETER` . Then move the cursor immediately after the equals sign and type `CIR1.DIAMETER` . The code to embed your form should look something like this:

```
CS1    =FORM/FILENAME=C:\PCDMIS35\GAUGETEST.FORM,
AUTOPRINT=NO
        PARAM/MYDIAMETER=CIR1.DIAMETER
        PARAM/=
        ENDFORM/
```

3. Mark the newly added commands and then execute the measurement routine. The diameter for CIR1 gets passed as a parameter into the form and the gauge object's percentile dynamically changes depending on the circle's measured diameter.

For example, if the measured diameter for CIR1 was .9987 you would get a form that looks something like this:

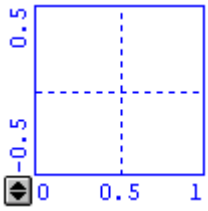


Example of a form showing the gauge object dynamically linked to a feature's measured diameter.

Graph Object

The **Graph** object inserts a graph into your form, custom report, or label template with the ability to dynamically update its data. This object is not available in the Report Template Editor.

The **Graph** object allows you to store and then display an array of data points. You can use X and Y values to specify the points of data on the graph.



Default Graph Object

Along with the ability to resize the object, change colors, and change other attributes, you can cause graph objects to update their data dynamically by using these properties in conjunction with PC-DMIS commands or Visual BASIC code:

ClearAllPoint
If set to `1`, all data points are cleared from the graph the next time your form enters run mode.

If set to `0`, all data points remain in the graph.

Clockwise
Setting this to `NO` reverses the Y axis, making the top negative and the bottom positive.

ConnectPoints
Setting this to `YES` draws lines between the data points on the graph, visually connecting them together.

DataPointColor
This changes the colors used on the X, Y data points and on any lines drawn between points.

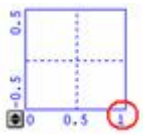
EnablePoint
If set to `1`, your graph accepts the addition of new data points.

If set to `0`, your graph rejects the addition of new data points.

LineWidth
This changes the diameter (in pixels) of the data point.

XDivisions
This divides the X axis of the graph, which essentially adds the specified number of columns.

XMaxEq
This is the X-axis maximum equation. The value used determines the ending value making up the X axis.



You can use a simple equation, or evaluate the data from a VBScript.

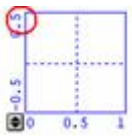
XMinEq
This is the X-axis minimum equation. The value used determines the beginning value making up the X axis.

You can use a simple equation, or evaluate the data from a VBScript.

XPointEq
This is the X value for your data point on the graph. You can use a simple equation, or evaluate the data from a VBScript.

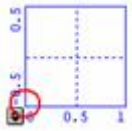
YDivisions
This divides the Y axis of the graph, which essentially adds the specified number of rows.

YMaxEq
This is the Y-axis maximum equation. The value used determines the ending value making up the Y axis.



You can use a simple equation, or evaluate the data from a VBScript.

YMinEq
This is the Y-axis minimum equation. The value used determines the beginning value making up the Y axis.



You can use a simple equation, or evaluate the data from a VBScript.

YPointEq
This is the Y value for your data point on the graph. You can use a simple equation, or evaluate the data from a VBScript.

Click on the up and down arrows near the graph's origin to re-scale the Y axis. These scaling arrows are only visible in the legacy Hyper View report editor and in the Forms editor.

Example of using PARAM to Display Data on the Graph

You can also use the REPORT/CUSTOM command with PARAM statements to specify the graph's data points.

1. Add the **Graph** object into one of the editors, and size it as needed.
2. Right-click on it, choose **Properties**, and make a note of the **Object Name** property value.
3. Set **XDivisions** to 5.
4. Set **YDivisions** to 5.
5. Set **DataPointColor** property to red (255.0.0).
6. Set **LineWidth** to 8.
7. Inside PC-DMIS, add code similar to the following:

```
CS1   =REPORT/CUSTOM, FILENAME=MyCustomReport,
Section=-1
       PARAM/GRAPH1.SETVALUES=0
       PARAM/GRAPH1.XVALUE=0.25
       PARAM/GRAPH1.YVALUE=-0.4
       PARAM/GRAPH1.SETVALUES=1
       PARAM/GRAPH1.XVALUE=0.65
       PARAM/GRAPH1.YVALUE=0.-0.3
       PARAM/GRAPH1.SETVALUES=1
       PARAM/GRAPH1.XVALUE=0.75
       PARAM/GRAPH1.YVALUE=0.45
       PARAM/GRAPH1.SETVALUES=1
       PARAM/=
```

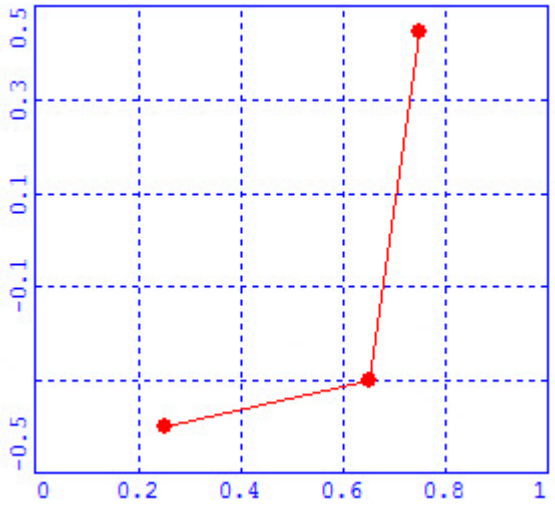
With the above code, PC-DMIS looks for a custom report named "MyCustomReport". Notice the PARAM statements in the REPORT command. They each modify an object named Graph1, and they use these properties to modify the graph:

XVALUE - This defines the X axis location of the data point on the graph.

YVALUE - This defines the Y axis location of the data point on the graph.

SETVALUES - This can be either 0 or 1. A value of 0 resets the list of data points. A value of 1 adds the data point to the list.

The resulting graph looks like this:



Example of Using BASIC Scripting to Dynamically Display Data on the Graph

You can also use BASIC code in the **VBS Mini-Editor** to dynamically display data points on the graph. Suppose you want to have a script that creates the data points automatically when a form is launched. You can do this by following this procedure:

Step 1 - Create the Form

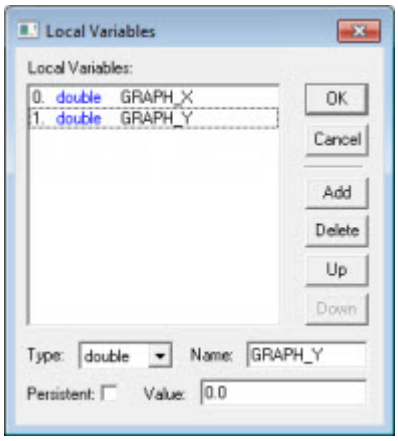
1. Create a new form and make sure it's in edit mode. First, you need to create some variables to take the X and Y values for each data point.
2. Create a **Graph** object, add it to the canvas, and size it as needed. Name it **Graph1**.
3. Right-click on it and choose **Properties**.
4. Set **XDivisions** to **5**.
5. Set **YDivisions** to **5**.
6. Set **DataPointColor** property to red (**255.0.0**).
7. Set **LineWidth** to **8**.
8. Set **YMaxEq** to **CONST** and **1.0**.
9. Set **YMinEQ** to **CONST** and **-1.0**.

Step 2 - Create the Variables

1. Right-click on the grid and make sure **The Frame / The View** is selected from the object list on the **Properties** dialog box.
2. In **LocalVariables** click the value to display the **Local Variables** dialog box.

3. Click **Add** to add a new local variable. Select **Double** from the Type list. In the **Name** box, type **GRAPH_X**. Leave the **Value** as 0.

4. Add another local variable in the same manner and give it a name of **GRAPH_Y**.



Local Variables dialog box showing the two variables

5. Click **OK** to return to the **Properties** dialog box. The two variables are now added to the form.

6. From the **Properties** dialog box, choose **Graph1**.

7. In **XPointEq** click the value. In the **VBS-Mini Editor**, from the left list, choose **+ EQ ($)** and from the right list, choose **<GRAPH_X>**. Then click **OK**.

8. Do the same for **YPointEq** and set it to **<GRAPH_Y>**.

Step 3 - Add the Script

1. In the **EventInitialize** property, click the value to display the **VBS Mini-Editor**.

2. Type this code in the editor:

```
GRAPH_X = 0.1
GRAPH_Y = -0.4
This.Refresh
GRAPH_X = 0.25
GRAPH_Y = 0.5
This.Refresh
GRAPH_X = 0.75
GRAPH_Y = -0.5
This.Refresh
GRAPH_X = 0.85
GRAPH_Y = 0.45
This.Refresh
```

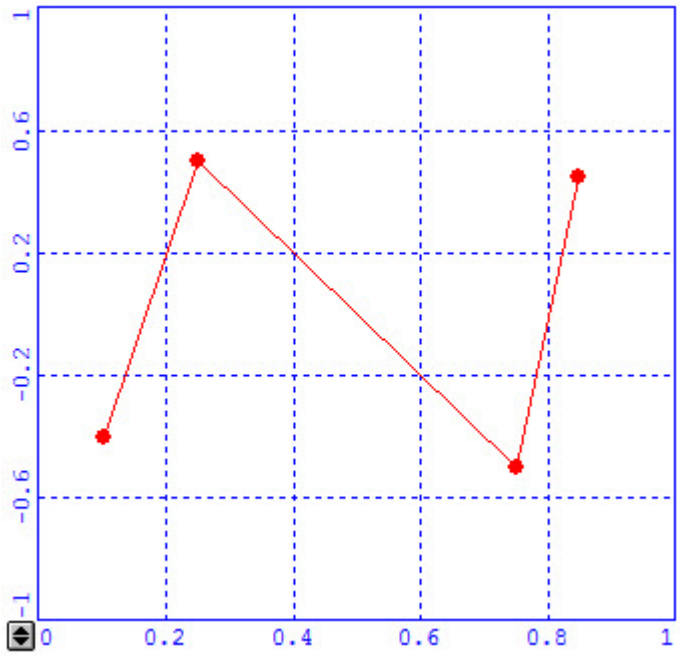
3. Click **OK**.
4. Save the form. This example gives the form the name of *graphtest.form*.

Step 4 - Insert a FORM/FILENAME Command and Execute

In the Edit window's Command Mode, type **FORM** and press Tab to insert a `FORM/FILENAME` command block into the Edit window. In the command block, for `FILENAME`, type a reference to the graphtest.form file that you saved in the above step.

Ensure the commands are all marked and execute the measurement routine.

When the form command is executed, and the form appears, the graph initializes. The script then runs and creates the data points on the graph like this:



Without Using Local Variables

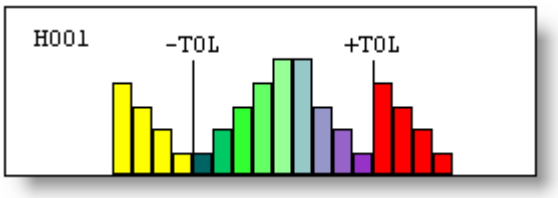
If you like, you can also do the above example without defining local variables at all. In this case, in step 3 above, your code in the **VBA Mini-Editor** would say this instead:

```
this.SetValues = 0
this.XValue = 0.1
this.YValue = -0.4
this.SetValues = 1
this.XValue = 0.25
this.YValue = 0.5
this.SetValues = 1
this.XValue = 0.75
this.YValue = -0.5
this.SetValues = 1
this.XValue = 0.85
this.YValue = 0.45
this.SetValues = 1
```

This code is similar to the code used in the PC-DMIS commands in the "Example of Using PARAM to Display Data on the Graph" topic.

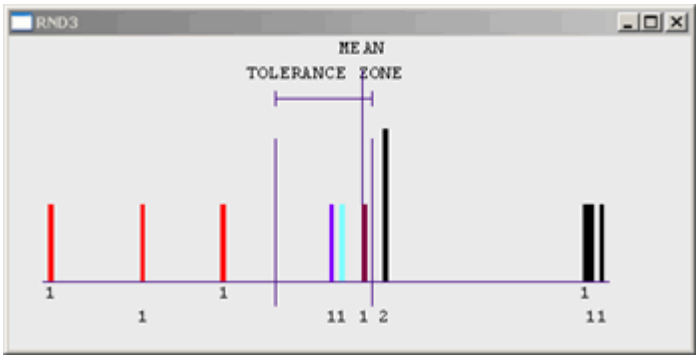
DimensionHistogram Object

The **DimensionHistogram** object displays, in a histogram bar-graph format, the number of deviations within each of the different tolerance zones.



A sample histogram.

When a dimension only has a single tolerance value, such as a form dimension, PC-DMIS displays the tolerance band as a range and not in a fixed location. PC-DMIS will not display -TOL or +TOL in the resulting histogram but will instead show the heading "TOLERANCE ZONE" at the top and include a location for the mean value like this:



A sample histogram with a single tolerance.

Inserting the DimensionHistogram Object

Use the Label Template Editor for a new or existing label template and then add and size the **DimensionHistogram** object so that it appears vertically or horizontally as desired in your label template editor's display area. Next, use the **Rule Tree Editor** dialog box for a new or existing report template to specify under what conditions (for what dimensions) PC-DMIS will call and display that label template. See the "About the Rule Tree Editor" topic in this chapter for information on how to do this.

DimensionHistogram objects are available only for dimensions that are directly calculated using the individual hits of their features, such as Location, Position, and Keyin dimensions.

Changing Histogram Colors

To see how these tolerance zones are defined, or to change the colors that represent each tolerance zone, see "Editing Dimension Colors" in the "Editing the CAD Display" chapter.

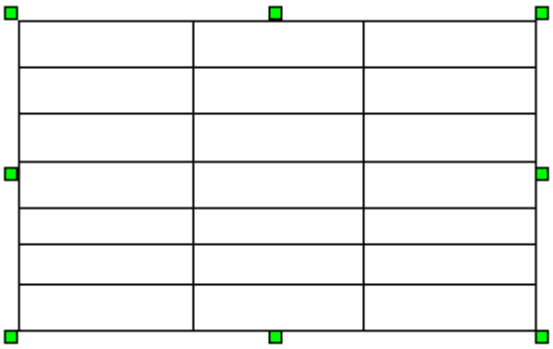
Properties

Properties specific to this object are listed below.

HatchStyle
Same as described in the "Ellipse Object" topic.

GridControlObject

The **GridControlObject** icon inserts a customizable grid of columns and rows into your report or label template.



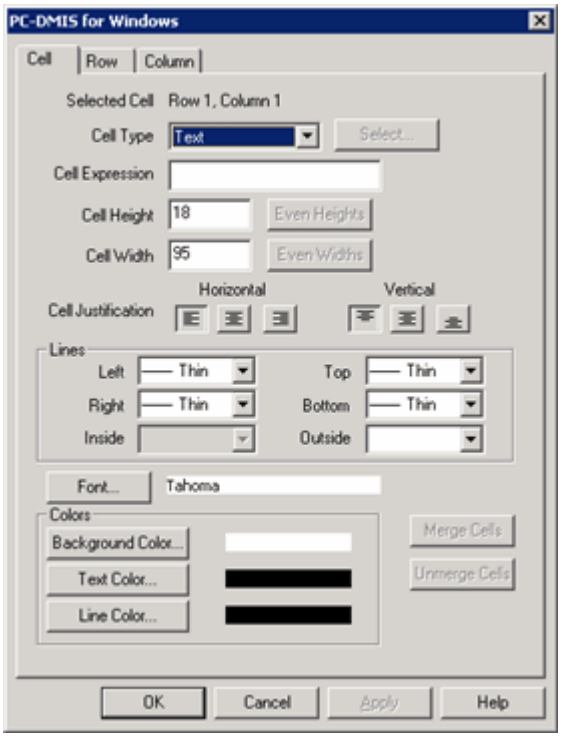
Example of a grid control object.

This unique object is more than just a standard table, though. In fact, you can program the individual cells, rows, or columns using the reporting expression language to cause PC-DMIS to pull and display specific reporting information from your measurement routine.

As with the other objects, you can easily resize and move this object to a new location. When you resize this object, be aware that PC-DMIS dynamically sizes the columns and rows to fit the new object's size.

Accessing the GridControlObject Editor

Once you insert a **GridControlObject**, select it, double-click inside any cell, and then right-click to display a tabbed dialog box. This dialog box gives you powerful formatting and editing abilities that let you program each grid cell to your individual specifications.



Cell tab of the GridControlObject's Editor

See these topics for information on using this editor:

- GridControlObject Editor - The Cell Tab
- GridControlObject Editor - The Row Tab
- GridControlObject Editor - The Column Tab

Properties

`NumColumns`
Determines the number of columns in the grid.

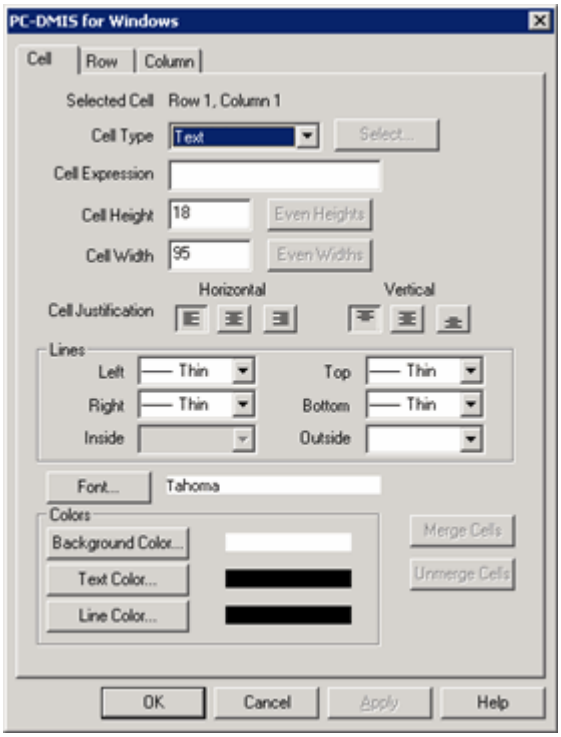
`NumRows`
Determines the number of rows in the grid.

`TableFormat`
Specifies a grid layout name to use. If this property matches the grid layout name in the **Rules Tree Editor**, then you can use the `TABLE/FORMAT` Edit window command to directly control row and column order and visibility for your label template.

`Transparent`
Determines whether or not the grid's background is transparent. If set to `Yes`, the usual white background color becomes transparent; other objects behind this object are then visible through the grid.

Increasing the `NumRows` property adds new rows to the bottom of the available rows. Decreasing them will delete the bottom-most row and work up. Similarly, increasing the `NumColumns` property adds new columns to the right of the available rows. Decreasing the number of columns will delete the right-most column and work left.

GridControlObject Editor - The Cell Tab



Cell tab of the GridControlObject's Editor

Cell Type - Lets you configure what you will insert into the specific cell. Options are:

- **Text** - This option lets you type straight text or an expression command into the **Cell Expression** box. The evaluated expression or the straight text appears in the cell.
- **Image** - This option inserts an image into the cell. You can select it using the **Select** button.
- **ActiveX** - This option inserts an ActiveX object into the cell. Generally PC-DMIS templates use these to display graphical analysis information. You can click the **Select** button to select which ActiveX control to add. For more information on ActiveX objects, see the "ActiveX Object" topic and the "Using PC-DMIS ActiveX Controls" topic.

Cell Expression - This box lets you type a reporting expression into the cell. PC-DMIS evaluates the expression when it uses the template to display the actual report data. For example, to display a feature's ID, type **=ID** into this box. For a robust listing of expressions you can include, see "About Report Expressions".

Cell Height and Width - These boxes let you define the cell's height and width in pixels. Note that if you modify a height, all cells in that row are modified. If you change the width, all cells in that column are modified. If you have varying heights and widths among multiple cells, the **Even Width** and **Even Height** buttons even out the height and width among the selected cells

Cell Justification - These buttons let you set the alignment of the text within the cell, either vertically or horizontally. If horizontal, you can align the text to the left, the center, or the right of the cell. If vertical, you can align the text to the top, the middle, or the bottom.

Horizontal Alignment Buttons

- Left alignment

- Center alignment

- Right alignment

Vertical Alignment Buttons

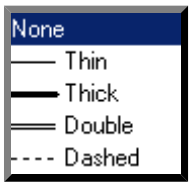
- Top alignment

- Middle alignment

- Bottom alignment

Note that when aligning text within the grid's cell, PC-DMIS uses a default internal value for cell padding. You can use the `GridCellTopMargin`, `GridCellBottomMargin`, `GridCellLeftMargin`, and `GridCellRightMargin` entries in the **Reporting** section of the PC-DMIS Settings Editor to override these default values.

Lines area - This area contains a list for each side of the cell. You can set each side of the cell (plus the outside or inside lines of multiple cells) to a particular line type. Options include: **None**, **Thin**, **Thick**, **Double**, or **Dashed**.



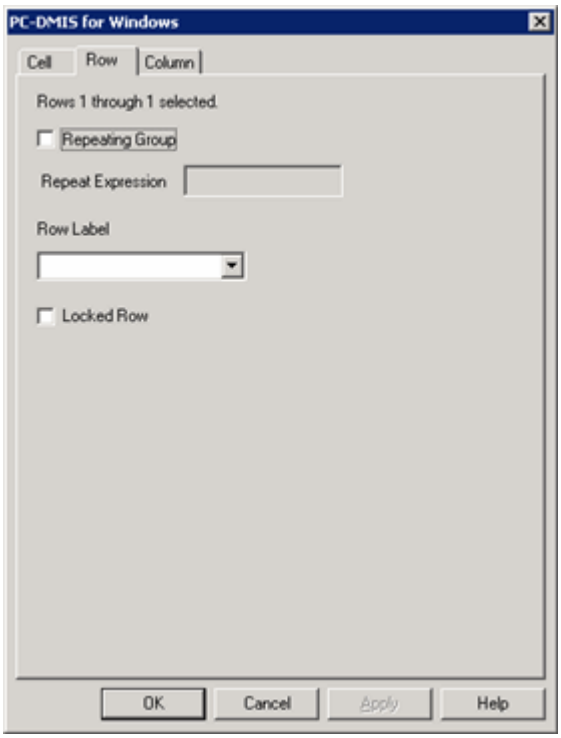
Font - This button displays a standard **Font** dialog box allowing you to set the font, size, style, effects, and color for the selected cell(s).

Colors - This area lets you set the colors for the background, text, or line of the selected cell(s). Clicking one of the buttons displays a standard **Color** dialog box from which you can select a specific standard color or create a custom color.

Merge Cells - This button merges multiple cells together into one cell.

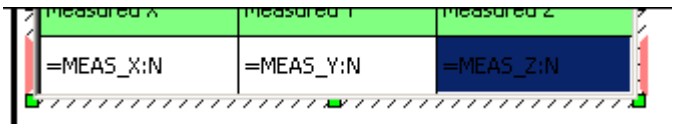
Unmerge Cells - This button unmerges previously merged cells into their original and previous unmerged state.

GridControlObject Editor - The Row Tab



Row tab of the GridControlObject's Editor

Repeating Group - This check box enables the **Repeat Expression** box and tells PC-DMIS that the selected row will continue repeating itself with data from the report until the expression in the **Repeat Expression** box is satisfied. When you select a cell in the GridControlObject, PC-DMIS indicates any repeating rows in the object by drawing small vertical orange bars on their left and right sides.



Example of the orange bars on the left and right sides of the row.

Repeat Expression - This box determines how many times PC-DMIS will repeat the row. You will usually put an expression in this box that will evaluate to a counted number of items. For example, to get the number of axes in a dimension, you would use this expression:

=COUNT(AXIS)

Then in each cell of the row, you would need to append this to existing expressions:

:N

This essentially tells PC-DMIS to repeat the row the counted number of times. So, if the **Cell Expression** box in the **Cell** tab used **=AXIS** to list a dimension's axis, and you changed it to read **=AXIS:N**, PC-DMIS would continue to repeat the row with unique dimension axis data until all axes were reported.

Row Label - The **Row Label** list lets you define a label for your row to use with the Table Format command. When you access the Table Format command's properties, you will see the row labels you've defined here. Using the Table Format command, you can re-define the order of the rows in the grid, among other things.

Locked Row - This check box locks the row so that you cannot modify it using the Table Format command.

GridControlObject Editor - The Column Tab

Column tab of the GridControlObject's Editor

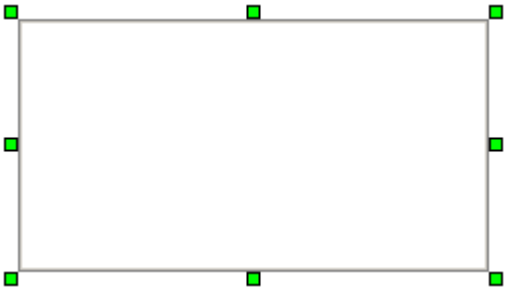
Column Label - The **Column Label** list lets you define a label for your column to use with the Table Format command. When you access the Table Format command's properties, you will see the column labels you've defined here. Using the Table Format command, you can re-define the order of the columns in the grid, among other things.

Label Object

The **Label** object icon inserts a **Label** object into your report template or custom report. Label objects are used to display report data in your report. The **Label** object behaves differently depending on the context in which it is used. If you use the **Label** object in the Report Template Editor, it functions differently than using it in the Custom Report Editor.

Using the Label Object within the Report Template Editor

Click and drag on the Report Template Editor to add this object. When you release the mouse button, the inserted **Label** object looks something like this:



A sample label object.

A **Label** object acts just like a **TextReportObject** or **CadReportObject**; it doesn't contain any real data and merely acts as a placeholder for a label template. The label template actually controls what data gets pulled from a measurement routine execution. If you don't define what the label object will display, then it doesn't appear in the report.

Assign a Label Template

To give a **Label** object something to display, you need to assign a label template for it to use. Follow this procedure:

1. Right-click on the label object. The **Properties** dialog box appears.
2. Click **Rules** from the **Rules Tree** property. The **Rules Tree** dialog box appears.
3. Using the **Rules Tree** dialog box, select an item from one of the expandable lists.
4. Click **Add** to add a rule that uses that item. The **Edit Rule** dialog box appears.
5. Select **Use Label Template for Report** option.
6. Click the browse button ..., and select a label template file (.lbl filename extension).
7. Click **OK** to close the **Edit Rule** dialog box.
8. Click **OK** to close the **Rules Tree** dialog box. The label object you inserted will now display a picture of the label template defined in the first rule in the **Rule Tree** dialog box.
9. Save and test your report template. PC-DMIS will display the selected label when the condition you specified is met.

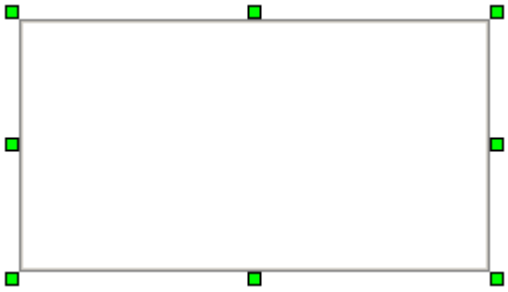
See "About the Rule Tree Editor" for information about creating rules.

Using the Label Object within the Custom Report Editor

One crucial difference between the Report Template Editor and the Custom Report Editor is that in the Custom Report Editor, **Label** objects *do not use their own Rule Tree Editor.* Instead, the **Page** object contains a Rule Tree Editor that defines what label template is used when the different commands and data are inserted into the Custom Report Editor.

Most of the time you will likely simply create your custom report by dragging items from the Edit window's Summary Mode into the Custom Report Editor. In this case, PC-DMIS uses the label templates defined in the **Page** object's Rule Tree Editor for the command you are inserting.

If you do insert a **Label** object into your custom report, PC-DMIS inserts a blank **Label** object, similar to how it would appear in the Report Template Editor:



A sample label object.

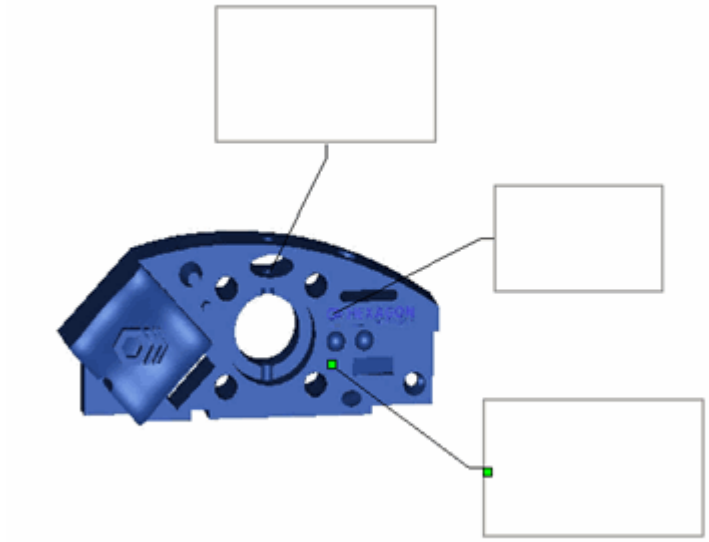
In this state, the object is merely an empty placeholder, and it won't display any data until you drag and drop a command from the Edit window onto the object. PC-DMIS will then display the label template defined in the **Page** object's Rule Tree Editor.

See "Creating Custom Reports" for more information.

Leader Line Object

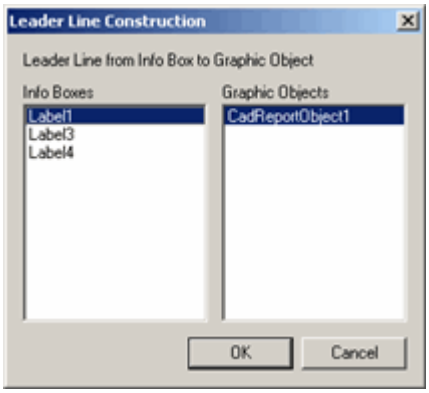
The **LeaderLine** object lets you draw leader line (a connecting line) between a **Label** object and a **CadReportObject**. To do this, simply select the object, then click and drag it from one object to the other.

If done properly, the leader line automatically snaps between the two objects so that if you later change their locations, the leader line changes size and moves accordingly. For example, this figure shows a **CadReportObject** object with three **Label** objects around it all with connecting **LeaderLine** objects.



Three Label objects with LeaderLine objects drawn to a CadReportObject

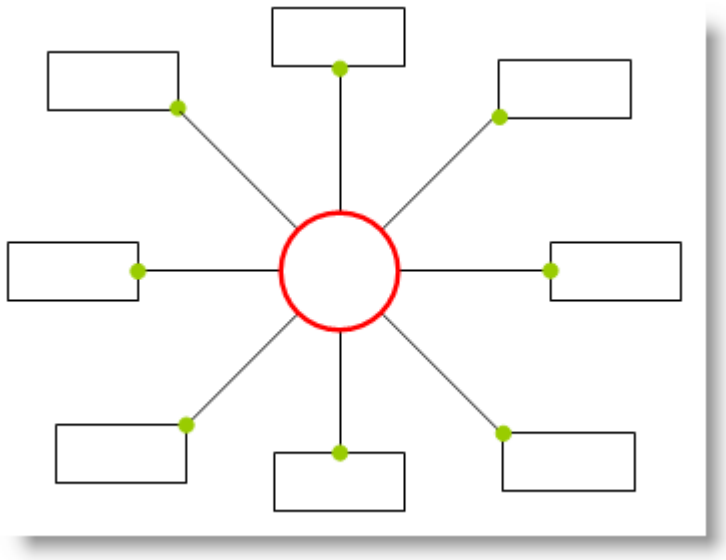
If the leader line can't detect the two objects, a **Leader Line Construction** dialog box appears, allowing you to select the two objects for the leader line.



Leader Line Construction Dialog Box

If you delete the Label or a **CadReportObject** object, PC-DMIS also deletes the leader line object automatically.

In the actual Report window, depending on the location of the label to the feature, the leader line attaches to either one of the edges or corners of the label. Consider this graphical example. It illustrates that where you position the labels (rectangles) in relationship to the feature (red circle) changes where the leader line connects (green dot) on the label.



Example showing the label and leader line connection positions.

The following properties are available:

LeaderLineVisibility
This sets the visibility state of the selected leader line. TRUE shows the leader line. FALSE hides it.

Object1
Defines the first of the two objects between which the leader line will be drawn.

Object2
Defines the second of the two objects between which the leader line will be drawn.

Line Object

The **Line** object allows you to insert a standard line into your form or template. You can then right-click on the inserted line and modify its properties.

Common properties used on a line object are:

PenWidth
This sets the width of the line in pixels.

PenStyle
This changes the line style to `Solid`, `Dash`, `Dot`, `Dash-dot`, and `Dash-dot-dot`.

Arrowhead
This determines if the line should contain an arrowhead and the direction the arrowhead points. You can choose one of these formats:

`----(none)`
`<---`
`--->`
`<--->(both)`

ArrowheadHeight
This determines the height of the arrowhead in pixels.

ListBox Object

The **ListBox** object inserts an open list inside the form.

Along with the ability to resize the object, change colors, and change other attributes, you can further customize this object by using these properties:

BorderDrawn
Setting this to `YES` draws a border around the control.

HorizontalScroll
Selecting `Always` places a horizontal scroll bar on the list box even if the text for a list item isn't long enough to cause it to scroll.

Selecting `None` removes the scroll bar.

ListID
This is the ID of the list. You can set or get this by using the control. It should be in this format:

`item 1\r\nitem 2\r\nitem 3` and so on.

RemoveSelection
Selecting `YES` removes the selection from the list box (`ListID`) when recorded.

Sort
Selecting `YES` sorts the list items alphabetically.

UseColors
This determines whether or not the control should use the colors specified in the `BackColor` and `TextColor` properties.

UseTabstops
This determines whether or not the list box should use tab stops. If set to `YES`, pressing Tab in run mode will eventually make this list box the active control.

VerticalScroll
Selecting `Always` places a vertical scroll bar on the list box even if there aren't enough list items to cause it to scroll.

Selecting `None` removes the scroll bar.

WantKeyInput
This determines whether or not the list box should accept keyboard input.

MultiEditBox Object

The **MultiEditBox** object inserts an edit box that supports more than one line of text. This has all the properties of the **EditBox** object (see "EditBox Object"), plus the following:

HorizontalScroll
Selecting `Always` places a horizontal scroll bar on the control even if the text isn't long enough to cause it to scroll.

Selecting `None` removes the scroll bar.

VerticalScroll
Selecting `Always` places a vertical scroll bar on the control even if the text isn't long enough to cause it to scroll.

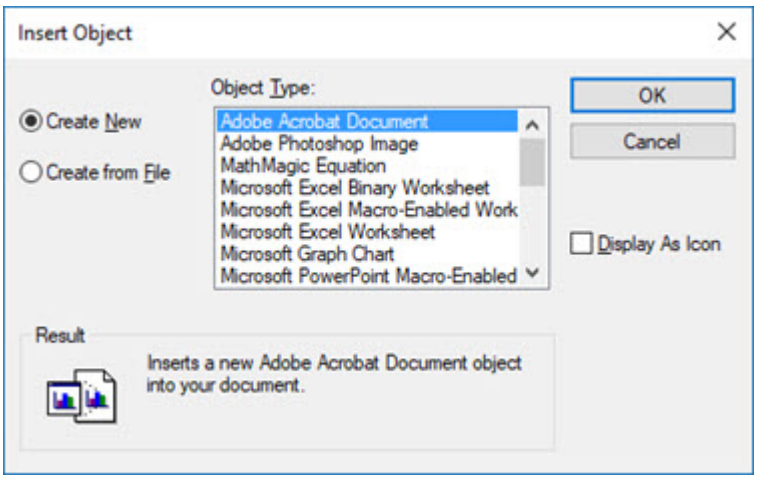
Selecting `Automatic` places a vertical scroll bar only when the amount of text exceeds the vertical height of the box.

Selecting `None` removes the scroll bar.

OLE Object

The **OLE** object embeds or links an object from another application into your template or form. For example, using this object you can insert an editable Microsoft Word file into your template or form to provide special instructions to the operator.

Once you insert an **OLE** object, a dialog box titled **Insert Object** appears.



Insert Object Dialog Box

This dialog box allows you to select the type of OLE object from a list of objects that are currently available on your computer. The OLE objects in the list are unique to your individual computer, depending on what is installed, etc.

If you select **Cancel**, nothing is added to the form or template, and the dialog closes.

DoVerbs
Setting this to `Yes` allows the object to be edited in run mode as well as in edit mode. If set to `No`, the object is read-only in run mode.

OLEProperties
This displays a properties dialog box that lets you choose general properties available to all OLE objects, such as how the object is displayed (icon or not), its size, and so forth.

Accessing an OLE Object's Automation Methods and Properties from Basic

You can access an OLE (Object Linking and Embedding) object's methods and properties from Basic to automate tasks related to the object. For example, you could change a bitmap image, populate an embedded Microsoft Excel spreadsheet object with data, or set the text in an embedded Microsoft Word document.

Suppose you add a Bitmap OLE control, and you name it "BITMAP" by setting its `(Object Name)` property to `BITMAP`, you can add program code to change the bitmap being displayed. To do this, you can access the OLE automation methods and properties for the object named `BITMAP`, with the following code:

```
AttachOLE("BITMAP_X")
```

You can then use the object variable BITMAP_X to set a new property value or perform an automation method call:

```
BITMAP_X.{property} = {value}
```

or

```
BITMAP_X.{automation method call}.
```

Consult your OLE's documentation for the particular OLE object you are using to know what methods and properties are available.

Providing Operator Instructions Using OLE Objects with Forms

You can use existing tools in conjunction with PC-DMIS to provide you with expanded functionality. For example, suppose you want to provide an operator with detailed instructions regarding part setup or measurement. One way to do this is to embed into a custom form, using the OLE object, an external file containing these instructions. Below are some examples of how to embed instructional files into forms using some common Microsoft Office tools.

These examples demonstrate how to use a Microsoft Word file and Microsoft PowerPoint file as OLE objects inside of a form to provide instructions to an operator during measurement routine execution. These examples provide you with a taste of what you can do with OLE objects inside of your report or form. How you put this knowledge to use for your own situation is up to you.

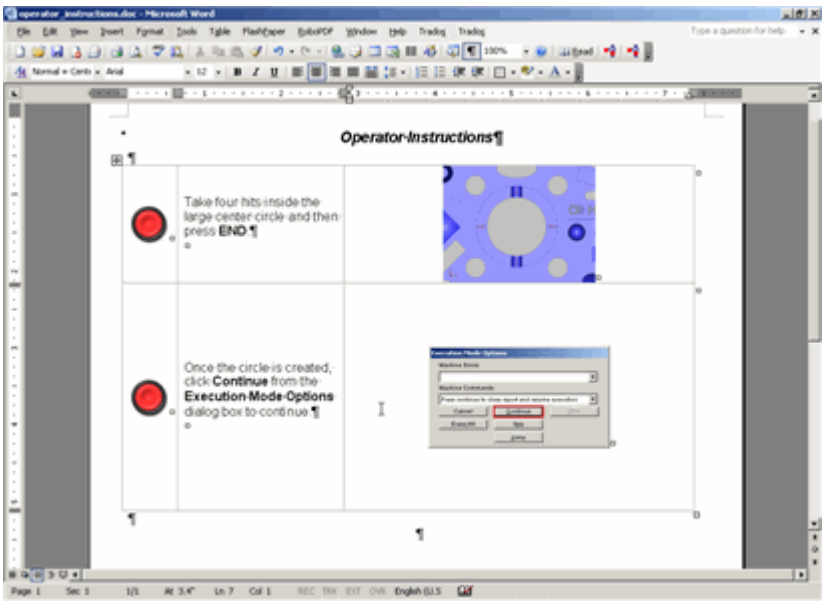
- **Providing Instructions Using a Microsoft Word OLE Object**

  One way to provide instructions to those who run your measurement routines is to use something like Microsoft Word.

  This topic and the steps below assume you have Microsoft Word installed on your computer.

  Step 1 - Create and Save the Instructions as a Word Document

  1. Create your part setup instructions inside of a Word document.
  2. Save the .doc file to a directory on your disk. Feel free to include pictures, tables, and other advanced formatting inside your .doc file. The sample doc file used for this topic looks like this:



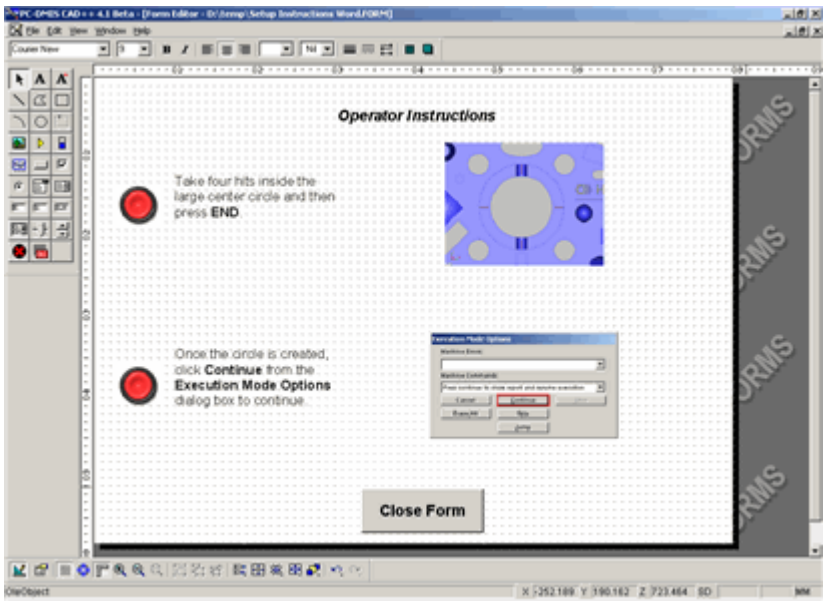
Step 2 - Create the Form

1. Select **File | Reporting | New | Form Report** to open the Form Editor.
2. If the Edit window is open, select **View | Edit Window** to close it giving you more screen space.
3. Maximize the Form Editor.
4. Size the form so that it properly houses the content of your file.
5. Add any other objects as desired. In the example form used for this topic, a **Close Form** button was added with the following properties:
   - Text = "Close Form"
   - ButtonType = OK

   When you click the button, the form closes.

Step 3 - Insert the OLE Object

1. From the **Object Bar**, click the **OLE Object** icon. Drag the object so that it fits the entire form except for a half inch margin on all sides. When you release the mouse button, the **Insert Object** dialog box opens.
2. Select the **Create from File** option.
3. In the **File** box, either type the complete pathway to the .doc file, or use the **Browse** button to locate it.
4. Select the **Link** check box. This maintains a link between the OLE object on your form and your .doc file, so that if you later update your instructions, PC-DMIS always uses the latest information.
5. Click **OK**. The document is embedded into your form like this:



6. Save the form and give it a name of your choice.

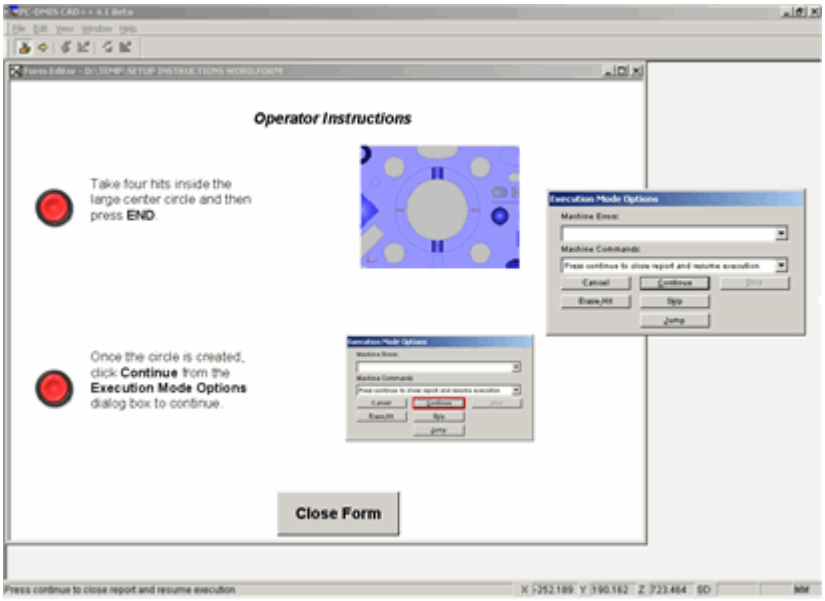
> The **Insert Object** dialog box is the same dialog box that appears if you choose to insert an external object into the Edit window from the **Insert | Report Command | External Object** menu option. If you need more information on this dialog box, consult the "Inserting External Objects" topic from the "Inserting Report Commands" section.

Step 4 - Insert and Test the Form

1. Select **View | Edit Window** to make the Edit window visible.
2. Select **Insert | Report Command | Form**. Use the dialog box to select your saved form.
3. PC-DMIS inserts a `FORM/FILENAME` command block into your measurement routine with a pathway to the form file:

```
CS1=FORM/FILENAME = <Pathway to Form File>
   PARAM/=
   ENDFORM/
```

4. Mark the command block for execution.
5. Execute the measurement routine. When PC-DMIS reaches the `FORM/FILENAME` command block, it displays the form:



6. Click **Continue** from the **Execute Mode Options** dialog box to close the form and resume execution.

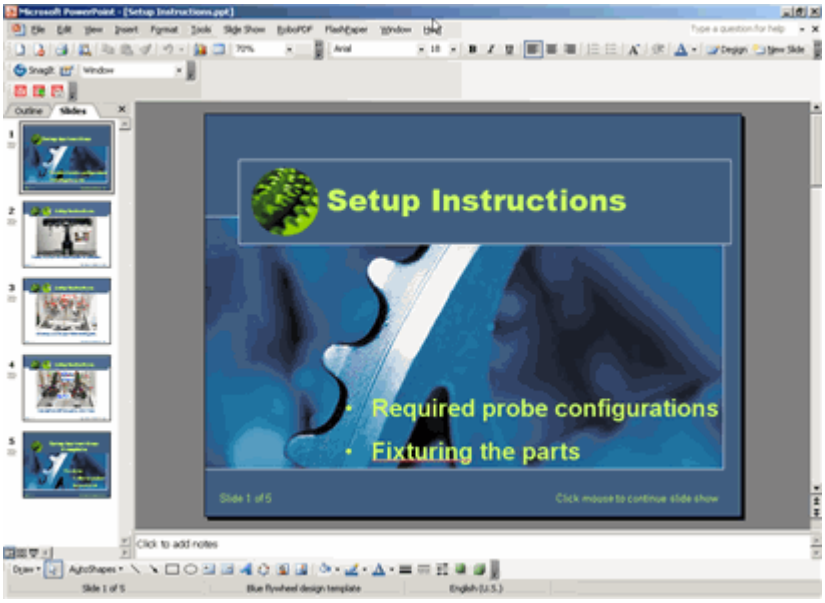
- **Providing Instructions Using a Microsoft PowerPoint OLE Object**

  This topic demonstrates how to use a Microsoft PowerPoint file (.ppt) as an OLE object inside of a form to provide detailed slide-show type instructions to an operator.

  > This topic and the steps below assume you have Microsoft PowerPoint installed on your computer.

Step 1 - Create and Save the Instructions as a PowerPoint

1. Create your part setup instructions inside of a PowerPoint file (.ppt).



2. Save the .ppt file to a directory on your disk.

Step 2 - Create the Form

1. Select **File | Reporting | New | Form Report** to open the Form Editor.
2. If the Edit window is open, select **View | Edit Window** to close it to give you more screen space.
3. Maximize the Form Editor.
4. Size the form so that it properly houses the content of your file.
5. Add any other objects as desired. In the example form used for this topic, a **Close the Form** button was added with the following properties:
    - Text = "Close the Form"
    - ButtonType = OK

    When you click the button, the form closes.

Step 3 - Insert the OLE Object

1. From the **Object Bar**, click the **OLE Object** icon. Drag the object so that it fits the entire form, except for a half inch margin on all sides. When you release the mouse button, the **Insert Object** dialog box appears.
2. Select the **Create from File** option.
3. In the **File** box, either type the complete pathway to the .ppt file, or use the **Browse** button to locate it.
4. Select the **Link** check box. This maintains a link between the OLE object on your form and your .ppt file, so that if you later update your instructions, PC-DMIS always uses the latest information.
5. Click **OK**. The PowerPoint file is embedded into your form.

6. Save the form and give it a name of your choice.

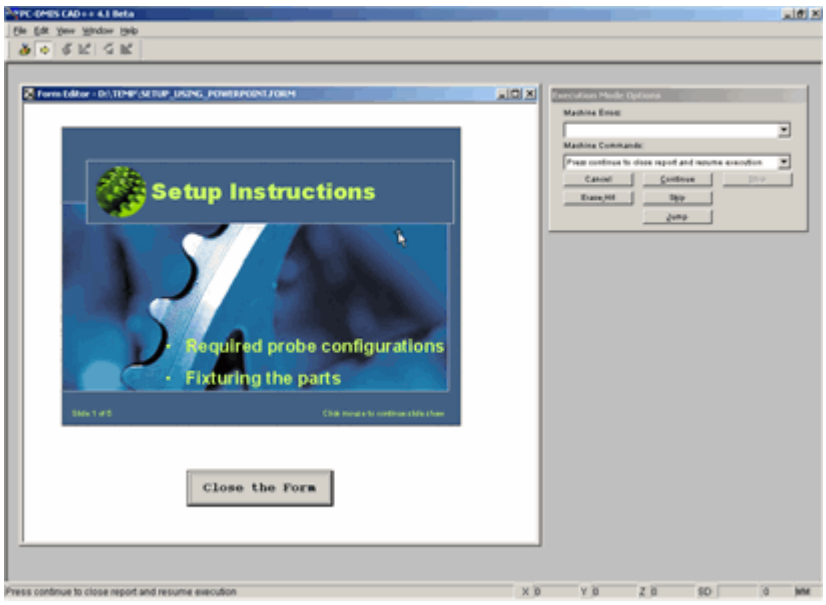
The **Insert Object** dialog box is the same dialog box that appears if you choose to insert an external object into the Edit window from the **Insert | Report Command | External Object** menu option. If you need more information on this dialog box, consult the "Inserting External Objects" topic from the "Inserting Report Commands" section.

Step 4 - Insert and Test the Form

1. Select **View | Edit Window** to make the Edit window visible.
2. Select **Insert | Report Command | Form** to show the **Insert Form** dialog box.
3. Use the **Insert Form** dialog box to select your saved form.
4. Click **Open**.
5. PC-DMIS inserts a `FORM/FILENAME` command block into your measurement routine with a pathway to the form file.

```
CS1=FORM/FILENAME= <Pathway to Form File>
   PARAM/=
   ENDFORM/
```

6. Mark the command block for execution.
7. Execute the measurement routine. When PC-DMIS reaches the `FORM/FILENAME` command block, it displays the form and the embedded PowerPoint presentation.



8. Double-click on the embedded presentation. The Microsoft PowerPoint software launches and displays the presentation. When it finishes, PowerPoint also closes.

9. Click **Continue** from the **Execute Mode Options** dialog box to close the form and resume execution.

Pointer Object

The **Pointer** object inserts a dynamic, movable, colored pointer into the form or template. The color defaults to yellow.

Along with the ability to resize the object, change colors, and change other attributes, you can cause this object to move by using these properties in conjunction with Visual BASIC code:

MaximumEq
Maximum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

-$, $, CONST

MinimumEq
Minimum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

-$, $, CONST

ValueEq
Value equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

-$, $, CONST

If you change the direction of the pointer, the movement of the arrow will also change to the direction specified.

PointInfo Object

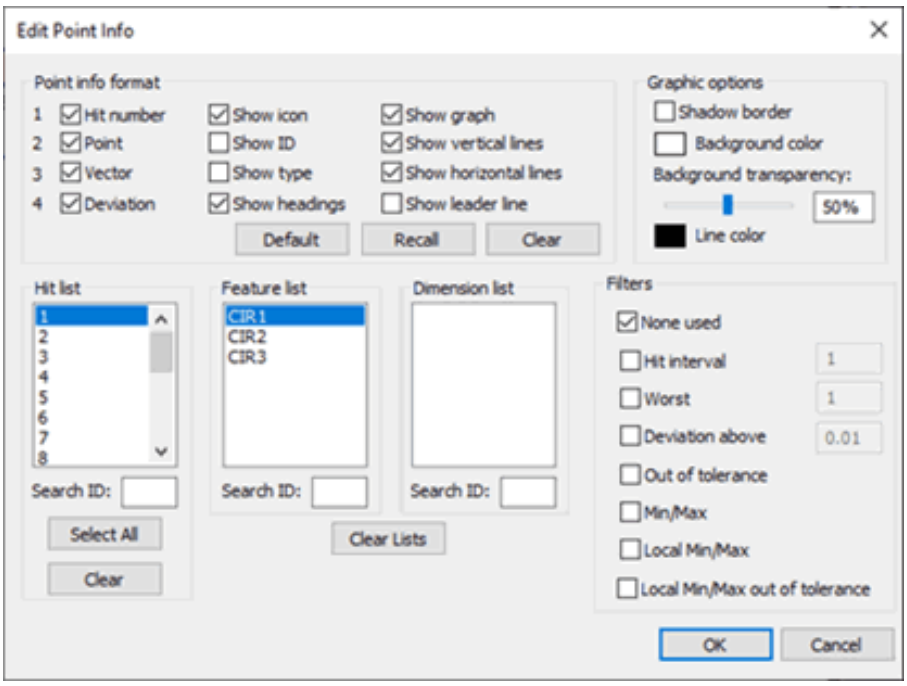
The **PointInfo** object acts as a container to hold individual point information in your report.

Using the PointInfo Object in a Label Template

Using the **Object Bar** for the Label Template editor, add and size the **PointInfo** object so that it appears as desired in your Label Template editor's display area. Save the label template. Next, use the **Rule Tree Editor** dialog box for a new or existing report template to specify under what conditions (for what dimensions) PC-DMIS will call and display that label template. See "About the Rule Tree Editor" in this chapter for information on how to do this.

Using the PointInfo Object in a Custom Report

Using the **Object Bar** for the Custom Report editor, drag or drop the **PointInfo** object into the Custom Report. PC-DMIS displays the **Point Info** dialog box allowing you to select the point to be associated with the object.



Select the feature or dimension and then a point from the dialog box, set whatever options you want, and click **OK**. PC-DMIS creates point info commands for the selected features or dimensions and closes the dialog box. PC-DMIS displays a table on your report that contains feature or dimension point information.

See "Inserting Point Info Boxes" in the "Inserting Report Commands" chapter for complete information on how to use this dialog box to display dimension information.

In a custom report, the **Analysis** object can display its own Point Info boxes as well. See "Analysis Object" for more information.

Properties specific to this object are listed below. Many of these you can set by clicking on the (Settings Dialog) property:

`DimFeatID`
Defines the feature or dimension ID containing the hits you want to display.

`HitNumber`
Defines what hit number for the information you want displayed in the point info box.

`OrderDeviation`
Defines the order of the deviation row with a value between `1-4`. A value of `1` puts the info on the top row. A value of `4` puts the info on the bottom row.

`OrderHitNum`
Same as above, except it defines the order of the hit number.

`OrderPoint`
Same as above, except it defines the order of the point row.

`OrderVector`
Same as above, except it defines the order of the vector row.

`ShowGraph`
A value of `1` displays the dimension percentage graph. A value of `0` hides the graph.

`ShowHeadings`
A value of `1` displays the heading row. A value of `0` hides the heading row.

`ShowID`
A value of `1` displays the ID of a particular feature or dimension row. A value of `0` hides the ID.

`ShowPointInfo`
A value of `1` displays the point info row. A value of `0` hides the ID. Disabled at this moment.

`ShowType`
A value of `1` displays the type of feature or dimension displayed. A value of `0` hides the type.

Polyline Object

The **Polyline** object allows you to link lines together. When you click and create the first line, a second line is automatically started at the end point of the first line. Polyline objects contain the same properties as a standard line object.

Radiobutton Object

The **Radiobutton** object inserts option buttons into the form. Option buttons are mutually exclusive. You can only have one option button selected inside your report at a time. The `ListItems` property allows you to define a set of option buttons.

Along with the ability to resize the object, change colors, and change other attributes, you can further customize this object by using these properties:

AlignTextLeft
If set to `YES`, this moves the text to the left of the option button like this:

Text Here ( )

If set to `NO`, this moves the text to the right of the option button like this:

( ) Text Here

Bitmap
This allows you to specify a bitmap to use for a selected option button.

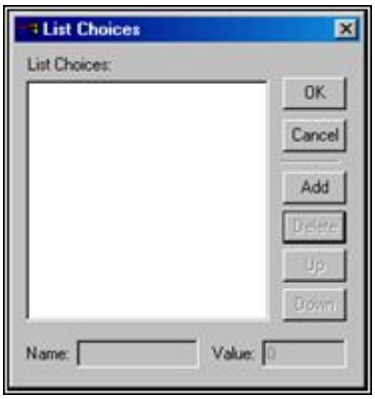
The specified bitmap must be the same shape as a standard check box.

BitmapOffState
This allows you to specify a bitmap to use for a deselected option

The specified bitmap must be the same shape as a standard option button.

ListItems
This defines a list of option buttons and its associated values (see the description on the **List Choices** dialog box below).

The `ListItems` property displays the **List Choices** dialog box.



List Choices Dialog Box

This dialog box allows you to add, rename, and delete option buttons and assign numerical values. Generally, when adding a new option, the value for that option increments automatically, so only one button is selected at a time. However, you can change the numerical values to allow the selection of groups of options with one mouse click when in run mode.

For example, suppose you have five option buttons that you've named Options A through E, and you change their assigned values to the following:

Option buttons	Value
Option A	0
Option B	0
Option C	1
Option D	2
Option E	2

When you enter run mode and select **Option A**, both **Option A** and **Option B** will be selected. This is because they're assigned the same value. Similarly, if you select **Option E**, both **Option D** and **Option E** will be selected. Only **Option C** has a unique value, so when you select it, only it is selected.

Select Object

The **Select** object isn't a traditional object that you insert into the template or form editor. Rather, it allows you to select an object already inserted into your editor. To select an object, click on the object in the report. To let you know when you select an object, PC-DMIS surrounds it with a border and small green squares. Once you select an object, you can move, resize, or edit the object.

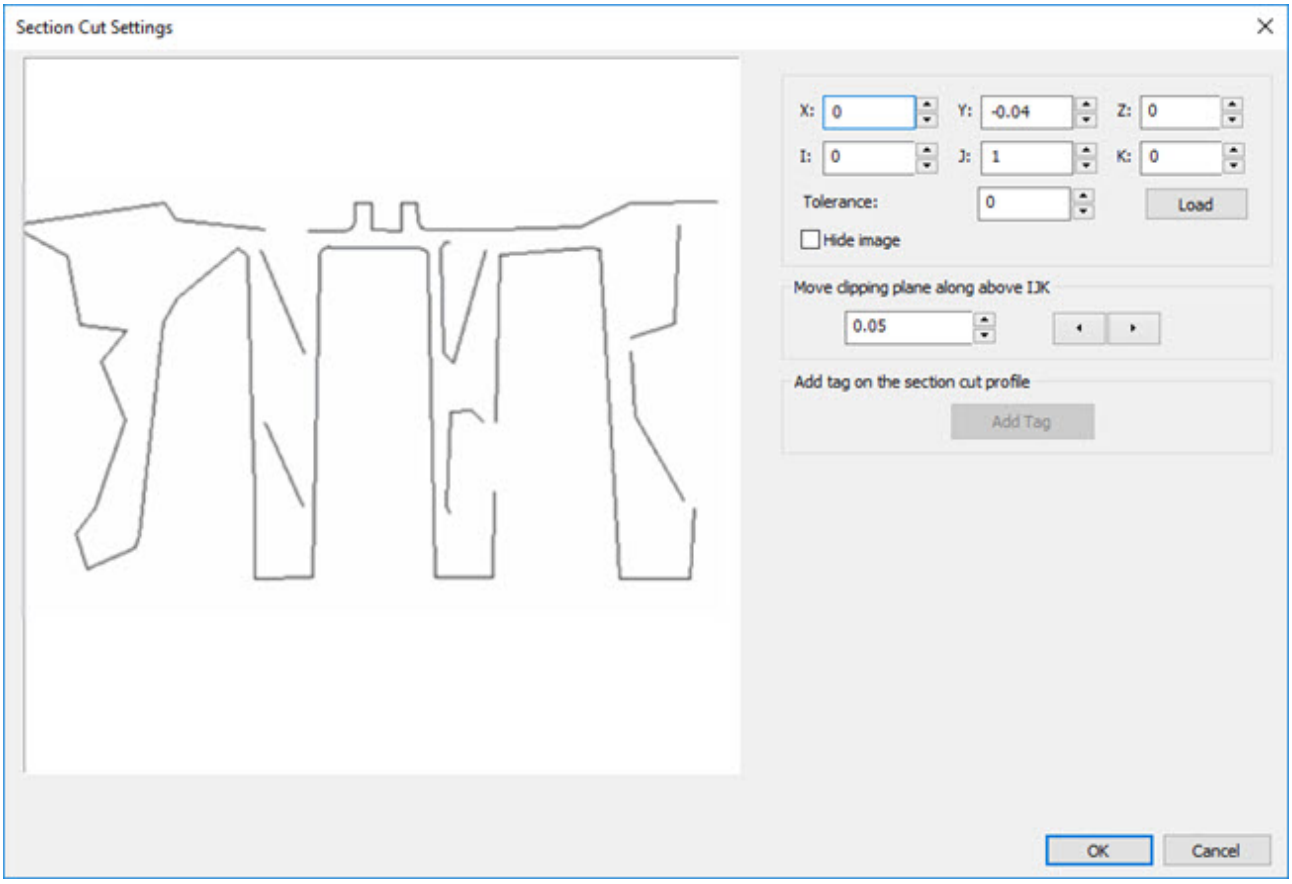
SectionCutObject

The **SectionCutObject** lets you define a section cut (cutaway view) of the CAD model. You can insert a **SectionCutObject** into the Report Template Editor as with other objects, or you can insert it into a final report page on the Report window directly. To insert this object, position the mouse pointer, and click and drag a box. When you release the button, the object appears, and—at least in the Report Template Editor—it displays "No Image!" until you move the clipping plane so that it intersects with the CAD model. You can move the clipping plane through the object's properties.

Changing the Object's Properties

If you insert the **SectionCutObject** into the Report Template Editor, right-click on it to access the object's **Properties** dialog box. You can modify the properties directly in the **Properties** dialog box or select the **(Settings Dialog)** property, and click the **...** button to use the **Section Cut Settings** dialog box to more easily modify its properties.

If you insert this object on-the-fly into a report page of the Report window, the same **Section Cut Settings** dialog box appears.



Section Cut Settings Dialog Box

You can use this dialog box to define a cutaway image of your part model that you want to show in your report.

Defining a Cutaway Image Using the Section Cut Settings Dialog Box

1. Fill out the **X**, **Y**, and **Z** boxes to define a point on the CAD model where the clipping plane intersects with the model.
2. Fill out the **I**, **J**, and **K** boxes to define the direction the plane faces.
3. Modify the **Tolerance** value as needed.
4. Click the **Load** button to see a live preview of the cutaway image in the **Section Cut Settings** dialog box.

5. Use the large left and right arrow icons to fine-tune the placement of the plane and get the exact cutaway view that you need. The preview in the dialog box updates to match the placement of the plane.

6. Reposition the image as desired. Pan the image by right-click dragging. Zoom in or out on the cutaway image by right-clicking above or below the imaginary horizontal line that splits the image view. 2D rotate the part image by holding the Ctrl key and right-click dragging.

7. Finally, place callouts onto the dialog box image by using the **Add Tag** button.

8. Click **OK** to accept your changes.

Section Cut Settings Dialog Box Items

X Y Z boxes
These boxes define a point on the CAD model where the clipping plane cuts the model.

I J K boxes
These boxes define the IJK direction vector of the clipping plane.

Tolerance box
The tolerance parameter is used to:

- Check if the polyline is closed. The start and end point distance is less than the tolerance value.
- Reduce the polyline.
- Merge polylines.

Load button
This button previews the section cut line on the CAD image as well as the section cut profile.

Hide Image check box
This check box hides the section cut profile image in the Preview window. If you click the **OK** button, PC-DMIS also hides the section cut profile image in the Report Template Editor or on the Report window's page.

Move clipping plane along above IJK area
This area contains an edit box and arrow buttons.

The edit box defines the distance that the clipping plane moves when you click the left arrow or right arrow button.

The arrow buttons move the clipping plane along the IJK vector by the specified distance with each click.

- If you have 1,0,0 then the plane moves along the X axis.

- If you have 0,1,0 then the plane moves along the Y axis.
- If you have 0,0,1 then the plane moves along the Z axis.

Add Tag button
This button allows you to place callouts on the cutaway image of the part model.

Clicking this button changes your cursor to a cross-hair icon. You can click and drag a leader line on the cutaway image in the dialog box. When you release the mouse button, a small text box appears, which allows you to type the callout text. The number of characters this text box can display is only limited by the size of the text box. Increasing the size of the box allows it to hold and display more characters.

You should place tags on the image only when the cutaway image has been completely finalized. If not, the tag is deleted with future modifications to the image.

OK and **Cancel** buttons
Clicking **OK** applies your changes to the section cut profile image and displays the image in the Report Template Editor or Report window. If your report contains a CadReportObject, the section cut line appears on the CAD image.

Clicking **Cancel** closes the dialog box without applying your changes. However, if you've added the object on-the-fly into your Report window, the SectionCutObject still exists. If desired, you can remove it by right-clicking on the object and selecting **Remove Object**.

Object Properties

Bottom
Common Properties

Enable
Common Properties

EventReportData
About Events and Visual Basic Code

Font
Common Properties

Hide Image
See description in table describing dialog box items above.

Left
Common Properties

Plane Anchor X
See description in table describing dialog box items above.

Plane Anchor Y
See description in table describing dialog box items above.

Plane Anchor Z
See description in table describing dialog box items above.

Plane Vector I
See description in table describing dialog box items above.

Plane Vector J
See description in table describing dialog box items above.

Plane Vector K
See description in table describing dialog box items above.

Right
Common Properties

(Settings Dialog)
This property displays the **Section Cut Settings** dialog box.

Tolerance
See description in table describing dialog box items above.

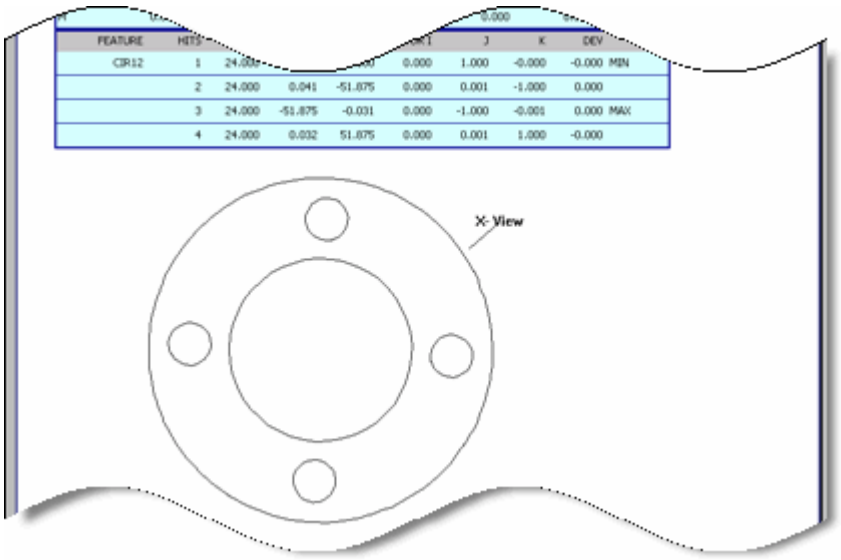
Top
Common Properties

Visible
Common Properties

Defining On-The-Fly Section Cut Profiles

Perhaps the easiest way to use section cut profiles is to define them using an on-the-fly creation method within the final report in the Report window itself. To do this:

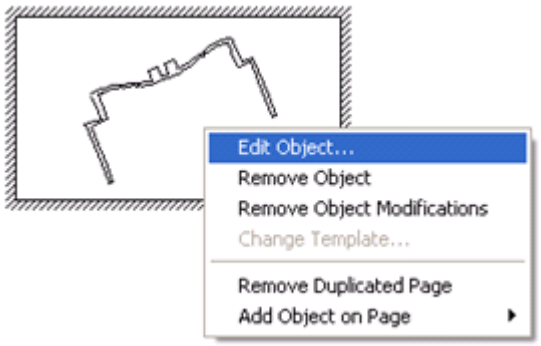
1. Right-click on the Report window. A shortcut menu appears.
2. Select the **Add Object on Page | Section Cut Profile** menu item.
3. Click and drag a box directly on the Report window with the size of the cutaway image you want. When you release the mouse button, the **Section Cut Settings** dialog box appears.
4. Use the dialog box to define the cut profile as described above. PC-DMIS inserts the section cut profile into the Report window.



Example of a section cut profile.

Modifying Section Cut Profiles in the Report Window

Right-click on the object and select **Edit Object** from the resulting shortcut menu.



Slider Object

The **Slider** object inserts a slider into your template. With a slider in your report, you can allow users to set a value from a range of continuous possible values.

Along with the ability to resize the object, and change other attributes, you can further customize this object by using these properties:

AutoTicks
Setting this to `YES` displays ticks on the slider. A tick is a short vertical/horizontal line that is placed at intervals (set by the `TickFrequency` property) along the range of the slider.

DisplayID
This determines whether or not the DisplayID gets updated when the slider gets adjusted.

HelpHotButton
Selecting `YES` shows a question mark help button next to the slider.

Clicking this help button in run mode launches the PC-DMIS Online Help.

MaximumEq
Maximum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

MinimumEq
Minimum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

Orientation
This sets the orientation of the slider.

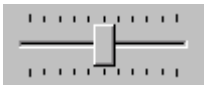
Selecting `Horizontal` sets the slider to move horizontally, side to side.

Selecting `Vertical` sets the slider to move vertically, up and down.

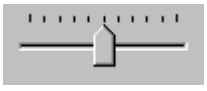
TickFrequency
This value determines the frequency that ticks are displayed. You can set a value between 2 and 200 ticks equally spaced along the range of the slider.

TickStyle
This allows you to set how ticks are displayed on a slider.

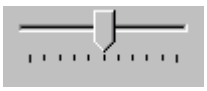
`0 - Both:` sets the ticks to above and below the slider (if a horizontal slider) or to the right and left sides (if a vertical slider).



`1 - Top/Left:` sets the ticks above the slider (if a horizontal slider) or to the left (if a vertical slider).



`2 - Bottom/Right:` sets the ticks below the slider (if a horizontal slider) or to the right (if a vertical slider).



ValueID
This is the ID of the value. You can set or get this by using the control.

Snipping Object

The **Snipping** object on the **Object Bar** puts PC-DMIS in a snipping mode. This object works with only the Custom Report Editor. In snipping mode, you can capture a section of your screen and insert it as an image into the Custom Report Editor.

To use the **Snipping** object and insert an image:

1. Click the **Snipping** object to enter snipping mode. The arrow of your pointer changes to a cross hair, and a semi-transparent white overlay covers the entire screen.
2. With your pointer, click and drag a box around the part of the screen you want in your report.
3. Release the mouse button to capture that part of the screen as an image. PC-DMIS inserts the image as an object with the name of Snipping1 into the Custom Report Editor. Additional captures are named Snipping2, Snipping 3, and so on.

The properties for this object are described in the "Common Properties" topic.

Spinner Object

The **Spinner** object inserts a spinner control into your template. You can set this control to work with an edit box, allowing you to select a numerical value by cycling through a range of numbers. The range of numbers is set using the `MinimumEQ` and `MaximumEQ` properties.

Along with the ability to resize the object vertically, and change other attributes, you can further customize this object by using these properties:

DecimalBase
This sets the base of the spinner to either decimal based (select `YES`) or hexadecimal based (select `NO`).

EditBuddy
Selecting `YES` associates the spinner to the edit control immediately preceding the

spinner in the tab order. You can view the tab order by selecting an object in edit mode and pressing Shift + Enter.

IncrAccelerator
This value determines the amount to increment or decrement the spinner each time you click an arrow. The default is `1`.

MaximumEq
Maximum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

MinimumEq
Minimum equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript).

```
-$, $, CONST
```

Orientation
This sets the orientation of the spinner control.

Selecting `Horizontal` rotates the orientation of the spinner arrows to point horizontally.

Selecting `Vertical` rotates the orientation of the spinner arrows to point vertically.

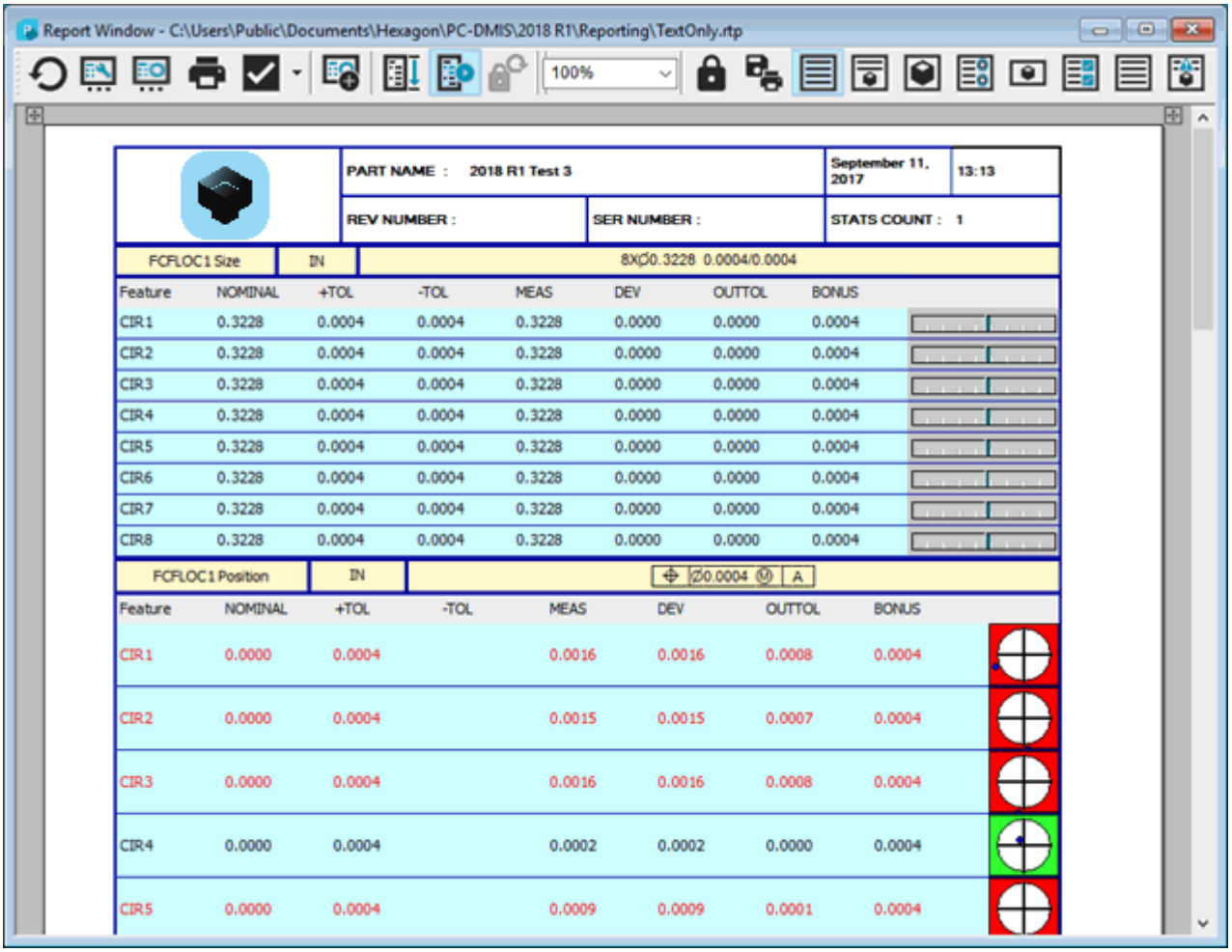
WrapAround
If incrementing or decrementing the list of values and you come to the minimum or maximum value, you can set this property to `YES`, and your report will wrap to the beginning or end of the list.

TextReportObject

The **TextReportObject** object allows you to insert a container for your textual report data into your report template. When you insert this object, PC-DMIS initially displays a box filled with some sample report data, which is useful for design purposes. Once your template gets applied to a report, your actual report data is used.

```
PART NAME  : PL54A.1
REV NUMBER :
SER NUMBER :
STATS COUNT : 1

Active alignment changed to ALIGN1

PLN1=PLANE MEASURED FROM 4 HITS
CYL1=CYLINDER MEASURED FROM 8 HITS
PLN2=PLANE MEASURED FROM 4 HITS
Active alignment changed to ALIGN2

PLN3=PLANE MEASURED FROM 4 HITS
DIM PLANEA= FLATNESS OF PLANE PLN3  UNITS=MM
AX   NOMINAL     +TOL       -TOL       MEAS        MAX         MIN
DEV      OUTTOL
M       0.000       0.050       0.000       0.007       0.004      -0.004
0.007       0.000 -#--------
```

A TextReportObject with Sample Report Data



A sample Report window showing actual report data

You can change an inserted TextReportObject's display by selecting the object, right-clicking on it, and editing its properties.

Properties specific to this object include:

Colors
Lets you change the colors of the text displayed. By default, PC-DMIS causes this object to use the same color scheme you used in the Edit window's Command Mode.

When you click this property, PC-DMIS displays a message box asking if you want to create an independent color set. Click **Yes** to access the **Color Editor** dialog box. You can then use this editor to define a new color scheme for the selected TextReportObject.

> See "Defining Edit Window Colors" in "Setting Your Preferences" for information on how to use the **Color Editor**.

Report Header After PRINT
This property determines how PC-DMIS handles a report header *after* a `PRINT/REPORT` command. For information, see the "Header after Print Command" area in the "Edit Text Reporting" topic.

RuleTree
Accesses the **Rules Tree Editor** for this object. This property allows you to create rules to determine when and how expressions or label templates are used in the object.

Label templates and expressions let you control the report data that appears in this object. See "About the Rule Tree Editor".

ShowAlignments
Shows or hides alignments in your report.

ShowComments
Shows or hides comments in your report.

ShowDimensions
Determines which dimensions get displayed in your report.

> `All` - Shows all dimensions.
> `None` - None of the dimensions are shown.
> `Out of Tolerance` - Shows only out of tolerance dimensions.
> `Within Limits` - Shows only dimensions within tolerance limits.

ShowFeatures
Shows or hides features in your report.

ShowHeaderFooter
Shows or hides the report's header and footer information.

ShowMoves
Shows or hides move commands in your report.

ShowScreenCaptures
Shows or hides screen captures in your report.

Text Object

The **Text** object allows you to insert textual labels, descriptions, and markings into your template. This is helpful in making your report more meaningful. You can change an inserted text object's display by selecting the text object, right-clicking on it, and editing its properties.

The properties for this object are described in the "Common Properties" topic.

TextVar Object

The **TextVar** object allows you to insert dynamic, numerical text into your template. This object differs from the **Text** object by deleting the TEXT property and adding these 2 properties:

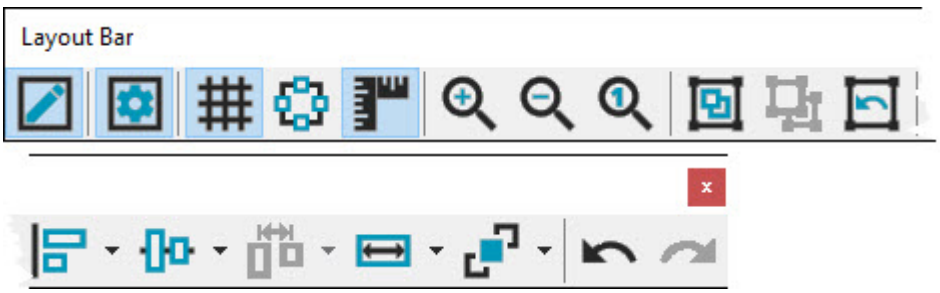
ValueEq
Value equation (This value can be a set constant number, a value evaluated from a simple equation, or a returned value from a VBScript). The value is formatted and displayed at run time.

```
-$, $, CONST
```

Format
This formats the numerical value using 'C' language standards. However, you should use only the numeric portion. For example, the format `%5.2lf` becomes 5.2, `%5d` becomes 5. One exception to this is when using BOOLEAN variables, the $ string gets converted into NO or YES.

To use this object edit the **ValueEq** property so that it's set to display a variable number value.

The Layout Bar



The **Layout Bar** gives you the tools you need to effectively arrange, organize, and display your objects inside the edit mode grid. The following topics discuss the functionality of each icon.

Edit

The **Edit** icon places the template into run mode. The Report Template and Label Template editors always remain in edit mode. This icon only functions when working with the Form Editor.

Property Sheet

The **Property Sheet** icon displays the properties for the selected object in the **Property Sheet** dialog box. The **Property Sheet** dialog box enables you to change the properties for a selected object. To change the object whose properties you're looking at, choose a different existing object from the list at the top of the **Property Sheet** dialog box.

For complete information on properties, see "About Object Properties".

Toggle Grid

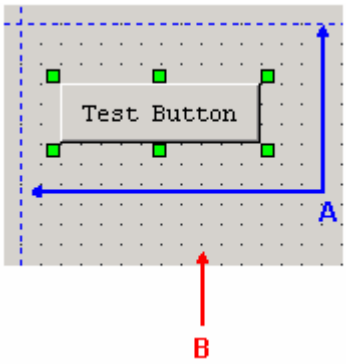
The **Toggle Grid** icon shows or hides the available grid. In edit mode, the grid helps you size objects precisely, because it contains a dot every tenth of an inch. For more information, see "Working with the Grid" below.

Working with the Grid

When working in a form or template editor, PC-DMIS can display or hide a grid on the background of your form or template in these ways:

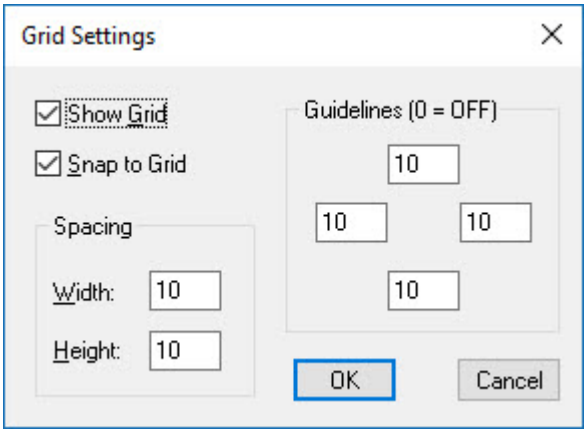
- Click the **Toggle Grid** icon ( ) from the **Layout Bar**.
- Press Ctrl + G.

You can use this grid as a tool for the accurate placing and sizing of objects. It also helps you control a template's margin guidelines.



A - Guidelines
B - Grid markings
Example showing the default grid's markings and guidelines

Select **Edit | Layout | Grid Settings** to access the **Grid Settings** dialog box.



Grid Settings Dialog Box

You can then use this dialog box to customize the grid. A description of the controls in this dialog box follows:

Show Grid check box
The **Show Grid** check box toggles the display of the grid.

Snap to Grid check box
The **Snap to Grid** check box determines whether controls placed on the grid get moved, or snapped, to the nearest grid location.

Spacing area
The **Spacing** area sets the amount of spacing between the grid markings. The smaller you set these numbers, the closer together the grid markings are.

Guidelines area
The **Guidelines** area allows you to determine the location of the top, left, right, and bottom margin guidelines for the template editor.

The value in each of these boxes sets the distance in grid markings to where the guideline is set. Setting the value to zero removes the guideline completely.

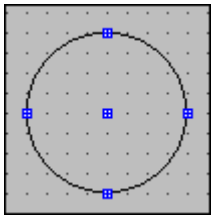
You cannot place objects beyond these guidelines.

View Snap Points

The **View Snap Points** icon shows the snap points on each object. Snap points are useful in tying objects together with lines. For example, you can create a line between a text box description pointing to an object, such as a circle, by using a line to connect snap points on the text box description and the circle object. Then, like a leader line, when you move the circle to a new location, the attached line stretches and shrinks to accommodate the new position of the circle.

To use snap points as described in the above example, follow these steps:

1. Create a circle object.
2. Create a line object.
3. In edit mode, select the **View Snap Points** icon from the **Layout Bar**. You will notice that objects on the grid that have an area display blue-colored points. These blue points are snap points.

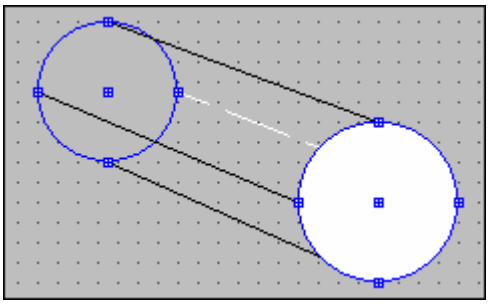
   Example of a circle with five snap points.
4. Select the line and drag one end of it until the end rests directly on the desired snap point of the circle. Try moving the circle object. The line is attached to the circle object's snap point.
5. Create a text object for a description for the circle and edit it with a brief description.
6. Select the other end of the line and drag it until it rests on one of the snap points of the text object.
7. Now select and drag either the text box description or the circle object. Notice that the line object gets longer or shorter and moves so that there's always a straight line between the used snap points of the two objects.

To detach objects attached to a snap point, simply drag the attached end of the object off of the snap point.

Snap Points vs. Leader Line objects

Most of the time you will probably use the **Leader Line** object instead of snap points. It has nearly all the same functionality as snap points and is easier to use (see the "Leader Line Object" topic). One advantage snap points have over the **Leader Line** object, is their

ability to support more than one line between objects and their ability to fit the connecting line exactly to a snap point. For example, with snap points, you can create a more complex shape directly in the edit mode, as shown here:



Example of a cylinder constructed from four lines fitted to snap points on two filled circles.

Supported Objects

Only Line and Polyline objects can attach to objects that have snap points. Below are the objects that have snap points:

- Text
- TextVar
- Border
- Ellipse
- Bitmap
- Dimension Color Key
- Histogram
- Feature Text
- Dimension Info
- Point Info

Viewing Attached Objects

On complex pages with a lot of objects, you may find it useful to use the **View | Route** menu option to see the chain of objects connected to the currently selected object. This menu option selects all objects connected in some way by snap points to the currently selected object.

Toggle Ruler

The **Toggle Ruler** icon allows you to turn on or off the ruler located on the top and left sides of the grid. The ruler shows the size of the grid in inches or centimeters. It also displays your cursor's position along the grid's horizontal axis (top ruler) and vertical axis

(side ruler). You can switch between Inches or Centimeters by right-clicking on the ruler and selecting the desired option from the shortcut menu.

Zoom In or Out

These icons allow you to zoom in or out of your Hyper report and then restore the original view:

Zoom In - This zooms in on your report by 100 percent per click. You can click this icon at most five times to get a 500 percent zoom.

Zoom Out - This zooms out of your report by 100 percent per click if you're already zoomed in. If you haven't zoomed in at all, you will only be able to zoom out by 100 percent one time.

Zoom 1:1 - This restores the size of your report to the original 1 to 1 zoom size.

No matter what you zoom to, PC-DMIS restores the original zoom whenever you activate a control that has an "activated" or "deactivated" mode (for example, double-clicking on the CadReportObject or GridControlObject). When you leave these controls and return to the editing environment, the zoom size goes back to what it was.

Group

The **Group** icon allows you to select multiple objects and group them together. Once grouped, the editor will show the grouped objects as one object.

To group multiple objects, either hold down the SHIFT button and select all objects in the group, or box-select all objects and then click the **Group** icon.

Ungroup

The **Ungroup** icon restores object in a group to their individual ungrouped states. To do this, select a grouped object and click **Ungroup**.

Regroup

The **Regroup** icon regroups the last ungrouped set of objects without your having to reselect the individual objects.

Align Edges

The **Align Edges** icon lets you arrange a set of selected objects along their left, right, top, or bottom edges:

Align Left - Aligns the selected objects along their left edges.

Align Right - Aligns the selected objects along their right edges.

Align Top - Aligns the selected objects along their top edges.

Align Bottom - Aligns the selected objects along their bottom edges.

Align Center

The **Align Center** button provides additional buttons that let you center selected objects either horizontally or vertically:

Center Vertically - Centers the selected objects vertically along the horizontal axis.

Center Horizontally - Centers the selected objects horizontally along the vertical axis.

Space

The **Space** icon provides you with these additional icons that allow you to equally space selected objects either horizontally or vertically:

Space Across - Equally spaces selected objects along the horizontal axis

Space Down - Equally spaces selected objects along the vertical axis

Make Same Size

The **Make Same Size** icon provides you with these additional icons that allow you to equalize the width, the height, or both the width and height of all selected objects:

Make Same Width - Equally sizes the widths of the selected objects

Make Same Height - Equally sizes the heights of the selected objects

Make Same Size - Equally sizes both the widths and the heights of the selected objects

To Front or Back

The **To Front or Back** icon provides you with additional icons that allow you to place a selected object (or objects) in front or in back of other objects. This is helpful when you have multiple objects on top of each other, and you want certain objects to be visible. The available icons include the following:

Bring to Front - Moves the selected objects in front of all other objects so that it is shown on top.

Send to Back - Moves the selected objects behind all other objects so that it is shown beneath.

Notes on Drawing Levels

Moving objects in front of or behind other objects is subject to an internal drawing level used by the different objects. This means you can only bring objects in front of other objects of the same drawing level or in front of a higher drawing level. Here is the drawing order:

CADReportObject - Drawing level 5

This object is drawn behind all objects of a lower drawing level. It can be drawn in front of other level 5 objects.

AnalysisWindow - Drawing level 4

This object can be drawn in front of level 4 or 5 objects.

Leader Line - Drawing level 3

This object can be drawn in front of level 3, 4, or 5 objects.

All other reporting objects - Drawing level 2

These objects can be drawn in front of level 3, 4, or 5 objects.

You can move objects of the same level in front of or in back of each other, but you cannot, for example, bring a CADReportObject in front of a Label Object. This type of sorting is only used by PC-DMIS version 4.x and higher reporting templates or reports. Legacy (HyperView) reports do not use a sort method.

Undo (Action)

The **Undo** icon undoes the last action you took in the editor.

Redo (Action)

The **Redo** icon redoes the last action that was undone when you clicked the **Undo** icon.

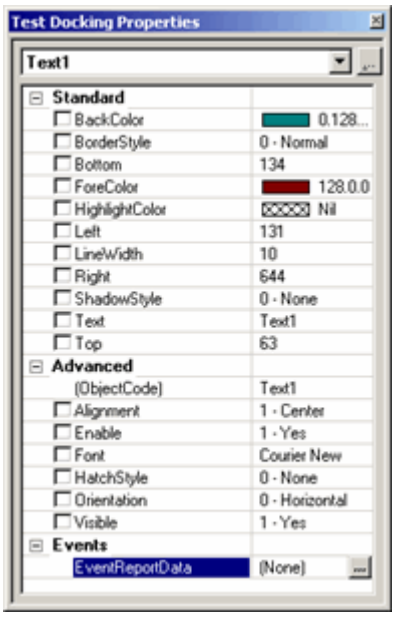
About Object Properties

Each object that you add to a template or form has a set of properties (or attributes) that control the object's position, size, colors, font, text, name, and so on. Each object also has a set of event handler functions that PC-DMIS triggers when certain events occur. For example, an event can occur when you click or double-click the mouse on that object.

You can use the **Properties** dialog box to view and modify properties and event handlers. Another name for this dialog box is "property sheet".

Viewing an Object's Properties

To view the **Properties** dialog box, select an object (single-click within its boundaries) and then right-click. This is an example of a property sheet that shows the text object **Text1** and its various properties:



Properties Dialog Box

Viewing Object Properties

A list appears at the top of the **Properties** dialog box. This list contains an entry for each object in your template or form. It also contains some objects that always exist and you cannot delete, such as the main editing area. In the Report editor, the main editing area is called the **Page**. In the Label Editor and the Form Editor, it is called **The Frame/The View**.

To view the properties for an object in your form or template, select a different object from this list. If you have a report template with more than one section, this list shows only the objects in the current section.

For report templates, you can access an additional object called **Report** only from this list. It has two properties in the **Advanced** category that you can use to specify your own external executable (an .exe file) to collect user-assigned properties. These properties are **Template Selection Program** and **Measurement Routine Execution Program**. PC-DMIS runs the specified measurement routines while you select a template and execute a measurement routine, respectively.

Property Organization

Each object's properties are in three main categories:

- **Standard** - These properties control color, position, and line styles.
- **Advanced** - These properties control whether the object is enabled or visible.
- **Events** - These properties enable you to use the BASIC language to code specific instructions when events occur.

To expand a category, click the **+** symbol to the left of the category. To collapse a category, click the - symbol.

ActiveX and OLE objects have an additional section. For more information, see "ActiveX Object" and "OLE Object".

For descriptions of properties, see "Common Properties". If that topic does not contain a description of a property, view object-specific documentation in "The Object Bar".

For more information on events, see "About Events and Visual Basic Code".

Name Identifier

When you insert an object into a template or form, PC-DMIS automatically assigns a unique name identifier to the object's **ObjectCode** property. Subsequent objects of the same type have the same name, with an incremented numerical value behind it. For example, the editor automatically names your first **Text** object "Text1". It names subsequent **Text** objects "Text2", "Text3", and so on.

You can change this value to a different name. If you change it, ensure that you give it a unique name. If you modify that object's name identifier, you must also update any BASIC scripting that uses an object's code.

Changing Property Values

To modify a property, select it and then click its value. If a list appears, select a new value. You can also type a new value in an editable field.

Expressions as Property Values

Some property values can hold reporting expressions instead of a constant value, just as a cell from a **GridControlObject** can hold expressions. Remember to preface the expression with an equal sign (=). For example, suppose that you have a **Text** object in a label template, and you also want it to display the current feature's ID instead of other text. In the **Text** property, type "=ID". When PC-DMIS executes the measurement routine, the object pulls the feature's name from PC-DMIS and displays it in that **Text** object.

Common Properties

The following are some common properties used on various objects. They make up the core set of properties for many other objects. Properties specific to an object are documented in that object's documentation. Events properties are detailed in the "About Events" topic.

Common Standard and Advanced Properties

Accelerator
This allows you to define the letter to use in conjunction with the Alt key to create an accelerator for the control. The letter or symbol defined should be unique from those used for other accelerators (including the accelerators on the menu bar). This is used with most form controls.

Alignment
Text alignment within the text object’s defining rectangle.

AnchorSnaps
Field that allows you to create, delete, or modify snap points in the **Enter Snap Points** dialog box. Snap points are found on the ellipse, rectangle, text box, or bitmap image objects.

You can connect **Line** or **Polyline** objects to an object's snap points by dragging the tip of the line onto the displayed blue snap point.

Once connected, when you drag the object to a new location, the resulting line remains connected to the snap point, rotating, elongating, and shrinking as necessary. This feature is helpful when you need to create a "leader-line" that constantly points to an object that you'll frequently be moving.

To display snap points, from the **Layout** toolbar, click the **View Snap Points** icon .

Auto Size
This determines whether or not the object sizes itself automatically based on the length of the text it contains.

> If this is set to `YES` and `Word Break` is set to `NO`, then the text object automatically adjusts its width.
>
> If this is set to `YES` and `Word Break` is set to `YES`, then the text object automatically adjusts its height.

BackColor
Background color of the object. This color is used to fill the object's defining rectangle.

`Nil` - This stands for no value. It means the background will be transparent, allowing other colors of objects behind this object to show through. By default, anything with a BackColor property is set to `Nil`.

Setting the color to `Nil` sets the color to RGB(255,255,254) which is then made transparent within the Report window.

If you try to specify a color of RGB(255,255,254), PC-DMIS automatically sets it to RGB(255,255,255), a non-transparent color. This does not affect how the object is displayed.

Be aware that if your label has a bitmap image that uses the color RGB(255,255,254), PC-DMIS sets that color to transparent inside the Report window.

BorderStyle
Style of border drawn around the object.

Bottom
Distance, in pixels, of the bottom of the object's rectangle from the top of the template or form.

CursorPointer
Cursor displayed when the user drags the mouse over the object.

Enable
Enable or disable the object. A disabled object may still be visible, but does not respond to user input.

Font
Font used for text rendered in the object.

ForeColor
Foreground color of the object. In the case of the text object, specifies the text color.

Group
This is used with form objects, such as buttons, check boxes, radio buttons and so on.

HatchStyle
This sets the type of pattern (or hash) to use with the object. Available patterns are:

Horizontal

Vertical

Diagonal

Reverse Diagonal

Cross

Diagonal Cross

Objects that use this include: Border, Ellipse, Histogram, TextVar

HelpContextID
This is the context ID of a specific help topic to call. This is often used in conjunction with the Help button type which invokes WinHelp( ).

Layer
This assigns the current object to the selected layer.

Left
Distance, in pixels, of the left side of the object’s rectangle from the left side of the template or form.

LineStyle
This changes the objects line style (or border style) to `Solid`, `Dash`, `Dot`, `Dash-dot`, and `Dash-dot-dot`.

LineWidth
This sets the line's (or border's) width in pixels.

(Object Code)
Unique name that identifies an object in a template or form. Whenever you need to access a property or method of an object using the BASIC language, use the following syntax:

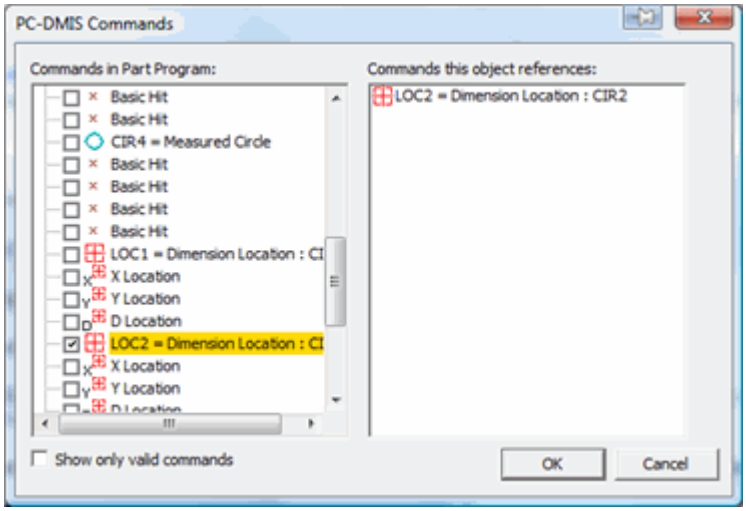
{Object Code}.{property or method name}

Orientation
Orientation of the text inside the object. Horizontal text flows from left to right. Vertical Up flows from bottom to top. Vertical Down flows from top to bottom.

Padding
This property adds pixels of white space around the text by the given amount.

PC-DMIS Commands
(Appears only for objects added to the Custom Report Editor)

Adds or Removes command references to existing objects in your report. Select an object in your report, and select this property. The **PC-DMIS Commands** dialog box appears.



PC-DMIS Commands Dialog Box

By default, all the commands in your measurement routine appear in the expandable and collapsible list on the left. However, for some objects, namely the Label object, GridControlObject, Analysis object, and DimensionHistogram object, some commands may not be valid. In those cases they are disabled in the list. You can choose to hide any disabled commands in the left-hand list by selecting the **Show only valid commands** check box.

The right-hand list displays all the commands referenced by the object.

1. Select the commands from the left-hand list that you want the selected object to reference. PC-DMIS displays them in the right-hand list if the object supports taking multiple commands. However, in most cases, you can only select one command.

2. Click **OK**. The dialog box closes and PC-DMIS now references the new command in the selected object.

This essentially provides an alternate approach to updating an object to use different data from the measurement routine. See "Dragging and Dropping Information into a Custom Report" for the usual way of updating objects.

For the CADReportObject or the AnalysisObject, features whose check boxes are marked are added to the object. Features whose check boxes are cleared are removed from the object. If nothing is marked, PC-DMIS displays items according to the `HideAllPointsonReport` and `HideAllMeasuredFeaturesOnReport` entries in the **Reporting** section of the PC-DMIS Settings Editor.

For objects that support multiple commands (such as the GridControlObject), this dialog box lets you change the order of referenced commands.

PC-DMIS References
(Appears only for objects added to the Custom Report Editor)

Tracks the commands referenced by the selected reporting object. During report generation, for each command, PC-DMIS checks the command references of each object. If that object indicates that it has a reference to the command being processed, PC-DMIS sends the command data to that object.

Most objects only support one command at a time, but some objects can support data from multiple commands (such as the GridControlObject). If you drag and drop commands into the Custom Report Editor, PC-DMIS takes care of managing the references for you internally. You should only need to use this property if you decide to have manual control over which commands are referenced for some reason.

PenWidth
Width of the pen used to draw the object. In the case of the text object, the pen width only affects the border drawn around the text.

Right
Distance, in pixels, of the right side of the object’s rectangle from the left side of the template or form.

ShadowStyle
This sets the location of the 3D shadow on the resulting object. Available locations are:

`Top/Left`, `Bottom/Right`, or `None`

Tag
Place holder that stores user-defined data.

Text
Text appearing in the text object. You can also type an ampersand symbol (&) in front of any alphanumeric character in this field to create a shortcut key (shown as an underlined character). Once defined, you can press the key along with the Alt key in run mode to cause that control to immediately have the focus. If you plan on using a lot of shortcut keys, you can ensure they are unique by using the **Edit | Layout | Check Mnemonics** menu item in edit mode.

Tabstop
This determines if pressing the Tab key cycles to the object. This is used with form objects, such as buttons, check boxes, radio buttons and so on.

ToolTipText
Text displayed when you move the mouse over the object inside run mode.

Top
Distance, in pixels, of the top of the object's rectangle from the top of the template or form.

ValueID
This contains the value of the ID, or on supported objects it contains the `CurPage` and `NumPages` variables. You get and set this property using the control. This is used with most of the form objects.

Visible
Visibility state of the object. A value of `1` corresponds to visible, `0` to hidden.

Word Break
Determines whether or not the text object can hold more than one line of text. If set to `YES`, then PC-DMIS wraps text strings that exceed the width of the object container within the object container. If set to `NO`, then text strings remain on a single line.

TheFrame/TheView Properties

The main editing areas in the Label Template Editor and Form Editor contain a special object called **TheFrame/TheView**. This object also contains properties that you can access and set just as with any other object.

- Properties that are followed by a single asterisk (*) are used in the Label Template Editor only.
- Properties that are followed by two asterisks (**) are used in the Form Editor only.
- All other properties are used in both the Label Template Editor and the Form Editor.

Height
Defines the height, in pixels, of the main editing area.

Horizontal Positioning **
Sets the horizontal position of the form within the window when executed.

LocalDecls
Declares variables with a scope that is available to your entire form or label.

LocalVariables **
See the "Declare Global Variables Using the LocalVariables Property" topic.

MaximizeBox **
If set to **Yes**, this property displays a **Maximize** icon in the upper-right corner of the form. You can click this icon in Run mode or during execution to maximize your form window to fill the available space.

If MaximizeBox is set to **Yes** but MinimizeBox is set to **No**, then the **Minimize** icon still appears but is unavailable for selection.

MinimizeBox **
If set to **Yes**, this property displays a **Minimize** icon in the upper-right corner of the form. You can click this icon during run mode or during execution to minimize your form window to the taskbar.

If MinimizeBox is set to **Yes** but MaximizeBox is set to **No**, then the **Maximize** icon still appears but is unavailable for selection.

Page Orientation *
Sets the label to one of the predefined standard orientations, either **Portrait** or **Landscape**.

Page Size *
Sets the label to one of the page sizes available on your default printer. You may see items such as Letter, A4, or one of several other sizes depending on your type of printer.

ReportLock **
Determines whether PC-DMIS can clear data from a form. If this is set to **Yes**, PC-DMIS does not clear data from the form. If it's set to **No**, PC-DMIS clears data from the form that is collected over time.

Supported Command Types *
Displays the **Supported Command Types** dialog box. With this dialog box, you can define the commands that should use this label template. Then in the Report window, when you right-click on a label and select the **Change Report** menu item, PC-DMIS only displays those labels that support the selected command type.

ToolTipText
See the "Common Properties" topic.

Vertical Positioning **
Sets the vertical position of the form within the window when executed.

Width
Defines the width, in pixels, of the main editing area.

Section and Page Properties

The main editing areas in the Report Template Editor and the Custom Report Editor are special objects called the **Section** and the **Page**, respectively. These objects contain properties that you can access and set just as with any other object.

Command Set
Defines how commands get processed within the current section (**Page** tab).

All Commands - The setting processes the entire command set in the current section (**Page** tab). This is the default value.

Continue from Previous Section - This setting continues to process the command set from the previous section, continuing where the last section stopped processing.

Same as Previous Section - This setting causes the section (**Page** tab) to take exactly the same command set as the previous section. The head section and all subsequent sections with this setting form a mini-group. All subsequent sections in the mini group create one page with the head page.

(Form Name)
Defines the name of the page.

Height
Defines the height, in pixels, of the main editing area.

LocalDecls
Provides a window for you to declare variables local to this object. Once declared, you can use these variables in events that affect the object. This type of declaration lets you give the variable a value in one event and then check that value in a different event. The variable is safe from accidental modification since it is invisible to other objects in the report or form.

LocalVariables
See the "Declare Global Variables Using the LocalVariables Property" topic.

Maximum Number Of Pages
Defines the maximum number of report pages that use the design of the current **Page** tab.

For example, if you set this value to 2, but you have easily enough dimensions in your measurement routine to fill five pages, the section does not process commands once the number of pages for that section reaches the specified maximum of two pages. The current process command needs a newly-created section (**Page** tab) to display additional information.

The default value of 0 means there is no maximum and so all pages are displayed.

ToolTipText
Common Properties

Width
Defines the width, in pixels, of the main editing area.

Examples of "Command Set" and "Maximum Number of Pages" Properties

In the following examples, suppose that your report template has four sections named A, B, C, and D, and that each one contains a TextReportObject. Additionally, suppose that your measurement routine has enough dimensions to display three pages of dimensions per section. For each example, a table details the property settings.

Example 1

Sections	Maximum Number of Pages	Command Set
A	0	All Commands
B	0	All Commands
C	0	All Commands
D	0	All Commands

In this example, PC-DMIS displays the pages in the final report like this:

A1,A2,A3,B1,B2,B3,C1,C2,C3,D1,D2,D3

Example 2

Sections	Maximum Number of Pages	Command Set
A	1	All Commands
B	0	All Commands
C	0	All Commands
D	1	All Commands

In this example, PC-DMIS displays the pages in the final report like this:

A1,B1,B2,B3,C1,C2,C3,D1

Example 3

Sections	Maximum Number of Pages	Command Set
A	1	All Commands
B	0	All Commands
C	0	Same As Previous Section
D	1	All Commands

In this example, PC-DMIS displays the pages in the final report like this:

A1,B1,C1,B2,C2,B3,C3,D1

Example 4

Sections	Maximum Number of Pages	Command Set
A	1	All Commands
B	0	Continue From Previous Section
C	0	Same As Previous Section
D	1	All Commands

In this example, PC-DMIS displays the pages in the final report like this:

A1,B2,C2,B3,C3,D1

Working with User Assigned Properties

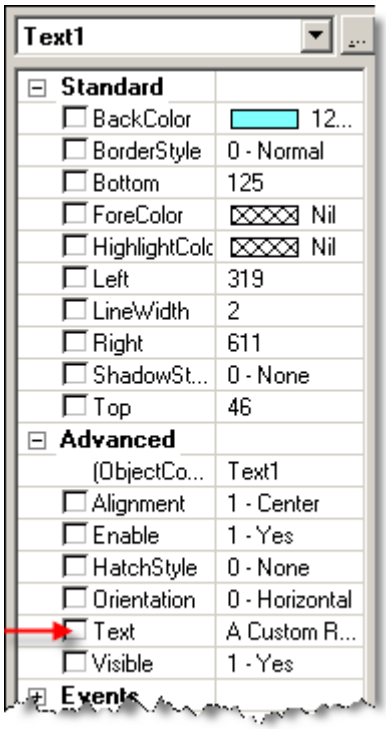
In the Report Template Editor, the **Properties** dialog box contains the ability to make a property user definable. This allows you to let measurement routine operators change object properties at runtime.

For example, suppose you want your report title to change dynamically according to whatever the current measurement routine operator wants. You can add an interactive approach such as this to your reports by using this functionality.

To Make a Property User Assigned

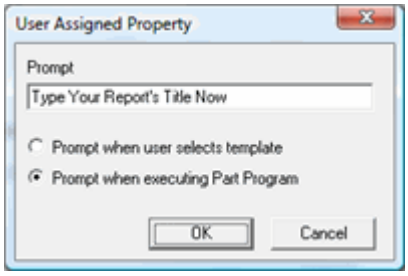
1. Select an object in one of the editors. For the example above, you would select a **Text** object.

2. Right-click on the object. This causes the property sheet for that object to appear on the right side of the editor. On the left side of the property sheet, you will see check boxes next to most properties. These check boxes determine what properties you can make user definable.



Example of a text object's property sheet with its user-definable check boxes on the left.

3. Select the check box next to the property that you want to make definable. For the example above, you would select the **Text** property for a **Text** object. A **User Assigned Property** dialog box appears, asking you to determine the message that will prompt the user for the property value and when the user should be prompted.



User Assigned Property Dialog Box

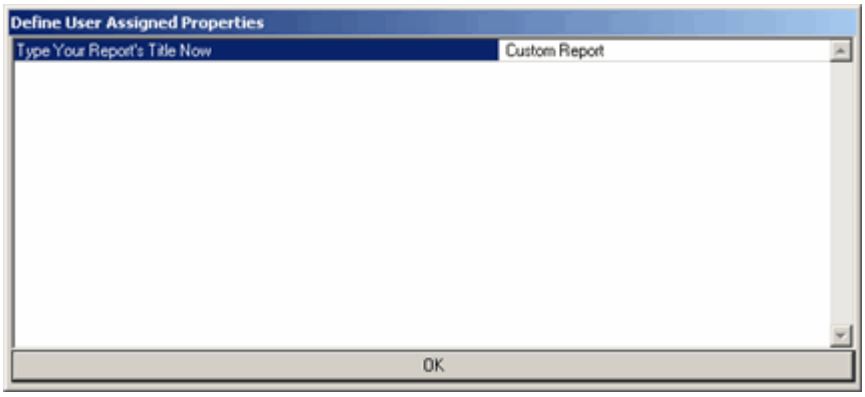
4. In the **Prompt** box, type the prompt the user will receive, and then select from one of the two option buttons when the user will receive the prompt. You can choose to prompt the user either during template selection or measurement routine execution.

5. Click **OK**. The **User Assigned Property** dialog box closes.

6. Save the report template and return to your measurement routine.

7. To test this, use the Report window and click the **Template Selection** icon. Select the report template from the dialog box and then execute your measurement routine.

8. At some time during the previous step, PC-DMIS will display the **Define User Assigned Properties** dialog box to prompt you for the properties' values according to the option button you selected in the **User Assigned Property** dialog box.

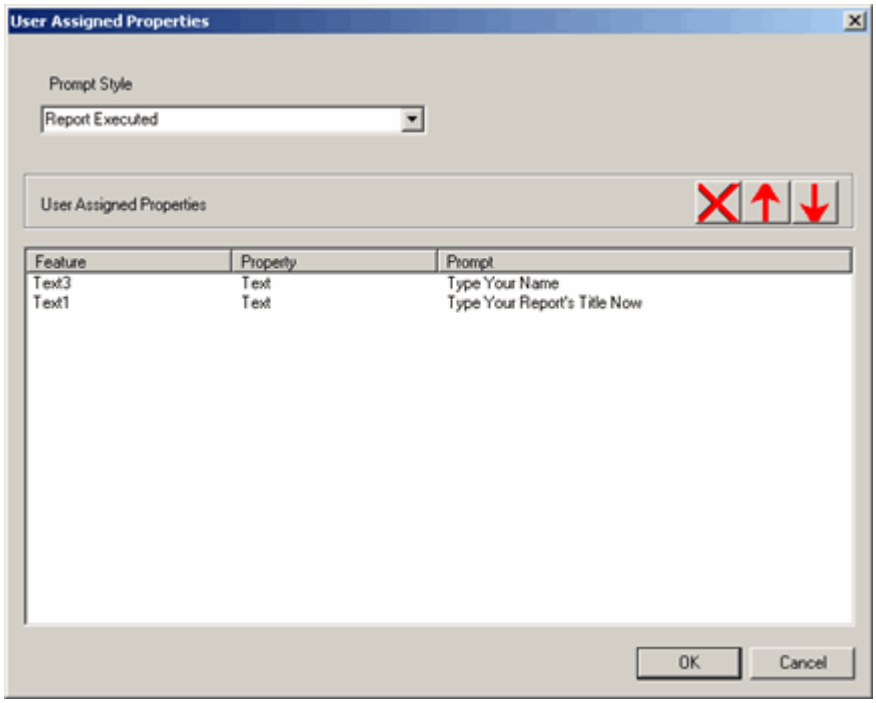


Define User Assigned Properties Dialog Box

To View and Manipulate all Your User Assigned Properties

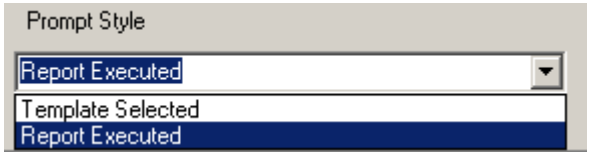
1. Access a report or label template that has user definable properties.

2. Select **Edit | User Assignable Properties**. The **User Assigned Properties** dialog box appears. This dialog box shows those properties that will be presented during template selection or measurement routine execution.



User Assigned Properties Dialog Box

3. Select an item from the **Prompt Style** drop-down list to filter what gets displayed in this dialog. The user assigned properties appear in the list box below. The list box tracks all the properties, what objects they belong to, and their prompts.



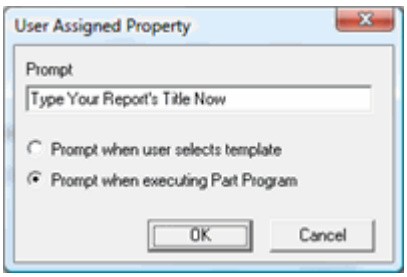
- *To remove a property*, select the property and click the red X. That object's property will no longer be user definable and will use the default value already defined in the template.

  Delete Property Button

- *To reorder a property*, select the property that you want to move and then click the red up or down arrows. This will move the selected property up or down in the list.

  Up and Down Arrow Buttons

- *To change a property's prompt or when the prompt should appear*, double-click on the property in the **Feature** column. PC-DMIS displays the **User Assigned Property** dialog box, allowing you to make the change.



User Assigned Property Dialog Box

If you selected **Prompt when user selects template** from the **User Assigned Property** dialog box, PC-DMIS will only prompt you once for any user assigned values. Thereafter, for that measurement routine, PC-DMIS will automatically use the initial property value until you clear out all template associated data. See the "Clearing Template Associated Data" topic in this chapter.

If you selected **Prompt when report is executed** then PC-DMIS automatically displays the **Define User Assigned Properties** dialog box each time the measurement routine executes.

Clearing Template Associated Data

The **File | Reporting | Clear Template Associated Data** menu item removes the following items used in the current measurement routine with the current selected report template:

- Any user-assigned property values. By clearing these, PC-DMIS will again prompt you for values to any user assigned properties in the current template.
- Table customizations on labels.
- Label placement customizations around **CADReportObject** objects.
- Changes to the rotation or zoom factor of any **CADReportObject** objects.
- Any other object modifications, additions, or page duplications.

About Events and Visual Basic Code

This section assumes that you have at least some background with the Visual Basic programming language. If you need to learn Visual Basic, please consult a book or other external resource on the subject.

The **Events** section on the **Properties** dialog box contains a list of event handler functions that your template or form object supports. An event handler function is called whenever

the event implied by the name of the function occurs for the given object. When that function gets called, any Visual Basic code inside that function gets executed.

Available Event Handler Functions

EventClick
Called when the mouse is clicked on the object.

EventDblClick
Called when the mouse is double-clicked on the object.

EventDragDrop
Called when a "draggable" item is dropped onto the object.

EventDragEnter
Called when a draggable item is initially dragged onto the object.

EventDragOver
Called when a draggable item is dragged over the object.

EventInitialize
Called when the form first enters into run mode before any other handlers are called on the object.

EventMouseDown
Called when a mouse button is clicked down while over the object. If you press the Shift key, parameters passed to the function indicate which button is pressed as well as the pointer's coordinates at the moment you clicked the mouse button.

EventMouseMove
Called when the mouse pointer passes over the object. If you press the Shift key, parameters passed to the function indicate which button is pressed as well as the pointer's coordinates when you move the pointer over the object.

EventMouseUp
Called when you release the mouse button over the object. If you press the Shift key, parameters passed to the function indicate which button is pressed as well as the pointer's coordinates at the moment you release the mouse button.

EventPumpData
This is a widely used handler function in the Form Editor. **EventPumpData** is called whenever BASIC variables that may affect the object change value. Your code can also force **EventPumpData** to be called on individual objects (or on all objects in a report) when you wish an object to update itself. Basically, any code that an object needs to reinitialize itself based on variable values or the state of other objects should occur in **EventPumpData**.

EventTerminate
Called when the report is being closed or when you toggle from Run mode to Edit mode in the Form Editor.

This handler function is only available for the **Section** editing environments.

EventReportData
This is another most widely used handler function in the Report and Label Template editors. It gets called whenever report data gets plugged into the current template.

Its primary function is to let you create your own ActiveX controls that get placed into a label. For example, PC-DMIS's best fit and feature analysis controls use this event to send data to the ActiveX control from the command. To see this, open the label named best_fit_analysis.lbl, click on the ActiveX control inside the label, and then look at the BASIC code inside its event handler. A single line passes data from the command into the control.

With data passed to your own ActiveX control, you can then use automation commands to extract and manipulate the data.

For details on PC-DMIS Basic, see the "PC-DMIS Basic" topic in the PC-DMIS Basic documentation.

For details on PC-DMIS Automation Object Library, see the "PC-DMIS Object Library" topic in the PC-DMIS Automation Objects documentation.

This event gets called once for each command when applying the report template or during measurement routine execution.

Adding Code to Event Handlers

In order to add BASIC language code to any of the event handlers:

1. Click on the function’s entry from the list of event handlers. A Mini Visual BASIC language editor opens, with the opening and closing statements of the subroutine already filled in.
2. Type additional BASIC language statements code into the editor.
3. Click **OK**.
4. Test your code.
    - *If you are using the Form Editor*, press Ctrl + E to enter Run mode to test your form.
    - *If you are using one of the template editors*, apply the template to a report and redraw the report to test your code. To do this, use the **Redraw Report** icon from the **Reporting** toolbar.

Since you don't interact with clickable buttons, lists, fields, or other interactive elements when you use a report, the only event handler function available is the **EventReportData** function.

Event Example 1: Calling Code on an EventClick Event

This event example uses a **Button** object in the Form Editor.

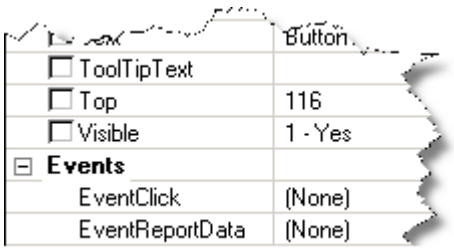
1. Select **File | Reporting | New | Form Report** to open the Form Editor.

2. First, insert a button into your form by clicking on the **Button** icon ( ) on the **Object** toolbar and then dragging out a rectangle for the button onto the form.

3. Make sure that the button is selected (highlighted).

4. Right-click on the button to bring up the dialog box that shows the button's properties.

5. If the list is not expanded, click on the **+** sign next to the **Events** section to expand the list. You should see a list of all of the available event handler functions for the button object.

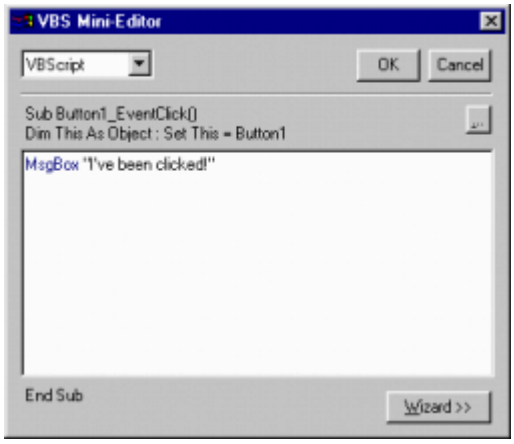


Example of a button object's events.

6. Double-click the **EventClick** function in the list. A Visual BASIC Editor entitled **VBS Mini-Editor** opens.

7. Add the following code to the **EventClick** function inside the VBS Mini Editor:

```
MsgBox  "I've been clicked."
```



8. Click the **OK** button in the upper right-hand corner of the **VBS Mini-Editor**. Close the **Property Sheet**.
9. Save the report by selecting **File | Save**. You are now ready to test the report in run mode.
10. Switch to run mode (press CTRL + E).
11. Click on the button you created. A message box with the text **I've been clicked!** should appear.

This simple example illustrates the basics of scriptable forms and event handlers.

Event Example 2: Modify Object Properties on Triggered Event

This exercise follows the above exercise, you should have completed it first. This example also uses the Form Editor.

Suppose you want to modify the properties of another object (say the text in a text object) when the button has been clicked.

1. In the Form Editor, switch back to edit mode (press Ctrl + E).
2. Create a new text object.
3. Open the text object's properties (select and then right mouse click).

4. Make sure that the text item's **Object Code** property reads "Text1". The **Object Code** property is the name that is used to refer to that object from BASIC code.
5. If not expanded, click on the **+** sign next to the **Events** section to expand the list.
6. Double-click on the **EventInitialize** handler function. The **VBS Mini-Editor** opens.
7. Add the following code to the **EventInitialize** function:
   ```
   Set This  = Text1
   ```
8. Press the Enter key to get to a new line. On the next line, type this code:
   This.Text = "The button has not yet been clicked"
9. Click the **OK** button to close the editor.
10. Once the object Text1 has been set up as described, open the property sheet for the button object added in the prior exercise.
11. Click the **Events** tab.
12. Double-click on the **EventClick** function.
13. Replace the basic code you added in the previous example with the following two lines:
    Line 1: `MsgBox  "I've been clicked, and am about to modify Text1's text"`

    Line 2: `Text1.Text  = "The button was clicked."`
14. Click the **OK** button to close the **VBS Mini-Editor**.
15. Close the **Property Sheet**.
16. Switch to run mode (press Ctrl + E). The code you added to Text1's **EventInitialize** function sets Text1 to initially read "The button has not yet been clicked".
17. Now try to click the button. The "**I've been clicked, and am about to modify Text1's text**" message should appear. As soon as you close the message box, Text1's text should be modified to read "**The button was clicked.**".

This example illustrates how the properties of any object in a form may be accessed from Basic code via the **ObjectCode.property_name** syntax.

Access Object Methods

Many of the objects support various methods in addition to their properties. You can access these methods from BASIC code via the **ObjectCode.method_name** syntax. This means you can see what properties and methods are available for any given object by typing the object's Object Code in the **VBS Mini-Editor** followed by a period. As soon as

the period is typed, a mini wizard pops up showing the available methods and properties for that object.

- A property is a routine that modifies how an object appears or behaves. This is similar to an adjective in a language.
- A method is a routine or function that performs an action. This is similar to a verb in a language.

So if you had a text object named "Text1" and you type "Text1" followed by a period, PC-DMIS shows a list of available methods or properties for a text object.

Variables declared within an event handler function (via the BASIC Dim **var_name** as **var_type** syntax) are in scope only during that event handler function. There will likely be situations where you want to declare variables, functions, or subroutines that are accessible from all BASIC code in a template or form—variables with global scope for a particular report. The template and form editors provide a place to make these declarations.

Declare Global Variables Using the LocalVariables Property

If you wish to declare global variables of any of the standard data types (String, Integer, Double, etc.) and have them show up in the Basic Wizard, you can use the **LocalVariables** property of **TheFrame/TheView or Section**.

To use the **LocalVariables** property:

1. Select then right-click an object to display the **Property Sheet** for **The Frame/The View**.
2. Select the **Settings** tab.
3. Scroll down to the **LocalVariables** property.
4. Double-click the **LocalVariables** property to display the **Local Variables** dialog box.

Local Variables Dialog Box

The **Local Variables** dialog box allows local variables of any of the basic types to be added, removed, named, and initial values to be set. Again, these variables are global to

this template or form and are accessible from any of the BASIC code written for this template or form. They also appear in the **Local Variables** area of the **Script Wizard**.

The name "LocalVariables" is something of a misnomer, since variables declared in the **Local Variables** dialog box actually have a global scope to that particular template or form. It indicates that the variables declared here are "local" to this particular template or form and cannot be accessed on other forms. Nevertheless, they are set as global variables, available to other routines elsewhere in the template or form they are defined from.

Declare/Define Variables, Functions, or Subroutines

To declare and/or define variables, functions, or subroutines that are to be accessible from all BASIC code in a report, use the LocalDecls property of **TheFrame/TheView** found on the PC-DMIS tab of the Property Sheet.

To do this:

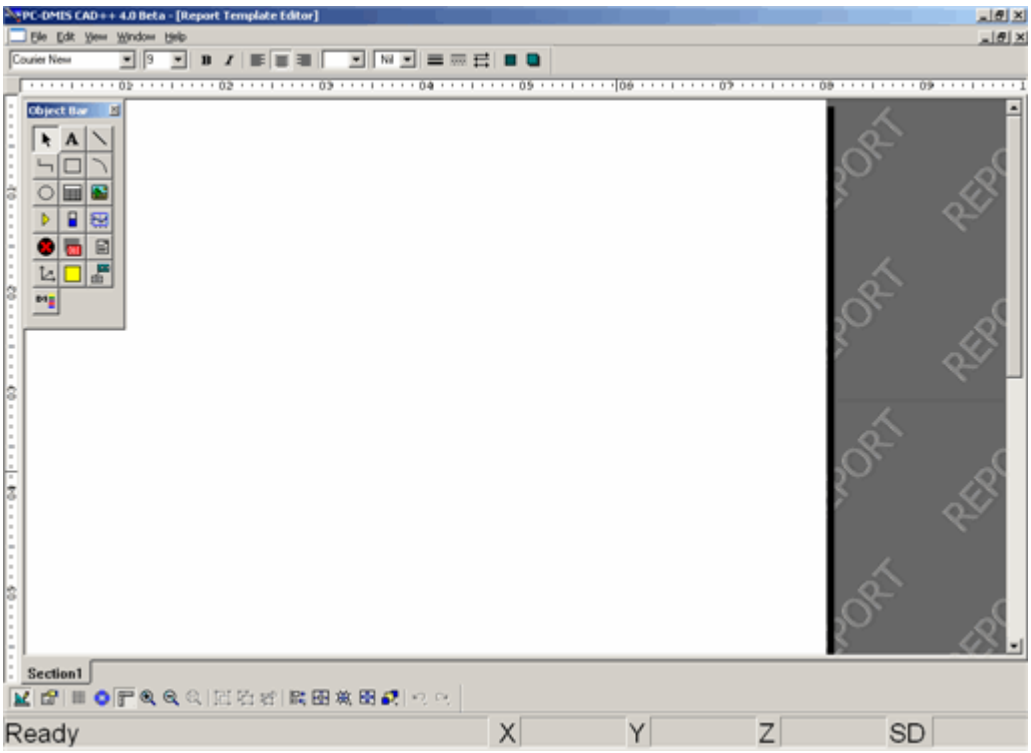
1. Open the Property Sheet for an object.
2. Select **TheFrame/TheView** from the list.
3. Click the **PC-DMIS** tab.
4. Double-click the **LocalDecls** item on the property sheet to display the **VBS Mini-Editor**.

You may add any BASIC code you wish to this window, including variable declarations and functions, or subroutine definitions. Any variables, functions, or subroutines declared here will be global, but will not appear in the **Script Wizard**.

Understanding the Report Template Editor

To access the Report Template Editor, select **File | Reporting | New | Report Template**. To open an existing report template, select **File | Reporting | Edit | Report Template**.

PC-DMIS displays the Report Template Editor in a new window.



Report Template Editor

The Report Template Editor acts as a work area. It enables you to drag, size, and place various objects, and set their properties. You place your objects on top of the white area, called the template or section. If you have worked with legacy (HyperView) reporting in previous versions of PC-DMIS, you will feel at home with this editor since it essentially functions much the same way and contains many of the same user interface elements.

With the template approach to reporting, you cannot place the Report or Label template editors into run mode. They only open in edit mode and cannot be switched. Run mode still functions with the Form Editor.

Run Mode - This mode only works with the Form Editor. In Run Mode you can run (or execute) your form, testing it for how it would look or behave in an actual measurement routine execution environment.

Edit Mode - This is the default mode for all the reporting editors and is the only mode for the Report and Label template editors.

You can switch between run and edit modes by pressing CTRL + E.

The editor contains the following elements:

- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)
- The Layout Bar (toolbar)
- Object Property Sheets

- Sections
- Working with the Grid

About Sections

Sample Sections

Sections make up the main editing area in a report template. These tabbed, scrollable areas in the Report Template Editor are where you can insert objects into your current report template. Sections give you greater control over when objects appear in your report.

With the Report Template Editor, you can add additional sections to your template and create powerful multi-section reports. Sections, however, are not the same as pages. Objects inserted into one section will not appear in other sections, but will appear in multiple pages of the same section. This approach lessens the complexity of creating report items like headers and titles that you want to remain static.

For example, you can choose to place a description of the report and author data on the first section of your template, a **TextReportObject** onto the second section, and a **CadReportObject** onto the third section. Then, depending on the length of your report data, each section in your report may spawn up to several pages.

Sections are *only* used in the Report Template Editor. Before PC-DMIS version 4.0, the main editing area for HyperView reports was called "TheFrame/TheView". This name is still used in the Form Editor and the Label Template Editor where sections are not used. For the Report Template Editor, the editor's "Section" is where you place your reporting objects.

To Add a New Section

1. Make sure your Report Template Editor window is maximized.
2. Scroll to the very bottom of your template.
3. Right-click on the **Section1** tab.
4. Select **Add Tab** from the shortcut menu. PC-DMIS inserts a new section, **Section2**, into your template. Your template should now have two sections, like this.

To Delete a Section

1. Make sure your Report Template Editor window is maximized.
2. Scroll to the very bottom of your template.
3. Right-click on the section tab you want to delete.
4. Select **Delete** from the shortcut menu. PC-DMIS deletes the section.

To Modify a Section's Properties

1. Make sure your Report Template Editor window is maximized.
2. Scroll to the very bottom of your template.
3. Right-click on the section itself. The **Properties** dialog box appears.
4. Modify the properties as desired. To rename a section, rename the section's **FormCode** property.

Spanning Command Sets Across Sections to Control Page Layout and Output Collated Pages

Two section (Page) object properties provide a way to span a command set from one report template section to another. This enables you to create different pages with different layouts while using the same command set. You can also use these properties to output the report as collated pages. See the "Section and Page Properties" topic for information and examples on the **Command Set** and **Maximum Number of Pages** properties.

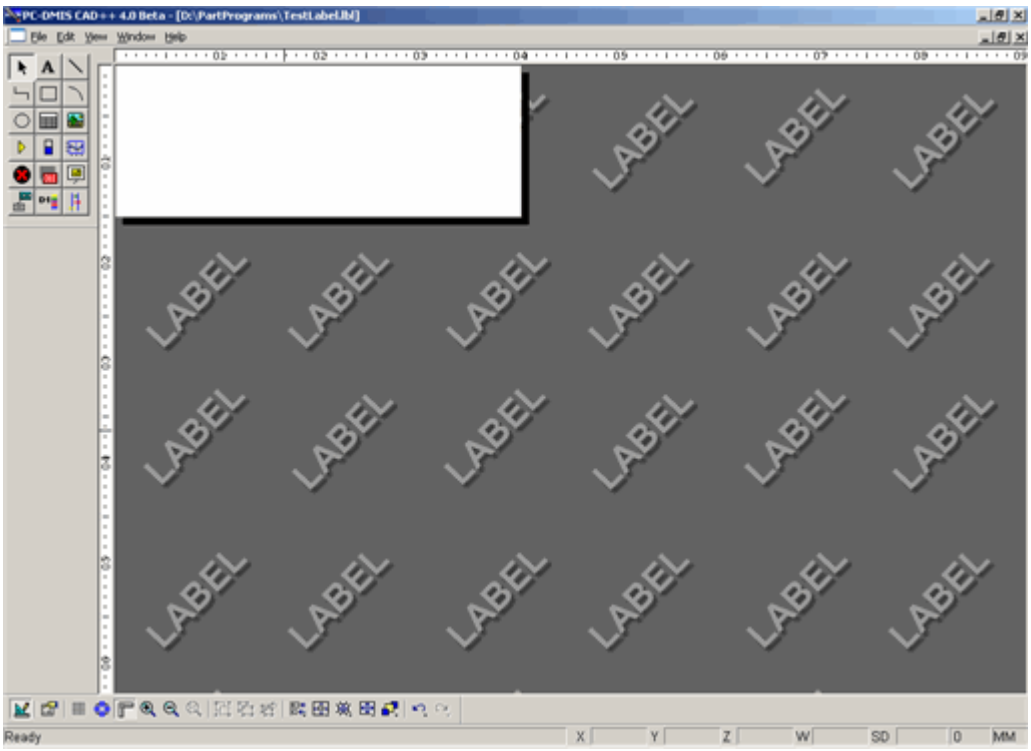
Properties Specific to Sections

See "Section and Page Properties".

About the Label Template Editor

To access the Label Template Editor, select **File | Reporting | New | Label Template**. To open an existing label template, select **File | Reporting | Edit | Label Template**.

PC-DMIS displays the Label Template Editor in a new window.



Label Template Editor

This editor is similar to the Report Template Editor and the Form Editor. With the exception of some differences in its **Object Bar**, the user interface is essentially the same. While you can technically put any object from the Label Template Editor's Object Bar into a label template, you will generally work with the GridControlObject. The label template is the white and sizable box on top of the "LABEL" background.

With the new template approach to reporting, you cannot place the Report or Label template editors into run mode. They only open in edit mode and cannot be switched. Run mode still functions with the Form Editor.

> **Run Mode** - This mode only works with the Form Editor. In run mode, you can run (or execute) your form. This enables you to test how it would look or behave in an actual measurement routine execution environment.
>
> **Edit Mode** - This is the default mode for all the reporting editors and is the only mode for the Report and Label template editors.
>
> To switch between run and edit modes, press CTRL+E.

The editor contains the following elements:

- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)
- The Layout Bar (toolbar)

- Object Property Sheets

About the Custom Report Editor

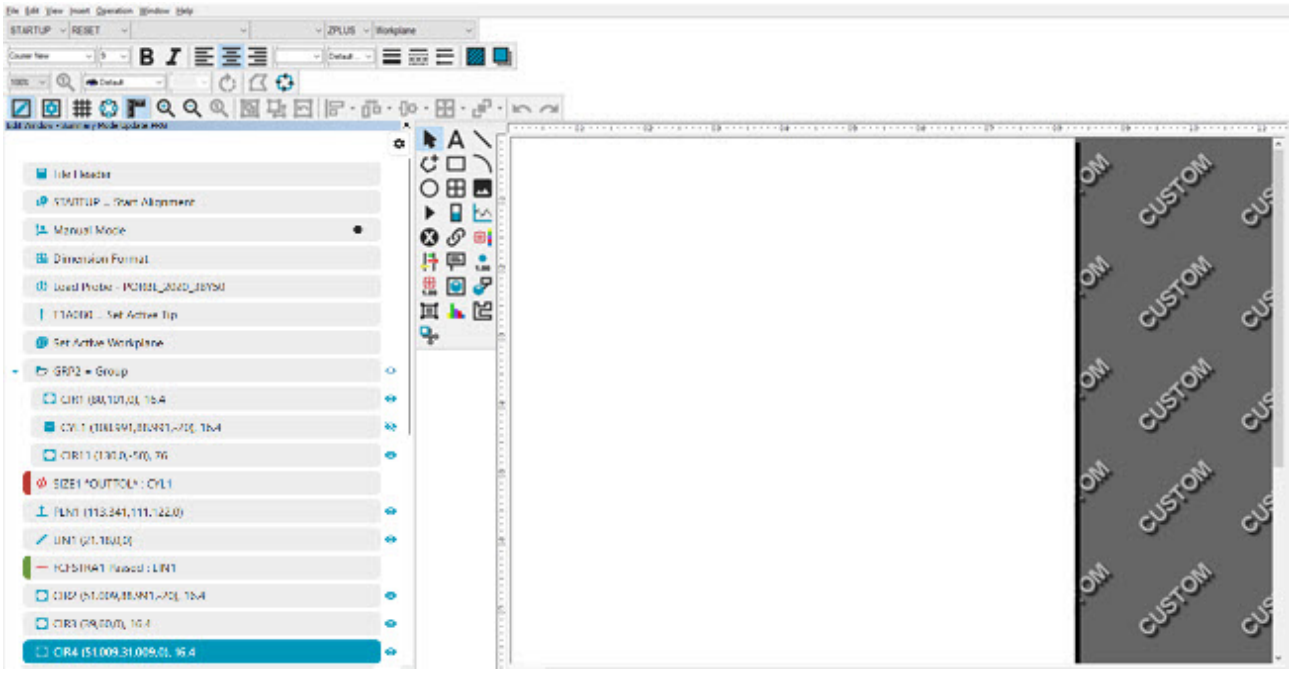
To access the Custom Report Editor, select **File | Reporting | New | Custom Report**. To open an existing custom report for your current measurement routine, select **File | Reporting | Edit | Custom Report**.

You can access a slimmed down version of this editor from the Report window's **Reporting** toolbar. This lets you edit a report without leaving the usual PC-DMIS menu structure.

You can access this editor through either of these options:

- Click **Custom Report selection dialog** ( ) and then click **Blank Report** and then **Open**.
- Click **Edit report** ( ).

PC-DMIS displays the Custom Report Editor with the Edit window placed into Summary Mode next to it.



Custom Report Editor with the Summary Mode to its left

This editor is similar to the Report Template Editor, Label Template Editor, and Form Editor. With the exception of some differences in its **Object Bar**, the user interface is essentially the same.

The custom report is the white and sizable area on top of the "CUSTOM" background. Note that when you use this editor, you do not create a template, but create the actual report that appears in the Report window. When you create a new custom report, this editor opens side by side with the Edit window set in Summary Mode. This allows you to drag the desired objects from Summary Mode onto your Custom Report. See "Creating Custom Reports".

With the Custom Report Editor, you cannot place the Report or Label template editors into Run mode as you did with the HyperView editor. The Custom Report Editor only opens in Edit mode and cannot be switched to Run mode. Run mode functions only with the HyperView editor or the Form Editor.

The editor contains the following elements:

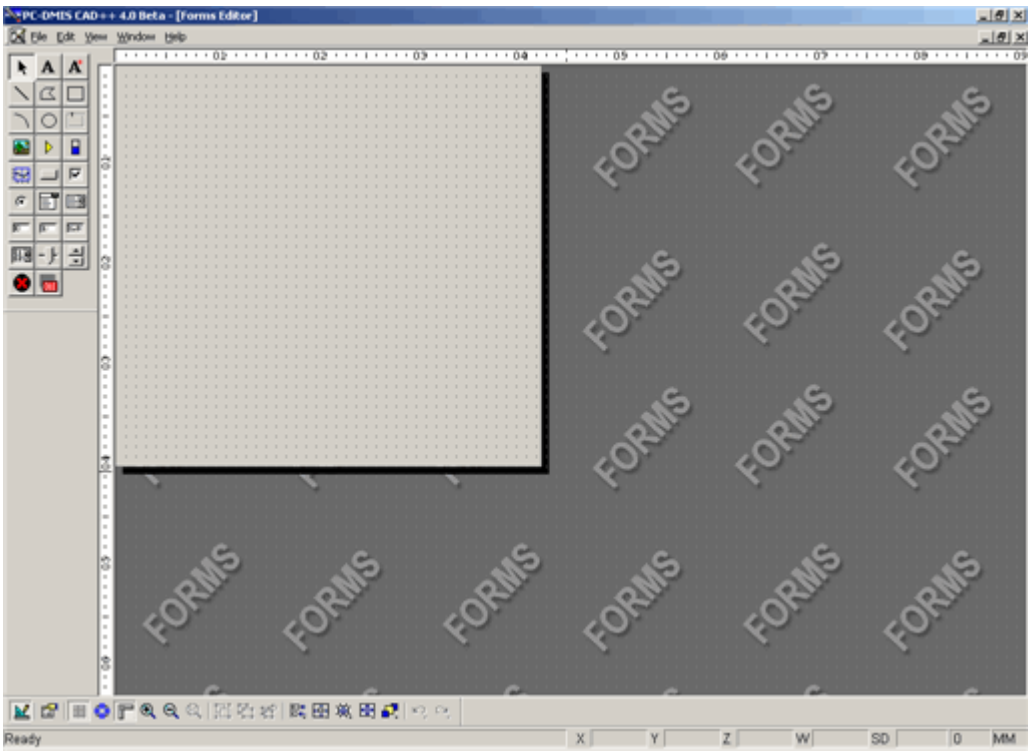
- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)
- The Layout Bar (toolbar)
- Object Property Sheets

Understanding the Form Editor

To create a new form in the Form Editor, select **File | Reporting | New | Form Report**.

To open an existing form in the Form Editor, select **File | Reporting | Edit | Form Report**.

PC-DMIS displays the Form Editor in a new window.



Form Editor

The Form Editor acts as a work area. It enables you to drag, size, and place various interactive form objects, such as buttons, list boxes, edit boxes, and so forth, and set their properties.

Objects are placed on the gray and sizable area, called a form. If you have worked with HyperView reporting in previous versions of PC-DMIS, you will feel at home with this editor since it essentially functions much the same way and contains many of the same user interface elements.

With the new template approach to reporting, only the Form Editor can use the run-mode capability. The Report and Label template editors do not use it.

> **Run Mode** - This mode only works with the Form Editor. In run mode, you can run (or execute) your form. This enables you to test how it would look or behave in an actual measurement routine execution environment.
>
> **Edit Mode** - This is the default mode for all the reporting editors. It is the only mode for the Report and Label template editors.
>
> To switch between run and edit modes, press CTRL+E.

The editor contains the following elements:

- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)
- The Layout Bar (toolbar)

- Object Property Sheets
- Object Sheet

Creating Templates

While PC-DMIS comes with some pre-designed templates that fit most users' needs, you can delve into the powerful template editors to create your own report and label templates. You can also use the Form Editor to create a more interactive approach to execution and reporting.

There are several report templates (.rtp) and label templates (.lbl) that ship with PC-DMIS.

File location: C:\Users\Public\Public Documents\Hexagon\PC-DMIS\2026.2\Reporting

For more information, see "Using Standard Reports".

The following topics discuss how to use the report template and label template editors to create your own templates.

Are you new to template reporting? Follow the "Creating a Report Template" and "Creating Label Templates" tutorials.

In addition to creating your own templates, you can download custom templates from: Hexagon Support. If you need further support for custom templates, contact your regional office or headquarters.

About Reports and Report Templates

A report template is not a report, but a description for a report. The template describes what data PC-DMIS should use to create a report, where it will go, and what it will look like. You can use report templates for more than one measurement routine to standardize the look and feel of several reports.

You create templates inside of the Report Template Editor. Report template files have an .rtp filename extension and are created inside the PC-DMIS report template editor.

You can make report template files as simple or as complex as you like. A simple report template might contain a single **TextReportObject**, while a complex report template might contain several different objects, bitmap images, shapes, or even items that use the reporting expression language and scripting to detail exactly what appears from the report data.

Do not confuse the report template's filename extension, **.rtp** with the older HyperView report filename extension, **.rpt**. These are completely different file formats.

Report Template Selection Precedence

PC-DMIS selects the report template based on this order of precedence:

1. The default report template assigned to the current measurement routine. You can define a default report template for each measurement routine by clicking the **Set this report as default template** button in the **Reporting** toolbar. For details on this button or the **Reporting** toolbar, see the "Reporting Toolbar" topic.
2. The report template specified as the default report template in the Settings Editor, if one exists. For details, see the "DefaultReportTemplate" topic in the Settings Editor documentation.
3. The TEXTONLY.RTP file.
4. The default.rtp file.

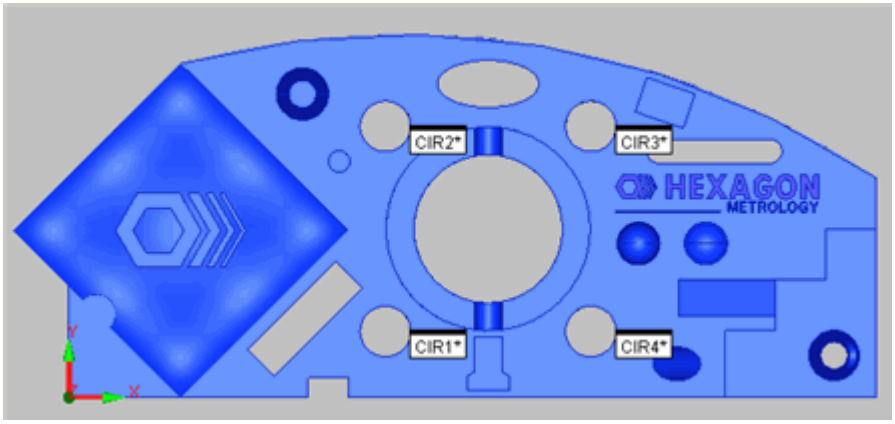
For information on the TEXTONLY.rtp and default.rtp template files, see the "Using Standard Reports" and "Viewing a Legacy Text Report" topics.

Tutorial - Creating a Report Template

This topic walks you through a very basic tutorial that creates a simple report template with a few objects and some labels. This should give you a basic overview of how things work together so that you can create and use your own report templates.

In order to give you a good foundation on how to use the template editors, this tutorial shows how to create a report template from scratch. However, in practical use, you will most likely find it much easier to customize one of the standard templates to suit your needs.

Before you begin this tutorial, create a simple measurement routine that can measure four circles on a simple part. This tutorial uses the Hexagon test block (Hexblock_Wireframe_Surface.igs).



Example of four circles to be measured.

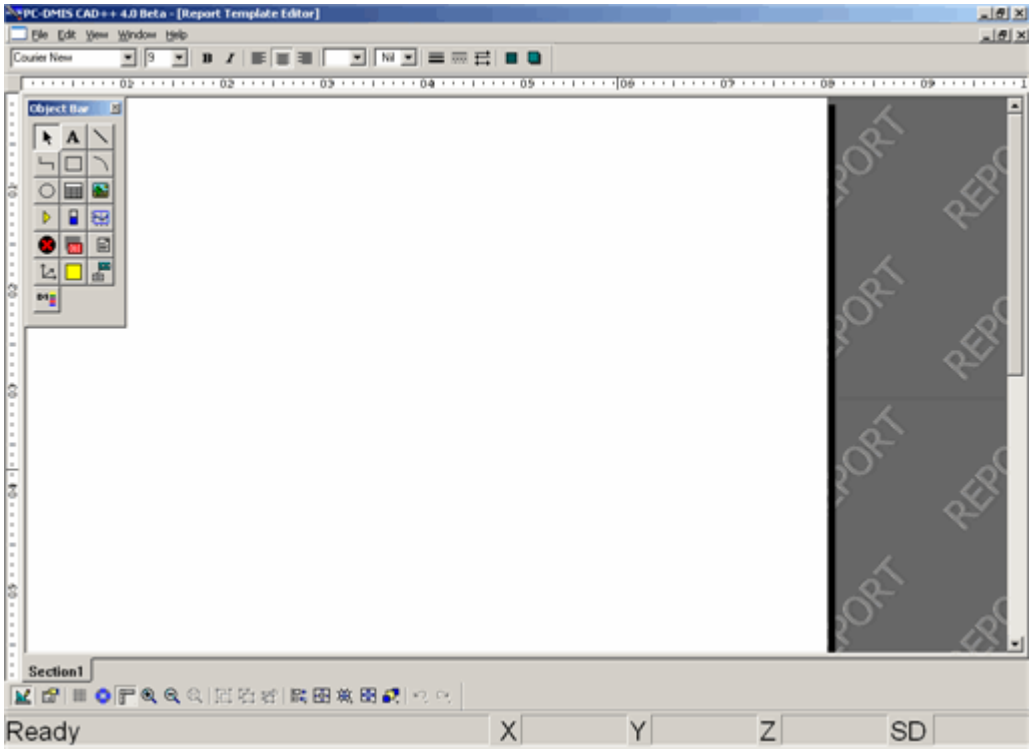
Step 1: Create a Blank Report Template

1. Select **File | Reporting | New | Report Template** to access the Report Template Editor. A blank report template automatically appears.

2. To hide any unused toolbars, right-click on the toolbar area and remove them.

3. To hide any unused PC-DMIS windows, select the open window from the **View** menu.

4. Click the maximize button on the editor's window to maximize the template editor. You should see the words "REPORT" on the background of the editor, and your editor should have a "Section1" tab at the bottom. Your Report Template Editor should now look something like this:



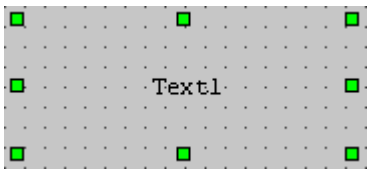
Report Template Editor

When working with one of the template editors, you may find it helpful to hide your usual PC-DMIS toolbars and windows, thereby freeing up some screen space. If you work frequently with templates, you may want to create a stored screen layout for your template. For information on layouts, see the "Window Layouts Toolbar" topic in the "Using Toolbars" chapter.

In addition, you may find it useful to create your own toolbar with the **File | Reporting** commands on it. For information on how to create custom toolbars, see the "Customizing Toolbars" topic in the "Customizing the User Interface" chapter.

Step 2: Insert and Format a Text Object for a Section Title

1. From the **Object Bar**, click on a **Text** object.
2. Insert it into the report by dragging a rectangle onto the current section. When you release the mouse, the object is selected, as shown by small green squares, called handles, at each corner of the object.



A sample text object with handles.

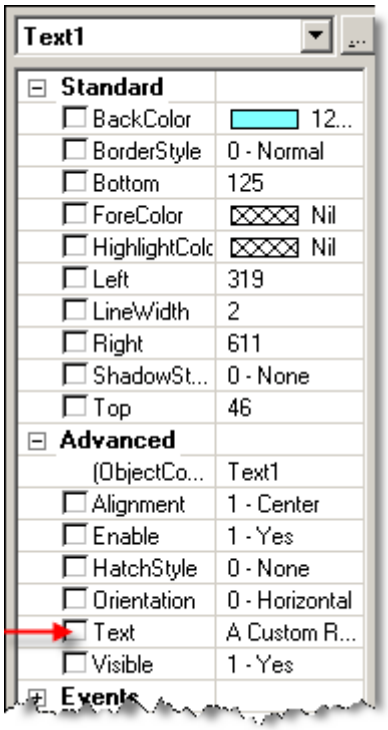
3. Select the **Text** object you inserted, and drag the green handles so that it has a height of 1 inch and a length of 6 inches. Use the **Ruler Bars** on the top and left border of the editor as needed. Select **View | Ruler Bars** if the rulers aren't available.
4. Right-click on the **Text** object.
5. Select the **BackColor** property and change it to a light blue color (0.255.255).
6. Click the **ForeColor** property and change it to a dark blue color (0.0.128).
7. Select the **BorderStyle** property and change it to **Normal**.
8. Click the **LineWidth** property and change the value to 5.
9. Click the **Text** property and change it to "A Custom Report".
10. Click the **Font** property. The **Font** dialog box appears.
11. Change the **Size** to 20, the **Font Style** to Bold, and the **Font** to Arial.
12. Click **OK**.

You have formatted your text object. It should look similar to this:

Step 3: Make Properties User Assignable

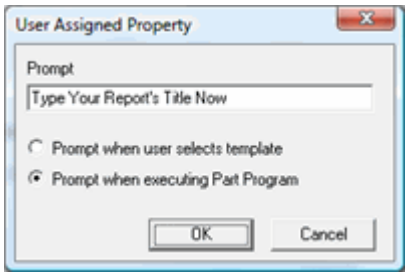
1. Select the **Text** object you created in the previous step.

2. Right-click on the object. This causes the **Properties** dialog box for that object to appear on the right side of the editor. On the left side of the dialog box you will see check boxes next to most properties. These check boxes determine what properties you can make user definable.



A sample property sheet for a text object with user-defined check boxes on the left.

3. Select the check box to the left of the **Text** property. A **User Assigned Property** dialog box opens, asking you to determine the message that will prompt the user for the property value and when the user should be prompted.



User Assigned Property Dialog Box

4. In the **Prompt** box, type "Type Your Report's Title Now".

5. Select the **Prompt when executing Measurement Routine** option.

6. Click **OK**. The **User Assigned Property** dialog box closes.

7. Select the check box to the left of the **BackColor** property. In the resulting dialog box, type "Choose Your Background Color Now" for the prompt, and once more, choose the **Prompt when executing Measurement Routine** option.

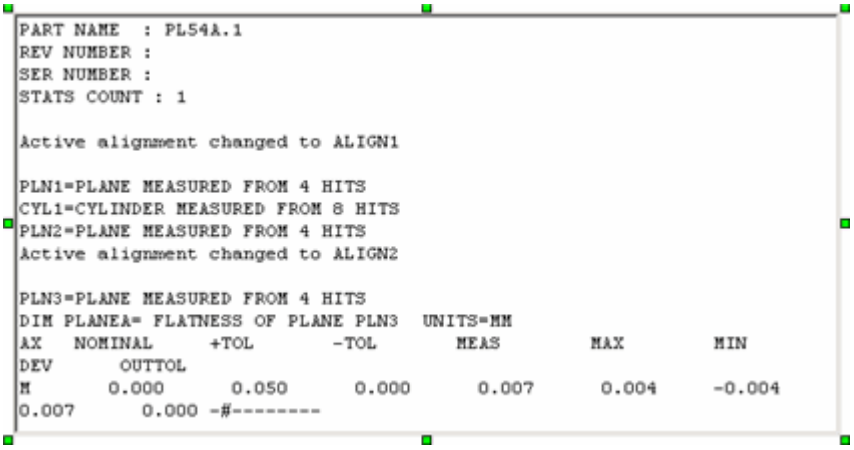
8. Click **OK**. The **User Assigned Property** dialog box closes.

In this step, you selected two properties and made them "user assignable". This means whoever executes your measurement routine will be able to set these properties.

Step 4: Add a TextReportObject

1. From the editor's **Object Bar**, click on the **TextReportObject** *icon* . Your mouse pointer changes from an arrow pointer to a cross.

2. Add the **TextReportObject** on the template by dragging a box on your template's section. PC-DMIS automatically adds some default filler report text. When you apply this template to your report, PC-DMIS uses your actual report data. This filler text simply helps you understand what your design will look like. Your object should look something like this:



A sample TextReportObject.

3. Size the object so that it's close to the width of the **Text** object you added earlier. To size the object, click and drag one of the green boxes (or "handles") that surround the object.

4. Select the **Text** object you added earlier. Hold down the Shift key and select the **TextReportObject**. This selects both objects.

5. With both selected, use the **Layout Bar**, and click the appropriate **Make Same Size** and **Align Center** icons to make the objects the same width and to align them so that they are centered in between the left and right sides of the editor.

Your template should now have a **TextReportObject** on the first section.

Step 5: Add a New Section to the Template

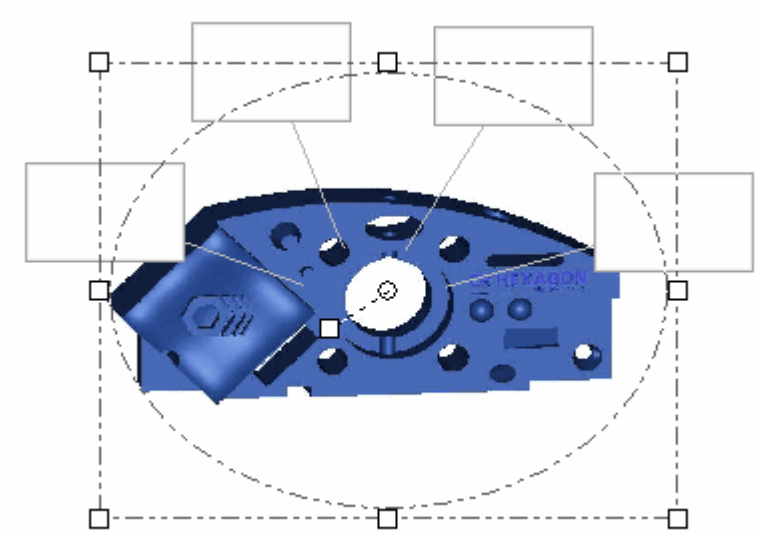
1. Maximize your Report Template Editor window.

2. Scroll to the very bottom of your template.
3. Right-click on the **Section1** tab.
4. Select **Add Tab** from the shortcut menu. PC-DMIS inserts a new section, **Section2**, into your template.

Your template should now have two sections, like this:

Step 6: Add a CADReportObject

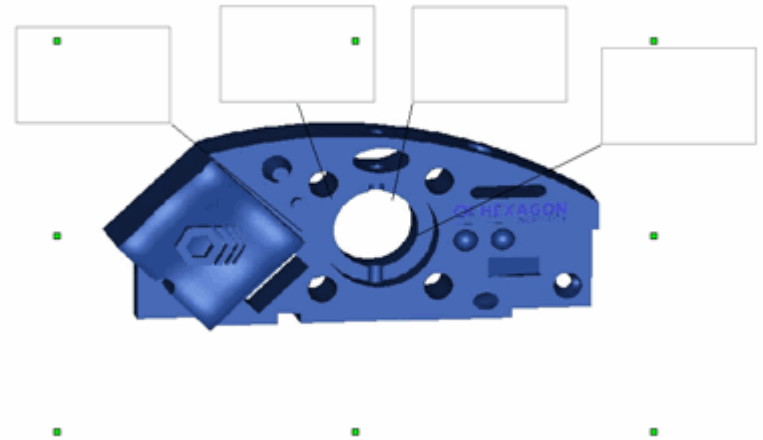
1. Click on the **Section2** tab to make that the active section.
2. From the editor's **Object Bar**, click the **CADReportObject** *icon* . Your mouse pointer changes from an arrow to a cross.
3. Drag a box on your template's section to add the **CADReportObject** on the template. Size the box so that it measures about six inches wide and four inches tall. Click and drag one of the green boxes surrounding the object to resize it.
4. When you finish drawing the box, PC-DMIS automatically displays the **Label Layout Wizard**. This wizard shows a fake part surrounded by some fake labels. These dummy items are simply design aids while working in the template editor. You can use this wizard to set up labels around your part. For in-depth information, see the "The Label Layout Wizard" topic. When you apply this template to your report, PC-DMIS uses your actual CAD drawing and label information.
5. Change the **Label Count** box number of labels to 4.
6. Change the **Layout Style** to **Elliptical Packed**.
7. Click on the small, square, white handle in the middle of the CAD drawing, and drag the square clockwise or counterclockwise to rotate the labels along the elliptical path.
8. Rotate the labels until all four are above the CAD drawing and the **Layout Preview** area of the **Label Layout Wizard** looks like this:



Layout Preview area showing four labels elliptically packed above the CAD drawing.

9. Click **OK**. PC-DMIS inserts the object into **Section2**.

Your **Section2** layout should now contain a CADReportObject looking something like this:



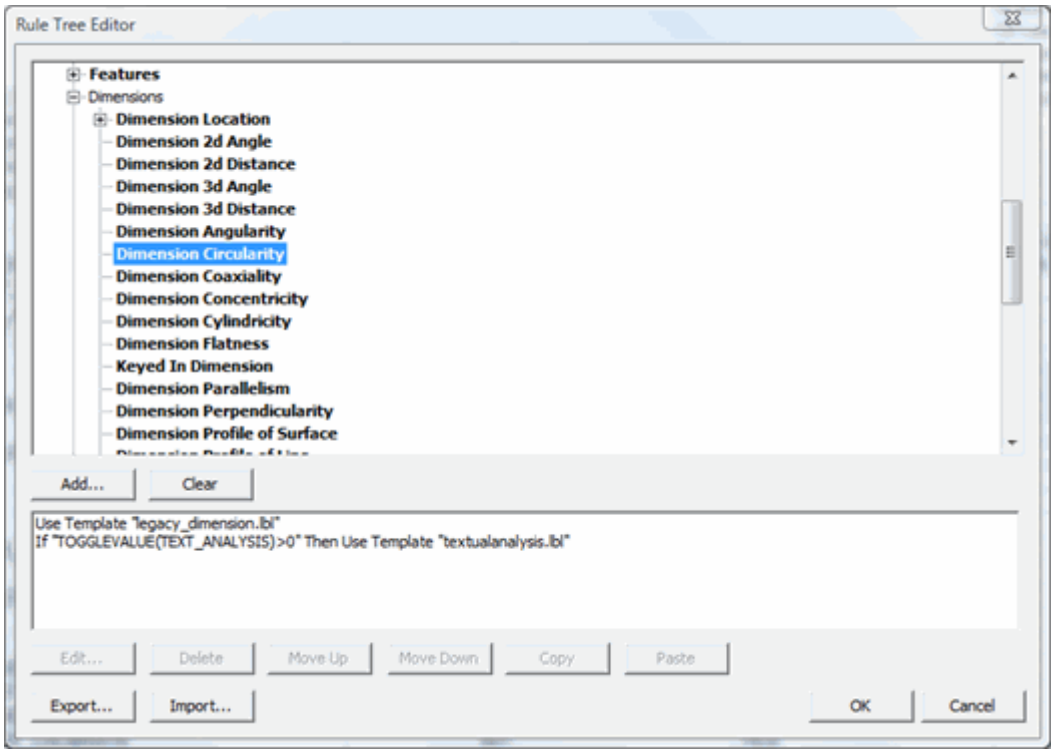
Example of a CAD report object showing the four dummy labels above the dummy part.

Step 7: Define What Measurement Information Should Appear

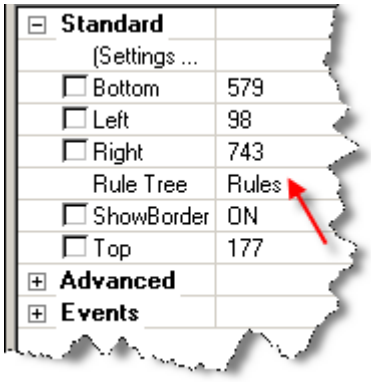
1. Click on the **Section1** tab and right-click on the **TextReportObject** you inserted earlier.
2. From the **Properties** dialog box, change these properties:
    - ShowAlignments - OFF
    - ShowComments - OFF
    - ShowDimensions - None
    - ShowFeatures - ON
    - ShowHeaderFooter - OFF

- ShowMoves - OFF
- ShowScreenCaptures - OFF

3. Click on the **Section2** tab.
4. Right-click on the **CADReportObject** you added into **Section2**. The right part of the screen displays a docked **Properties Sheet** dialog box. This dialog box contains various properties specific to the selected object.
5. Click on **Rules** in the **Rules Tree** property. The **Rule Tree Editor** appears:



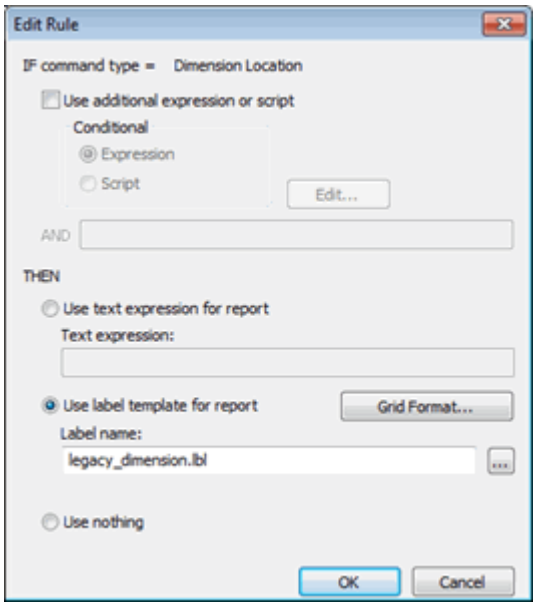
Rule Tree Editor Dialog Box



Rules Tree Property Option

6. Expand the **Features** heading. A list of features appears.
7. Expand the **Circles** heading. A list of various circle features appears.

8. Click the **Measured Circle** item, and then click the **Add** button. This tells PC-DMIS you are adding a rule that you want the template to follow. The **Edit Rule** dialog box appears. This dialog box allows you to make decisions for your report template. In this case, you are telling your report template to do something when it encounters measured circles in the measurement routine's report data.



Edit Rule Dialog Box

9. Select the **Use Label Template for Report** option. The **Label Name** box becomes available for edit.

10. Click the **...** button to open the **Open** dialog box that allows you to select a label template file.

11. Navigate to where you installed PC-DMIS, and open the Reporting sub-directory. Select the *summary.lbl* label, and click **Open**. In the **Edit Rule** dialog box, PC-DMIS displays the name of the template you selected. See "About Labels and Label Templates" for information and a tutorial on creating labels.

12. Click **OK** to close the **Edit Rule** dialog box and apply the rule. You will notice that the **Measured Circle** item in the list has a bold font to indicate that a rule exists for that feature type.

13. Click **OK** to close the **Rule Tree Editor** dialog box.

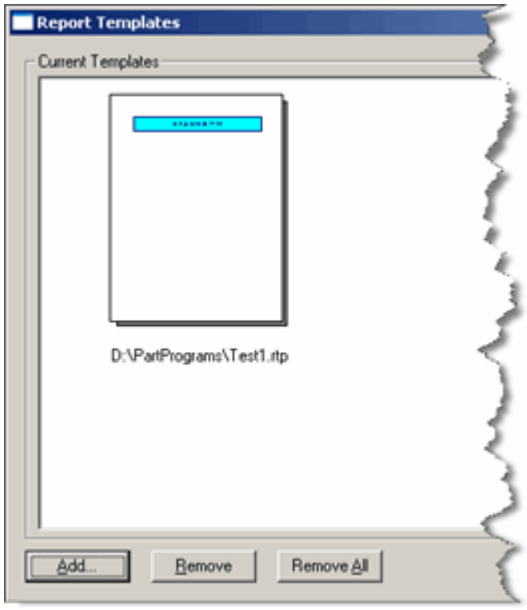
You have created a rule that tells PC-DMIS to use the *summary.lbl* label template to display summary information about the measured circle feature type.

Step 8: Save and Test the Template

1. Inside the Report Template Editor, select **File | Save**. In the **Save As** dialog box, save the template as **test1.rtp**.

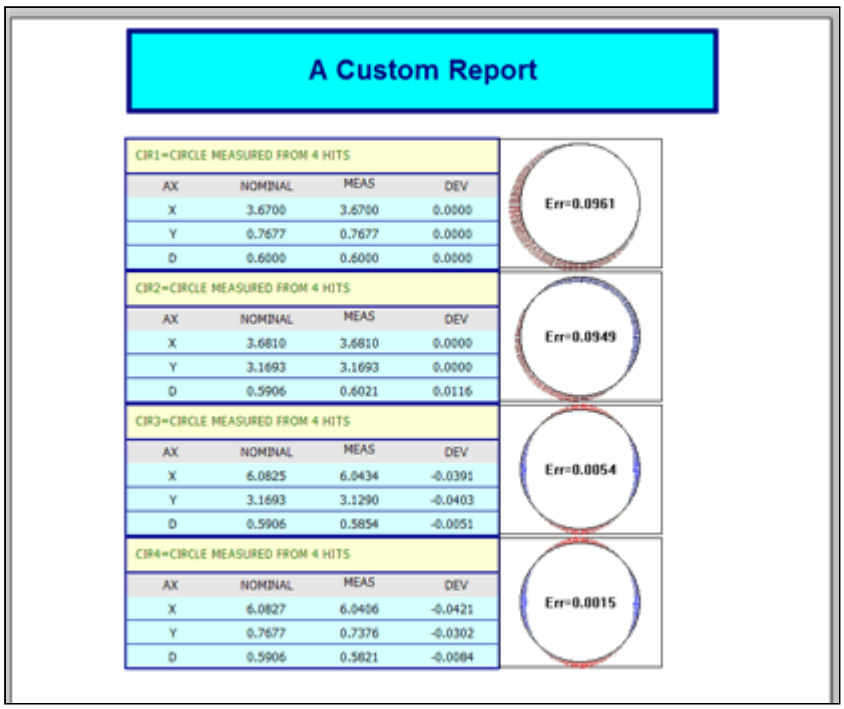
2. Select **File | Close** to close the Report Template Editor.

3. Select **View | Report Window**. The Report window opens, showing the default report template.
4. From the **Reporting** toolbar on the Report window, click the **Template Selection Dialog** *icon* . The **Report Templates** dialog box opens.
5. Click the **Add** button to add your report to the **Report Templates** dialog box.
6. Navigate to and select test1.rtp, then click **Open**. PC-DMIS adds a thumbnail view of your report into the **Report Templates** dialog box.



Report Templates dialog box showing Test1.rtp

7. Select your template's thumbnail icon, and click **Open**. PC-DMIS opens the Report window using your newly created template. It should look something like this:



Report Window Showing Report Data Using the Test1.rtp Report Template

8. Finally, execute your measurement routine. The first time you execute the measurement routine using this report template, PC-DMIS displays a **Define User Assigned Properties** dialog box, asking you to give the report a new title and to set the background color.

9. Set these properties as desired, and click **OK**. The dialog box closes and PC-DMIS runs your measurement routine.

10. When execution finishes, PC-DMIS shows the report data with the new template.

Congratulations! You have finished the Custom Report Template tutorial.

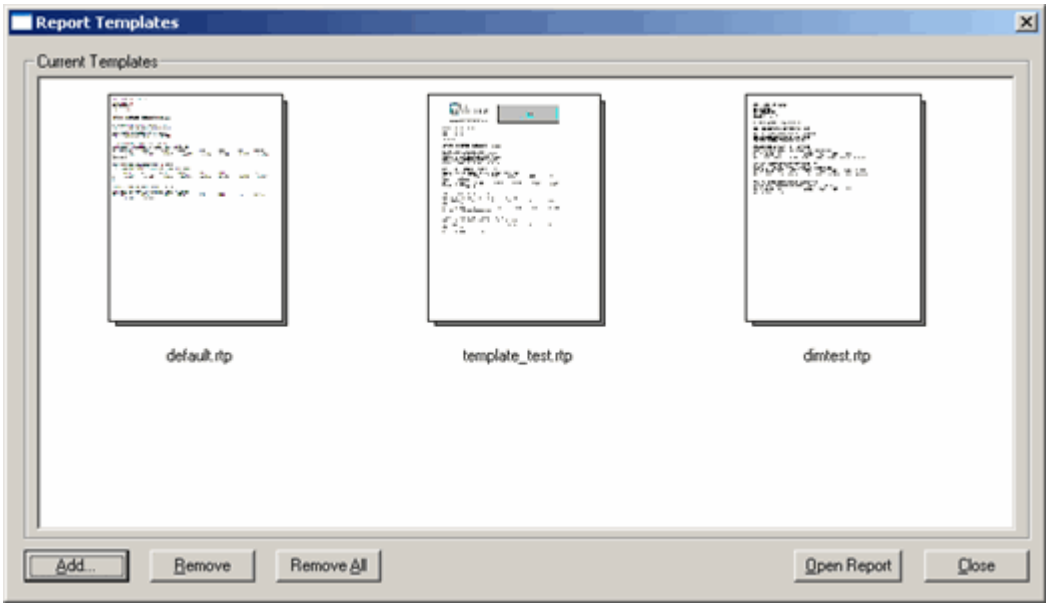
Applying or Removing a Report Template

To apply a report template to your measurement results, or to remove a report template, you must first access the **Report Templates** dialog box:

To Access the Report Templates dialog box:

1. Select **View | Report Window** to access the Report window.

2. From the **Report** toolbar on the Report window, click the **Template Selection** icon . The **Report Templates** dialog box appears showing available report templates as .rtp files:



Report Templates Dialog Box

You can use this dialog box to manage your report templates.

To Add a Report Template:

1. From the **Report Templates** dialog box, click the **Add** button. A standard **Open** dialog box appears.
2. Navigate to the report template file, select it, and click **Open**.
3. PC-DMIS adds the template, and a thumbnail picture of the template will appear in the **Report Templates** dialog box.
4. Your dialog box might contain few or many templates. To resize the dialog box to better suit your needs, drag the dialog box's edge to a new location.

To Remove a Report Template:

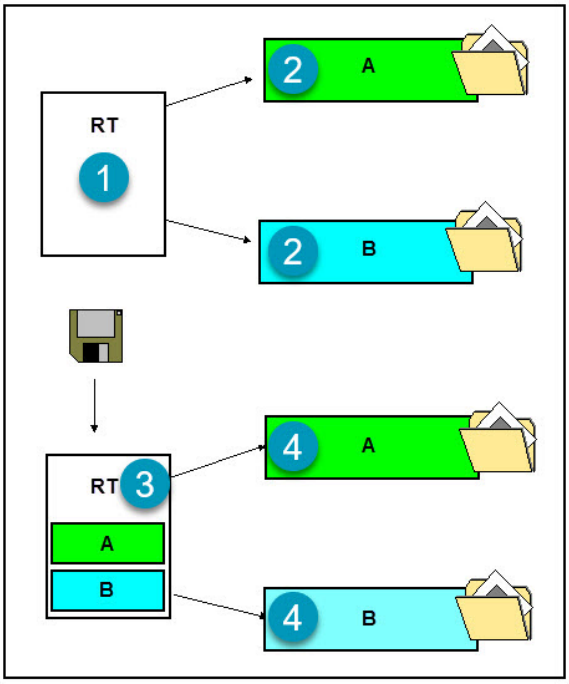
1. From the **Report Templates** dialog box, select a template.
2. Click the **Remove** button (to remove all templates, click **Remove All**).
3. PC-DMIS removes the report template from the **Report Templates** dialog box.

Removing report templates doesn't delete them. It merely removes them from this dialog box. You can always add them later using the **Add** button.

The templates listed in the **Report Templates** dialog box are different for each user account on the Windows operating system.

Sharing Report Templates

You can easily share a report template with other users. When you save a report template, PC-DMIS automatically saves a copy of any associated label templates into the report template itself. This way, if you want to share a report template, you don't have to worry about sending all the associated label templates along with it. Even so, the report template will always check for the actual label template file in the directory first. If it finds the actual label template file, it will use that. If the actual label template file doesn't exist as expected, then the report template will use the copied version of the label template file that is stored with the report template itself.



1 - In this example, the report template calls two label templates, label A and label B.

2 - The label template files (files with a .lbl filename extension) are stored in the Reporting directory (C:\Users\Public\Documents\Hexagon\PC-DMIS\\Reporting).

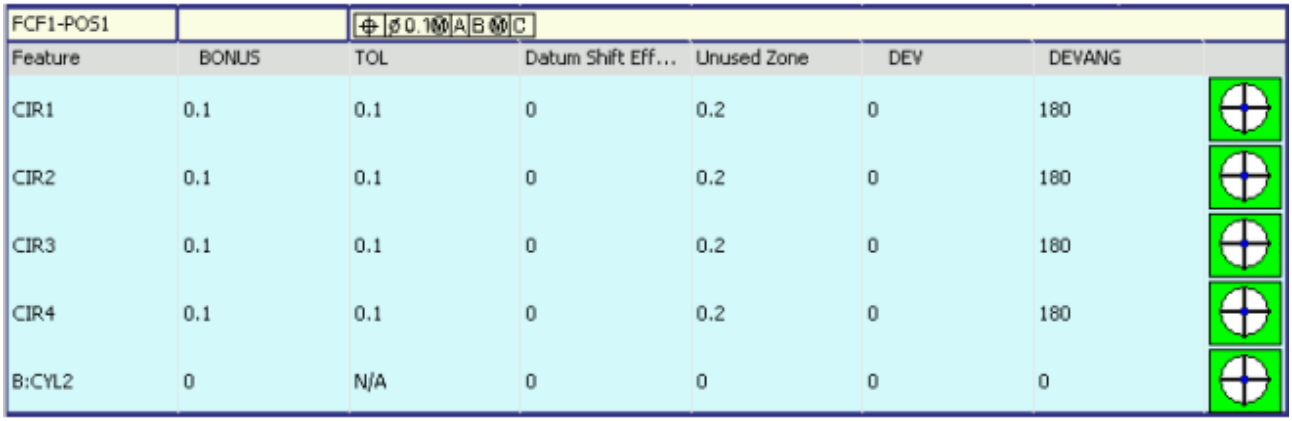
3 - When you save the report template, copies of the label templates are stored within the report template.

4 - When used, the Report template first looks for any label template files and uses those if found. If none is found, it then uses the copied label templates.

About Labels and Label Templates

A label template essentially acts like a mini-report template that you associate with a specific command (or commands) in the report. This gives you a lot of freedom with data

that you can include in your report. For example you can display labels for dimensions, labels for measured features, for Auto features, and so forth.



FCF1-POS1		⌖ ⌀0.1Ⓜ A B Ⓜ C				
Feature	BONUS	TOL	Datum Shift Eff...	Unused Zone	DEV	DEVANG
CIR1	0.1	0.1	0	0.2	0	180
CIR2	0.1	0.1	0	0.2	0	180
CIR3	0.1	0.1	0	0.2	0	180
CIR4	0.1	0.1	0	0.2	0	180
B:CYL2	0	N/A	0	0	0	0

Example of a label showing a position table with a Feature Control Frame (FCF).

PC-DMIS ships with several standard labels that you can associate with objects in your Report templates. However, as with Report templates, you can create your own label templates as well. Label templates can contain any object from the Label Template Editor's **Object Bar**, but they usually contain the **GridControlObject** or the **Graph** object.

You can use the Label Template Editor to create label templates.

Tutorial - Creating Label Templates

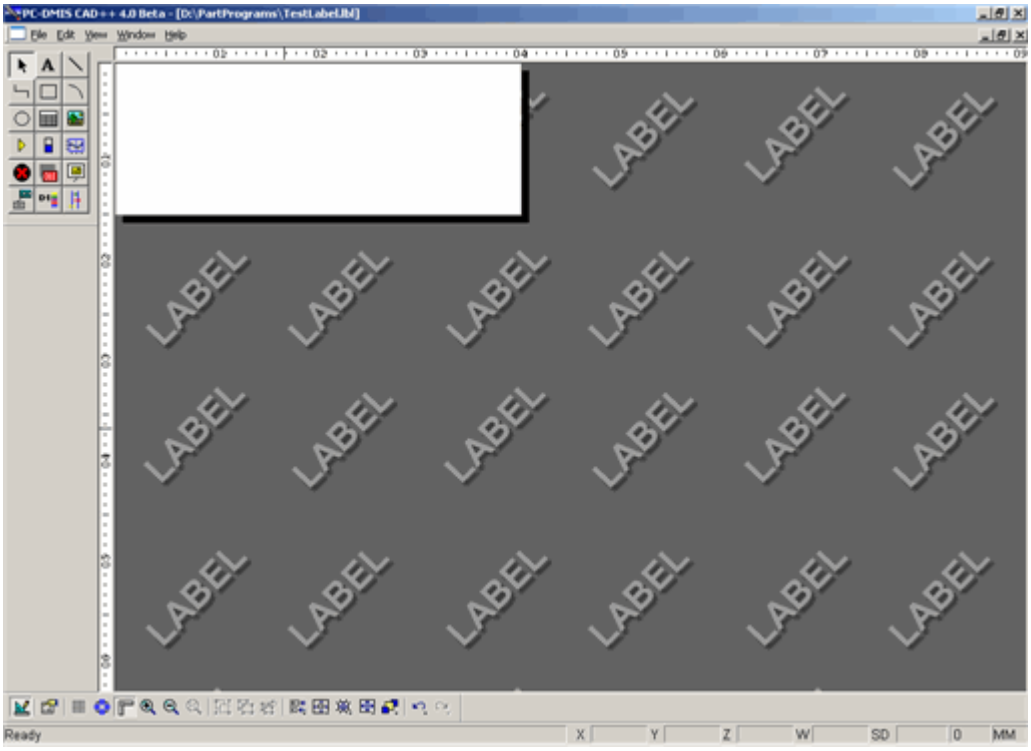
This topic walks you through a basic tutorial to create a simple label template with a few objects and a **GridControlObject**. This provides you with a basic overview of how label templates work and how to use them to display custom information in your reports.

> This tutorial builds on skills you learned in the "Creating a Custom Report Template" tutorial; be sure to complete that section first.

Step 1: Create a Blank Label Template

1. Select **File | Reporting | New | Label Template** to access the Label Template Editor. A blank label template automatically appears.
2. To hide any unused toolbars, right-click on the toolbar area and remove them.
3. To hide any unused PC-DMIS windows, select the open window from the **View** menu.
4. To maximize the template editor, click the Edit window's *maximize button* . You should see the words "LABEL" on the background of the editor.
5. Size **The Frame/The View** in the label template editing area to four inches wide and one and a half inches high. To do this, click and drag the bottom right hand corner of the label template with your mouse. Use the rulers as a guide.

You have created a blank label template. When finished with this step, it should look something like this:



Label Template Editor

When working with one of the template editors, you may find it helpful to hide your usual PC-DMIS toolbars and windows, thereby freeing up some screen space. If you work frequently with templates, you may want to create a stored screen layout for your template. For information on layouts, see the "Window Layouts Toolbar" topic in the "Using Toolbars" chapter.

Step 2: Add a CommandTextObject and Static Information into the Template

1. Click the **CommandTextObject** *icon* . The pointer changes to a cross-hair.
2. With your mouse pointer, drag a box so that it stretches about three inches wide by one half inches high. PC-DMIS inserts a CommandTextObject into your label template. Right-click on it to access the **Properties** dialog box.
3. Click the **Colors** property. If PC-DMIS asks if you want to define a color set independent from the default application colors, click **Yes**.
4. The **Color Editor** opens. PC-DMIS uses this editor to define Edit window colors. However, in this case, it only affects the current CommandTextObject.
5. Under **Background Color**, click **Edit**. A standard **Color** dialog box opens.
6. Give the object's background color a dark blue color. To do this, change the values in the **Red**, **Green**, **Blue** boxes to 0, 0, 128 respectively. Click **OK**.

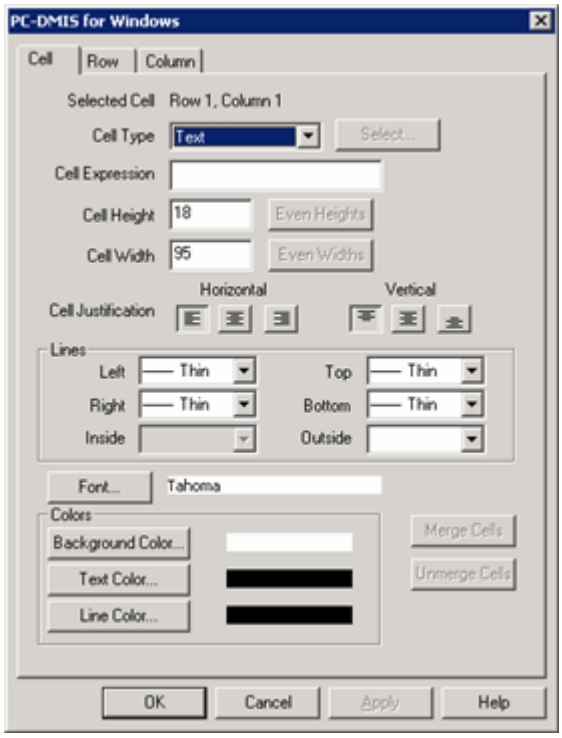
7. In the **Unmarked** box, click **Edit**. A standard **Color** dialog box opens.
8. Select white, and then click **OK**. If PC-DMIS asks if you want to change the item's children, click **No**.
9. Use a bitmap image of a circle (or use the Ellipse object to draw a circle) and place it in the top right of the canvas.

You have added the **CommandTextObject** and a static **Bitmap** or **Ellipse** object for a circle shape into the label template. At the end of this step, your label template should looks something like this:

Label Template showing the CommandTextObject and a static Bitmap object

Step 3: Insert and Format a GridControlObject

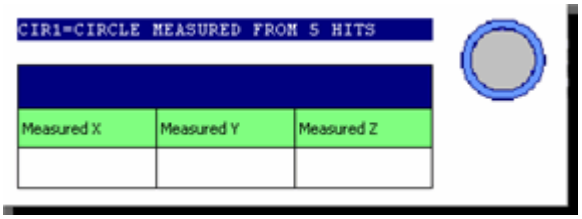
1. Click the **GridControlObject** *icon* . The pointer changes to a cross-hair.
2. With the pointer, drag a box for this object under the previously inserted CommandTextObject, so that it has the same width.
3. Size the height so that it nearly fills the rest of the canvas. When inserted, the grid shows several rows and columns.
4. Access the object's properties, and then change the **Columns** and **Rows** properties to 3 each.
5. Select the object and select a cell. To do this, double-click on the first cell in the top row. It will highlight the cell in a blue background to show that you have selected it.
6. Select the entire top row. To do this, with the first cell selected, press the Shift key, and then click on the last cell in that row. PC-DMIS highlights the entire row.
7. Right-click the GridControlObject. The object's dialog box appears. This dialog box lets you control the formatting of the selected cells as well as insert text and expressions.



The GridControlObject's dialog box

8. Click the **Merge** button to merge the selected cells.
9. Set the background color for the merged cell. Click the **Background color** button to open the **Color** dialog box. Choose a dark blue color and click **OK**.
10. Set the text color for the merged cell. Click the **Text color** button, and select white in the same way.
11. To create headings, select the first cell in the second row, and right-click to bring up the dialog box. In the **Cell Expression** box, type "Measured X". Close the dialog box, and repeat the process on the second cell, typing "Measured Y". Finally, repeat it again for the last cell, typing "Measured Z".
12. Select the entire second row. Right-click and use the dialog box to select a light green background color.
13. Under **Cell Justification**, under **Vertical**, click the *middle button* to vertically center the text in their cells.
14. Click **OK** to close the dialog box and then deselect the object.

You have inserted a GridControlObject and learned how to change the number of rows and columns. You learned how to add text into a cell and how to format the grid by using this object's special dialog box. When finished with this step, your label template should look something like this:

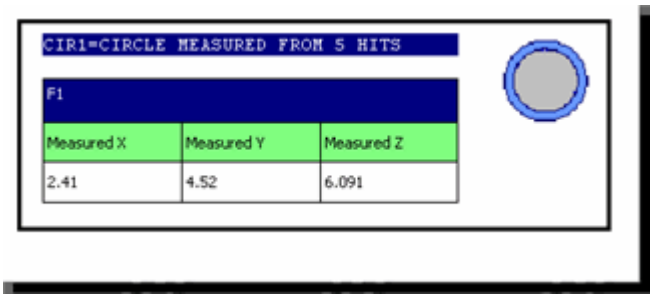


Label showing the inserted and formatted GridControlObject

Step 4: Add Expressions Into the GridControlObject

1. Select the GridControlObject's first row.
2. Once it's selected, type "=ID" directly into the cell. This is an expression that tells PC-DMIS you want to display the feature's ID in the cell.
3. Select the first cell in the third row and type "=MEAS_X"
4. Select the second cell and type "=MEAS_Y".
5. Select the third cell and type "=MEAS_Z". These expressions tell PC-DMIS you want to display the feature's X, Y, and Z values. When you deselect the object, PC-DMIS displays the expression values from the filler text used by the object. See "About Report Expressions" for information on expressions.
6. With the **Border** object, draw a border object around the label. You may need to resize the label to close to four and a half inches wide by two inches high.
7. Right-click the border, and give it a line width of 2.
8. Adjust the location of the contents so they are within the border.

You have added some report expressions into your label template and then drew a border around the template prior to testing it. When you finish this step your template should look something like this:



Label showing the results of the expressions

Step 5: Save and Test the Label Template

1. Select **File | Save** to save the label template. In the **Save As** dialog box, name the template "TestLabel.lbl".
2. Select **File | Close** to close down this template editor.
3. Select **File | Reporting | Edit | Report Template** and then select the **Test1.rtp** template file you created from the previous tutorial.

4. Select the TextReportObject you inserted into the first section of the report template and then access its properties.

5. Use the **Rules Tree Editor** and the **Edit Rule** dialog boxes to create a rule that will display TestLabel.lbl in that TextReportObject object for all measured circles. If you can't remember how to create rules, follow the procedure in the "Defining a Rule topic".

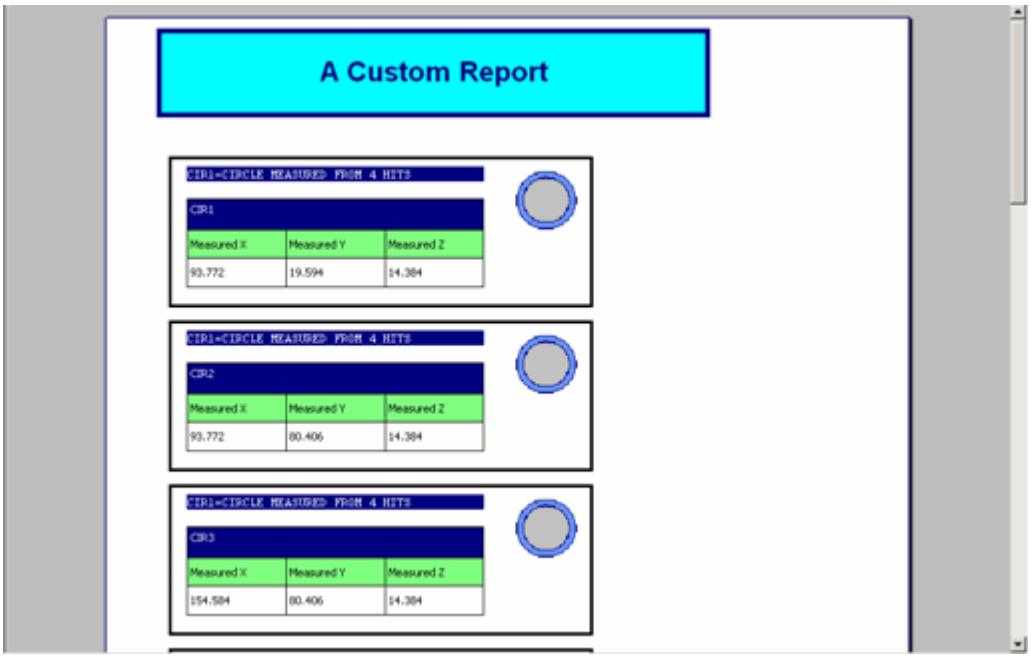
6. Select **File | Save** to save the report template.

7. Select **File | Close** to close it.

8. Execute the measurement routine and apply the template.

9. Select **View | Report window** to display the final report.

After you save and test your label template, it should look something like this:



Report window example showing the newly created label. Notice the measured X,Y, and Z center point data for each circle feature.

Step 6: Create a Repeating Row

So far, your label displays the XYZ of your circles' centers. Suppose, however, you wanted to get individual hit data instead. You can do this using repeating rows.

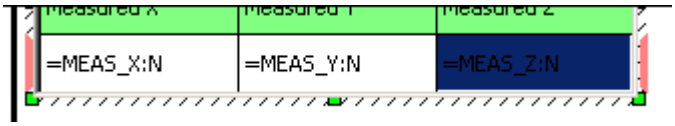
1. Select **File | Reporting | Edit | Label Template**. Select TestLabel.lbl from the dialog box and click **Open**. PC-DMIS loads the Label Template Editor and displays TestLabel.lbl.

2. Access the GridControlObject.

3. Select the first cell in the third row. Once you select it, right-click to bring up the GridControlObject's dialog box.

4. Click the **Row** tab.

5. Select the **Repeating Group** check box. This tells PC-DMIS that you want to make that row repeatable. The **Repeat Expression** dialog box becomes available. Notice that the row, at this point, contains orange markers on the left and right sides. This indicates that the row is a repeating row.



A row showing orange markers

6. Type "=N_HITS" in the **Repeat Expression** box. This tells PC-DMIS to get the total number of hits in the feature and to repeat the row for each available hit.

7. Click the **Cell** tab. You should see "=MEAS_X" in the **Cell Expression** box. Append ":N" to the expression so that it reads, "=MEAS_X:N". This tells PC-DMIS to repeat the expression in that cell for each available hit.

8. Select the other two cells in the row, and modify them so that they have the ":N" code added as well: "=MEAS_Y:N" and "=MEAS_Z:N".

9. Because the labels will expand to include all the data passed into the GridControlObject, when the row repeats, you do not need to adjust the label's size for this.

10. Click **OK** to close the GridControlObject's dialog box.

11. Save the label template and access the Report window. Click the **Redraw the Report** icon from the **Reporting toolbar** to see your latest changes. Notice that instead of displaying the center point data for the circles, PC-DMIS displays individual hits.

You have learned how to create a basic repeating row to count and display several items of data without needing to create different report templates with different row configurations.

Step 7: Use the Columns Tab to Define Columns

1. Access the TestLabel.lbl label template, and select the existing GridControlObject.

2. Select its first row and then right-click it to bring up the GridControlObject's dialog box.

3. Click the **Unmerge Cells** button. The single cell making up the first row now becomes three cells.

4. Delete the "=ID" expression text in the first cell. You don't really need this expression in your final label template because the CommandTextObject displays it automatically.

5. Use the **Background Color** and **Text Color** buttons to change the cell's background color to white and its text to black.

6. Delete the inside column lines in the first row. To do this, select the first cell, access the dialog box, in the **Cell** tab, in the **Lines** area, change the line value of the **Right** list from **Thin** to **None**. Repeat this for the middle cell as well.

7. Click **OK** to close the GridControlObject's dialog box. While your label template may appear to have its cells merged, in reality, column lines in those cells are merely hidden.

8. Select the left cell in the first row and access the dialog box.

9. Click the **Column** tab. In the **Column** list box, type "Measured X". Click **OK** to close the GridControlObject's dialog box.

10. Repeat this for the middle and right cells, typing "Measured Y" and "Measured Z" respectively. You will use these column headings to control each columns visibility and ordering later on.

11. Select the CommandTextObject you added to the template earlier, and drag it down so that it rests on top of the GridControlObject's first row.

12. Modify the location or size of the Ellipse or circle Bitmap object you created earlier so that it's centered along the right side of the GridControlObject.

13. Select and then drag the top line of the Border object down so that it's just above the GridControlObject.

14. Drag a box around all the objects in the label to select them all. Then drag those objects up to the top and to the left as far as they will go on the label's canvas.

15. Right-click to access the GridControlObject's properties.

16. Type "TestTableFormat" in the **Table Format** property, and press the Tab key.

17. Save and close your label template.

In this step, you learned how to unmerge cells, set cell line properties, insert column headings, and select and move multiple objects. Defining column headings will let you control the visibility and ordering of your columns on the fly. When you have finished this step, your label template should look something like this:

Step 8: Define a Grid Format Table

1. Open the report template, Test1.rtp, in the Report Template Editor.
2. Select the TextReportObject, and right-click to access the **Properties** dialog box.
3. In the **Rules Tree** property, click **Rules** to open the **Rule Tree Editor** dialog box.
4. Use the Rule Tree Editor and navigate to the rule you created earlier.
5. Select the rule and click **Edit** to open the **Edit Rule** dialog box.
6. Click the **Grid Format** button. The **Table Format Properties** dialog box opens.
7. In the **New Table Name** box, type "TestTableFormat", and click **Add Table**.
8. Under the **Columns** list in the edit box, type "Measured X", and then click **Add Column**.
9. Add columns for "Measured Y" and "Measured Z" in this same manner.
10. Click **OK** on the various dialog boxes until your return to the Report Template Editor.
11. Select **File | Save** to save the report template.
12. Select **File | Close** to close the Report Template Editor.

In this step, you tied the GridControlObject's columns to the TestTableFormat table in the **Table Format Properties** dialog box by adding columns with the exact same name and naming the table the same name you gave the **Table Format** property. Since you haven't changed the order or visibility at this point, the newly created table in the dialog box should look like this:

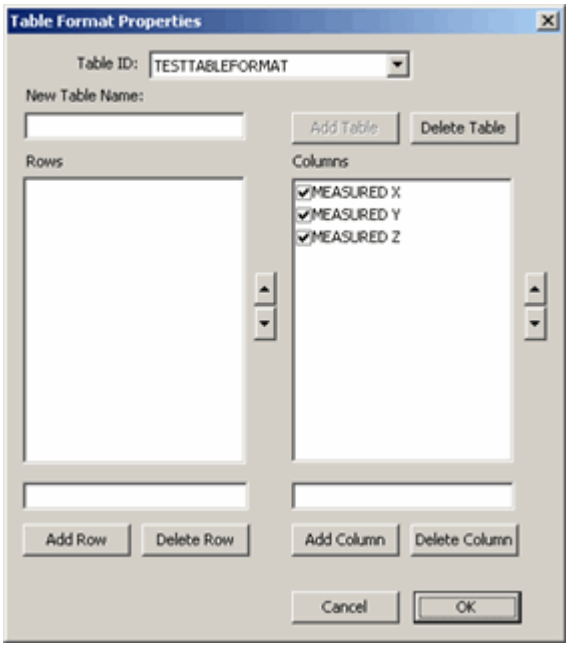


Table Format Properties dialog box showing the newly created TestTableFormat table

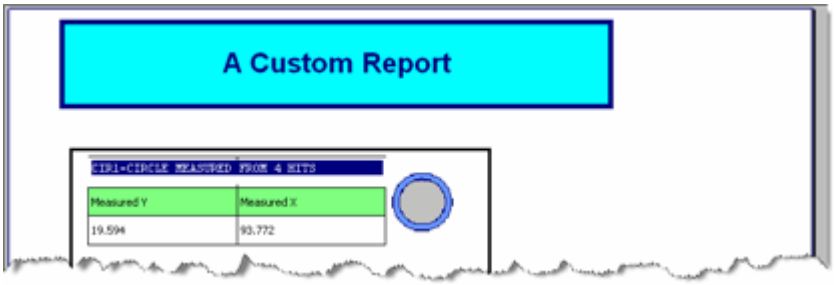
Step 9: Use Table/Format Command to Control Column Visibility and Order

1. If the Edit window is not visible, select **View | Edit Window** to display the Edit window.
2. Place the Edit window in Command Mode.
3. Select **Insert | Report Command | Table Format**. The **Table Format Properties** dialog box opens.
4. From the **Table ID** list, select the **TESTTABLEFORMAT** table. The **Columns** area shows the columns available to this custom table format.
5. Clear the **Measured Z** box to hide that column in the final report.
6. Select **Measured Y** and click the up arrow icon next to the **Columns** area to reorder the **Measured Y** column so that it gets displayed first in the report.
7. Click **OK**. PC-DMIS inserts a `TABLE/FORMAT` command into the Edit window. This command controls the visibility and order of your columns.

```
TABLE/FORMAT,TESTTABLEFORMAT
COLUMN/ORDER,MEASURED Y,MEASURED X, ,
ROW/ORDER
```

8. Save and execute your measurement routine. When PC-DMIS runs, notice that the order of the Measured X and Measured Y values have changed, and the Z column is hidden

In this final step, you learned how to insert and use a `TABLE/FORMAT` command to define the order and visibility of the columns in the label template. Your finished report should look something like this:



Finished report showing only the Measured Y and X columns

Using Labels in Reports

To use labels with your reports, you need to open a report template and add at least one of these objects:

- Label
- TextReportObject
- CadReportObject

Once an object exists, right-click on it to access its **Properties** dialog box. Finally, use the **Rules Tree Editor** to define rules that use one or more label templates.

When you apply your report template, PC-DMIS follows the rules you specified and uses the chosen label templates.

Understanding Stand-Alone Label Ordering

If you use stand-alone Label objects in your report template, be aware that, by default, PC-DMIS fills them with measured data in the report in front-to-back order. To illustrate this, when you add stand-alone labels, PC-DMIS defines each label by a progressive alphanumeric ID (Label1, Label2 …LabelN where N indicates the final label number). While label IDs don't control the order in which the labels are filled, they are discussed here to more easily illustrate how ordering takes place. During report creation, assuming you haven't modified the label IDs in any way, LabelN is activated and filled first, while Label1 is activated and filled last.

This only happens with stand-alone labels, not with labels used within the TextReportObject or CadReportObject.

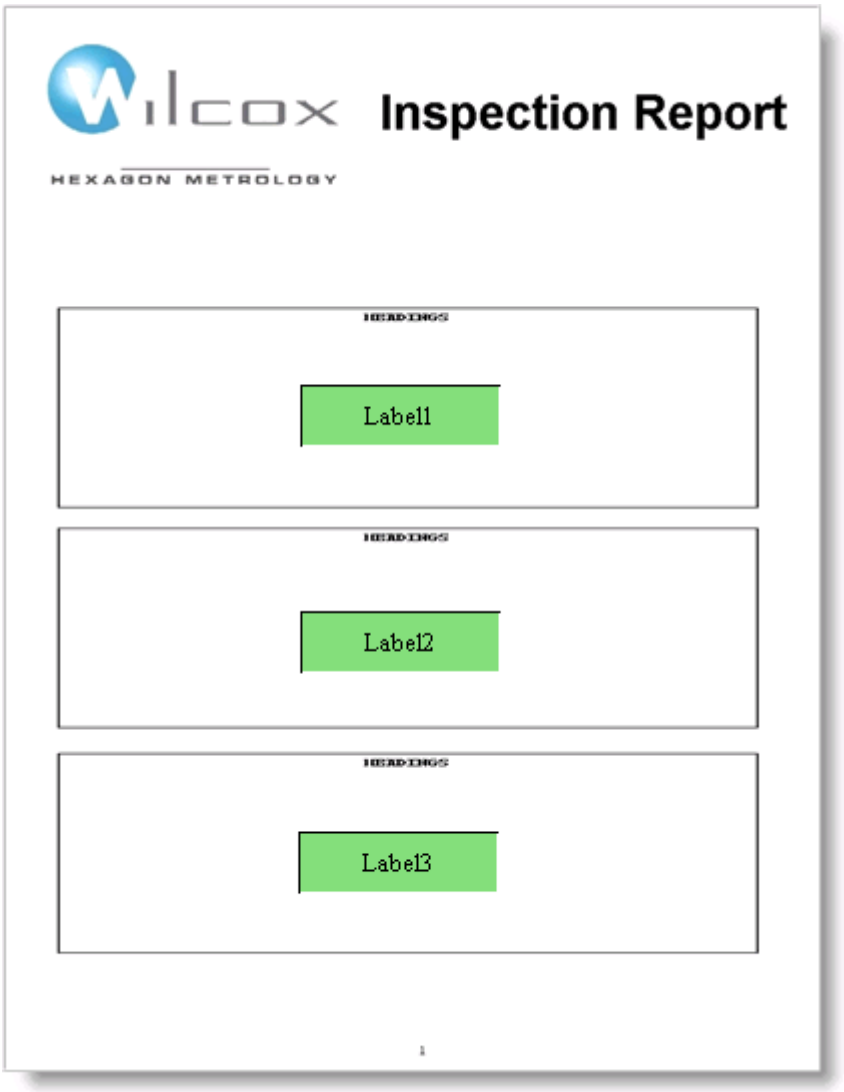
Changing Stand-Alone Label Ordering

Since PC-DMIS may not initially fill the labels in the order you would like, you can change the drawing order of the labels to determine which labels are filled first with menu items located on the **Edit | Order** submenu.

You can also click the **To Front or Back** icon on the **Layout Bar**, and then click the **Bring to Front** or **Send to Back** button.

After you change the drawing order, PC-DMIS fills the top most drawn label first and the bottom most drawn label last.

For example, suppose you have three large labels that fill up a single section with Label1 at the top, Label2 in the middle, and Label3 at the bottom.



A sample report template with three large labels

Initially, it fills up Label3 first, then Label2, and then Label1. To guarantee that PC-DMIS displays your data in measurement routine order, you can change the drawing order of the labels, like this:

1. Select Label1 and choose **Edit | Order | Bring to Front**.
2. Select Label2 and choose **Edit | Order | Sent to Back**.
3. Select Label3 and choose **Edit | Order | Send to Back**.

Arranging Labels in the Report Window

When the report template receives measurement routine data, it has to size the labels dynamically to fit that data. So, these labels may not be in the best location or may overlap other labels after measurement routine execution. However, you can modify where to put labels, by selecting a label and then dragging it to a new location.

You can select a label in one of these ways.

- Double-click on the label.
- Press the Ctrl key and left-click the label.
- Drag a box around the label.

The latter two methods are used to select multiple labels as well.

Using the Table Format Command

The `TABLE/FORMAT` Edit window command lets you control a **GridControlObject's** column and row ordering and visibility on the fly from within your measurement routine. This means you don't have to use the Label Template Editor to create different variations of the same label template if you only want to reorder or hide certain rows or columns. Instead, you can simply insert this command into your measurement routine to determine how to display the **GridControlObject** in the label template.

For this command to work properly, you must set the value of the **GridControlObject's** `TableFormat` property to the exact same table format name defined in the **Grid Format** button of the **Edit Rule** dialog box.

Inserting the Command

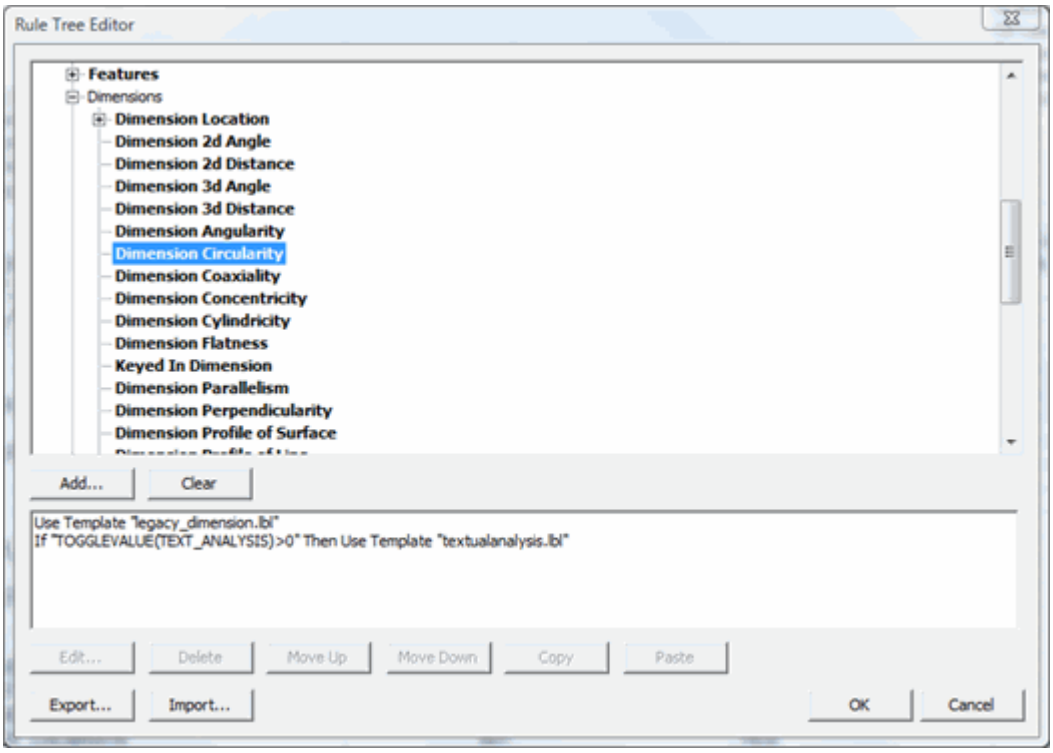
To insert this command, select **Insert | Report Command | Table Format**. The **Table Format Properties** dialog box appears. This dialog box, like the dialog box used in the **Rules Tree Editor**, also controls the visibility and ordering of rows and columns for label templates using the **GridControlObject**. When you finish manipulating the rows and

columns, click **OK**, and PC-DMIS inserts the `TABLE/FORMAT` command into the Edit window.

An inserted `TABLE/FORMAT` command takes precedence over any ordering you may have already defined with the **Edit Rule** dialog box's **Grid Format** button.

About the Rule Tree Editor

The **Rule Tree Editor** dialog box lets you define rules–or conditions and responses–that certain objects follow when displaying report information. For example, you may only want to show roundness dimension information for circle features in your report, even though your measurement routine contains many other features. You can do this by using this editor. In your report template, you might add a **CadReportObject** and then access the **Rule Tree Editor** for that object. Inside the editor, you can then specify a circle feature type and then select a label template that is programmed to display roundness data.



Rule Tree Editor Dialog Box

Supported Objects:

The **Rule Tree Editor** works with these objects available from the Report Template Editor:

- TextReportObject
- CadReportObject
- Label Object

It also works with **Page** object available within the **Custom Report Editor**. Only these objects can access data from a measurement routine using the **Rule Tree Editor**.

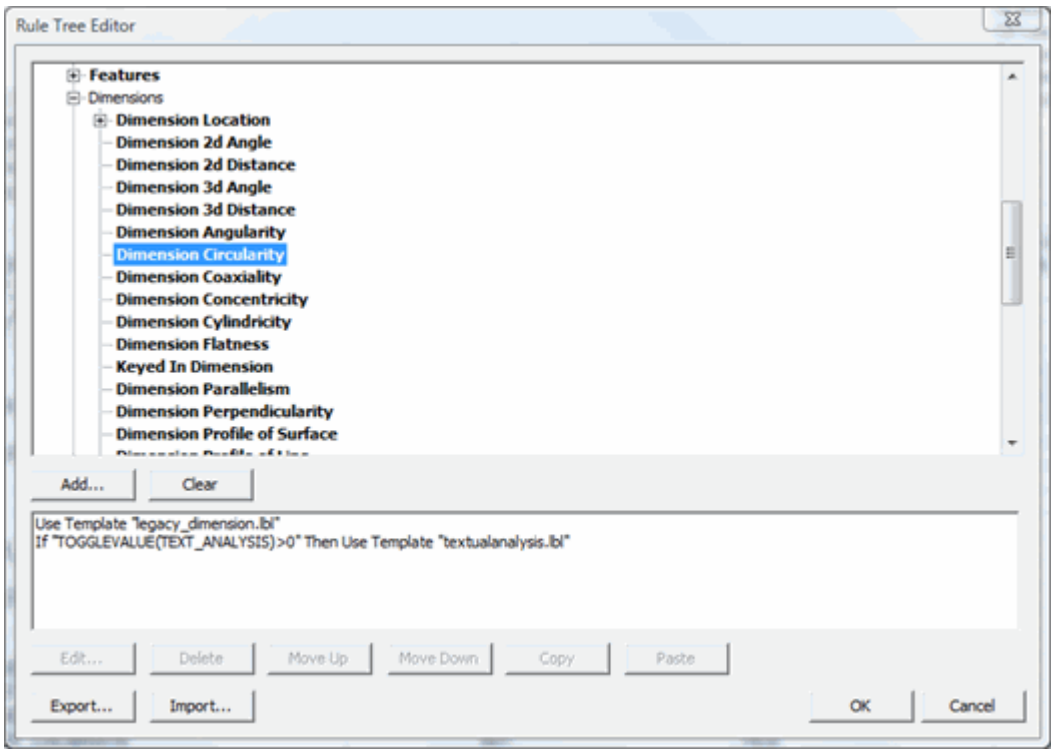
Accessing the Rule Tree Editor:

1. Insert a supported object into your report template, or select the **Page** object in the **Custom Report Editor**.
2. Right-click on the object.
3. Click on **Rules** next to the **RuleTree** property to open the **Rule Tree Editor** dialog box.

Defining a Rule in the Rule Tree Editor

To define a rule using the **Rule Tree Editor** dialog box, follow these steps:

1. Select **File | Reporting | New | Report Template** to open the Report Template Editor.
2. Add a **TextReportObject**, **CadReportObject**, or **Label** object to the template.
3. Right-click the object to open the **Properties** dialog box.
4. Select the **Rules Tree** property to open the **Rule Tree Editor** dialog box. The dialog box lists various items that make up a measurement routine. To expand an item, click the plus (+) symbol to view more specific items or commands. Some standalone items that are not inside expanding lists include:

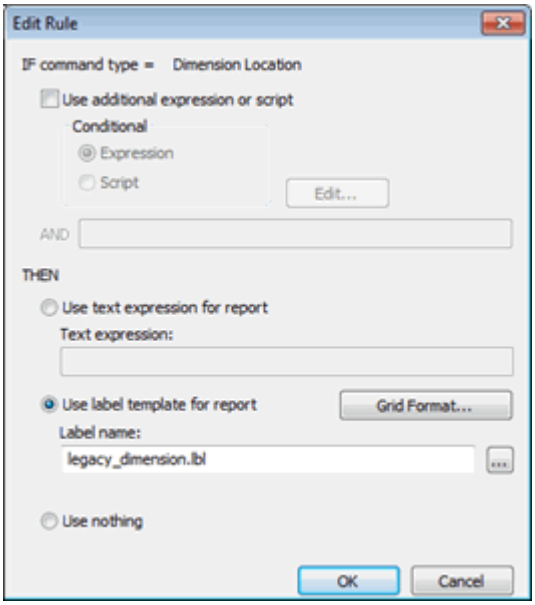


Rule Tree Editor Dialog Box

- **Top of First Page in Section** - You can use this item to display a label or text expression at the top of the first page in the current section of the report.

- **Bottom of Last Page in Section** - You can use this item to display a label or text expression at the bottom of the last page of the current section.
- **File Header** - You can use this item to replace the standard report header (such as PART NAME, REV NUMBER, SER NUMBER, and STATS COUNT) with a custom label or text expression. The File Header appears on the first page of the current section of the report.

5. Define the conditions to be met:
   - From the list, select an item to define as a condition.
   - Click **Add** to open the **Edit Rule** dialog box.



Edit Rule Dialog Box

   - If you want to define another condition, you can choose either a reporting language expression or a BASIC script. To do this, select the **Use Additional Expression or Script** check box, and then in the **Conditional** area, select either the **Expression** or **Script** option.
     - If you select **Expression**, type the condition in the **Conditional Expression** box. For example, to create a rule for a measured circle, you can test any circle data type field values to determine whether to display a label. Suppose you want to display a label only if the measured circle has less than four hits. In the **Conditional Expression** box, type:

       ```
       N_HITS < 4
       ```

       If this condition is evaluated as TRUE, PC-DMIS uses the associated label. You do not need to include the entire IF statement. The IF statement is already understood and evaluated in this box.

- If you selected **Script**, click the **Edit** button to create your BASIC script. A **VBS Mini-editor** appears where you can type your code. The script must return TRUE for the condition to be met. When finished, right-click and select **Check Syntax**, then click **OK**. PC-DMIS automatically stores the script with the defined rule.

If the rule is deleted, the script is lost. To prevent this, store your script in a separate text file.

6. Define a label, an expression, or nothing to display if the conditions are met. In the **Edit Rule** dialog box, select one of the following:

- **Use text expression for report**
  - *If you use a text expression*, type a simple text message or define an expression in the **Text Expression** box. For example, whenever PC-DMIS measures a circle, you can type static text, such as:

```
"We just measured a circle!"
```

  - You can also use expressions to include information about the circle, such as:

```
"We just measured a circle! It had " +  N_HITS  + " hits!"
```

- **Use label template for report**
  - *If you use a label template*, select a label template. This field also accepts expressions, provided they evaluate to a label template name. If you use an expression, precede it with the "=" symbol. If desired, click the **Grid Format** button to open the **Table Format Properties** dialog box. Use this dialog box to define a grid format for your report. A grid format lets you reorder or hide your columns and rows without creating a brand-new label template. Defining a grid format also lets you use the `TABLE/FORMAT` Edit window command to control row and column ordering from within the measurement routine. For details on how to define a grid format, see the "Using the Table Format Properties Dialog Box" topic in the PC-DMIS Core documentation.

- **Use nothing**
  - *If you use nothing*, then PC-DMIS displays nothing for that condition. You may find this useful if you don't want to output anything for a specific feature, command, or condition. **Use nothing** only applies to the TextReportObject.

7. After you fill out the **Edit Rule** dialog box, click **OK**. PC-DMIS displays the item in bold and adds its rule at the bottom of the **Rule Tree Editor** dialog box.

8. Organize your rules. You can create multiple rules for a given command type. If a command has more than one rule, PC-DMIS evaluates them in the order they appear. To change the evaluation order, select a rule and click **Move Up** or **Move Down** in the **Rule Tree Editor** dialog box.

    - To copy and paste rules, use the **Copy** and **Paste** buttons. You can copy rules to another item, multiple items, or an entirely different tree. For details on how to copy and paste rules, see the "Copying and Pasting Rules" topic in the PC-DMIS Core documentation.

    - To remove rules from an item, click **Delete** or **Clear**. To remove rules from multiple items, select them and click **Clear**. For details on clearing rules, see "Clearing Rules" topic in the PC-DMIS Core documentation.

9. Click **OK** to save your changes in the **Rule Tree Editor**.

10. Test the rule.

    - Save the report template.

    - Apply the report template to the report data using the **Reporting** toolbar in the Report window.

    - Execute the measurement routine.

A rule assigned to a child element always overrides one assigned to a parent element. A rule assigned to a parent node applies to each child element unless the child element has its own rule.

Importing and Exporting Rules

If you work in a collaborative environment, you will probably want to share rules that are created from different report templates with each other. Using the **Rules Tree Editor** dialog box, you can easily do this and import someone else's rules by using the **Import** button, or you can share your own rules with someone else by using the **Export** button.

A rules file has a ".rul" filename extension.

Importing a Rules File:

1. Access the **Rules Tree Editor** dialog box.

2. Click the **Import** button. An **Open** dialog box appears.

3. Navigate to the directory containing the rules file (.rul) you want to import.

4. Click **Open**. PC-DMIS imports the rule file into the **Rules Tree Editor**.
5. Click **OK** to save the imported rule.

Exporting to a Rules File:

1. Access the Rules Tree Editor dialog box.
2. Click the **Export** button. A **Save As** dialog box appears.
3. Navigate to the directory where you want to store the rules (.rul) file.
4. Type a name for the file In the **Filename** box.
5. Click **Save**. All rules associated with that object get exported. Someone else can now import and use your exported rules file.

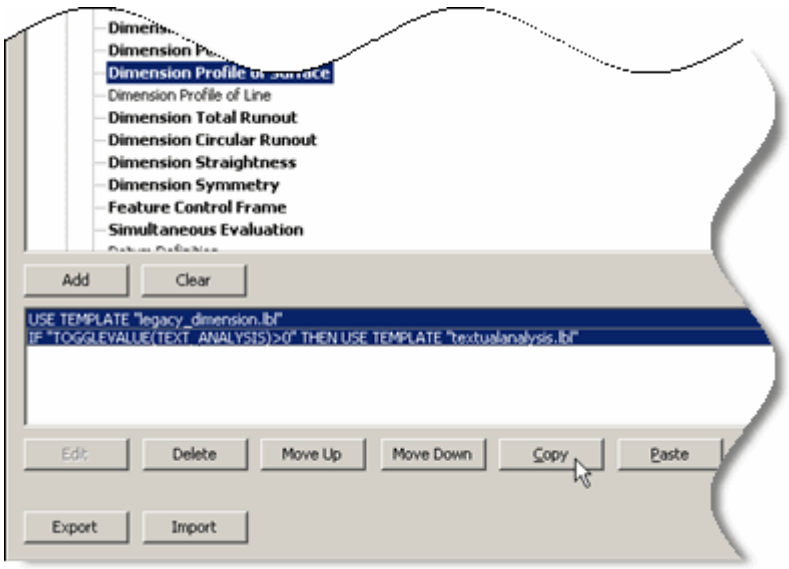
Copying and Pasting Rules

The Rule Tree Editor contains **Copy** and **Paste** buttons so that you can copy and paste rules between the current **Rule Tree Editor** and a **Rules Tree Editor** for a different object or to a different item within the same **Rule Tree Editor**.

A copied rule is not stored in the Windows Clipboard; so you cannot copy and paste a rule outside of the specific context discussed here.

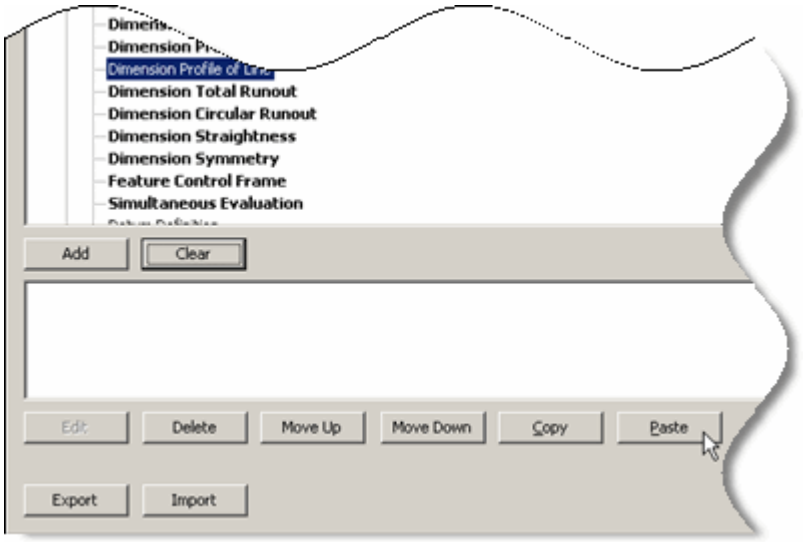
To Copy and Paste Rules:

1. Select a single item from the **Rule Tree Editor**. If you select more than one item from the tree editor, the **Copy** button will never become enabled.
2. Select one or more rules from the rule list box. The **Copy** button becomes enabled for selection.



Example showing the copying of two rules.

3. Click **Copy**.
4. Select and highlight one or more items from a rule tree. Hold down the CTRL key while clicking to select multiple items. The **Paste** button becomes available.

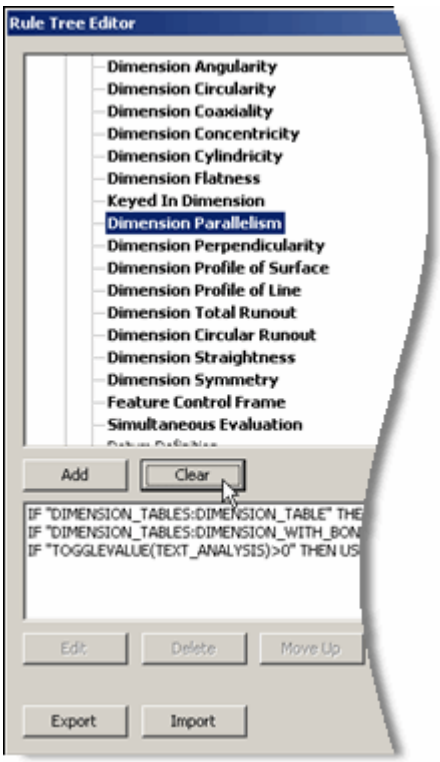


Example showing the pasting of copied rules into an empty item.

5. Click **Paste** to paste the copied rule(s) into the selected item(s). Note that if you paste a copied rule or rules into an item that already contains one or more rules, the existing rules are not overwritten by the new rules, but the new rules are instead appended to the existing list. You can click **Paste** multiple times into several different rule tree items.
6. Click **OK** on the **Rule Tree Editor** to save your changes.

Clearing Rules

The **Clear** button removes any rules associated with one or more selected items from the Rule Tree Editor. You can select more than one item by pressing the CTRL key while selecting additional items.



Example showing the clearing of rules for a selected item.

Using the Table Format Properties Dialog Box

The **Table Format** dialog box works with a **GridControlObject's** defined column and row names, which enables you to control the visibility and order of columns and labels in a **GridControlObject** in a label template. To access this dialog box, from the **Edit Rule** dialog box, click the **Grid Format** button. For information on the **Edit Rule** dialog box and the **Grid Format** button, see "Defining a Rule in the Rule Tree Editor".

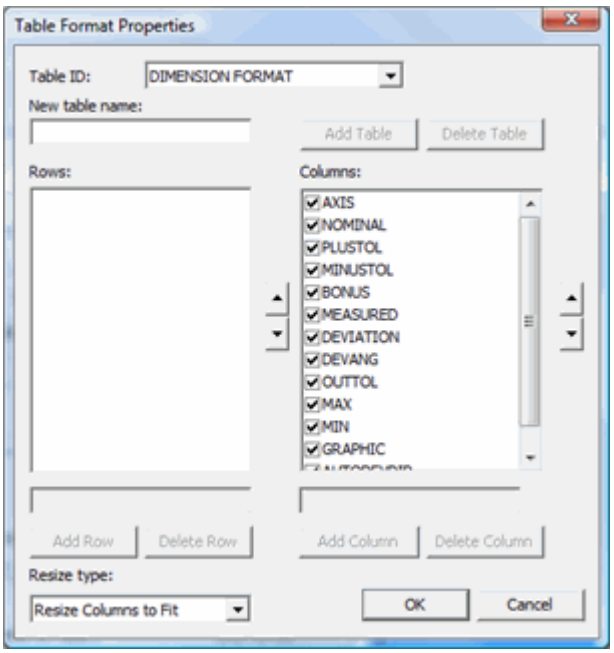


Table Format Properties Dialog Box

This dialog box contains different table formats. A table format simply defines how you want named rows and columns to appear in your final report. This dialog box lets you change the order and visibility state of all existing table formats. You can also use this dialog box to create your own custom table formats.

For the pre-existing table formats, the only thing you can modify is their visibility state and the order of their rows or columns. You cannot delete these table formats nor change what rows or columns they have.

To use table formats, you must first use a label template with a **GridControlObject** that has its columns or rows already named.

The following properties are available:

Table ID
Lists all available table formats.

New Table Name
Defines a new table format.

Add Table
Adds the new table format. This button only becomes enabled when you type a name in the **New Table Name** box.

Delete Table
Deletes the selected custom table format. This button only becomes enabled when you select a custom table from the **Table ID** list.

Rows
Lists the rows and columns in your table format. You can reorder them by clicking the up and down arrows.

Add Row/Column
Adds a row or column name to the **Rows** or **Columns** area. You can only add to custom table formats.

Delete Row/Column
Deletes the selected row or column from the **Rows** or **Columns** area. You can only delete items from custom table formats.

Resize Type
Determines what the table will when you show or hide columns. The list contains three items:

Resize Grid - This retains existing column sizes and resizes the grid to fit the new width.

Resize Columns to Fit - This retains the grid's existing width and resizes each column equally to fit that width.

Hide Text - This doesn't resize the grid or the columns. It simply hides the text.

Cancel
Closes the dialog box without applying any changes.

OK
Uses the selected table format for the rule in the **Rule Tree Editor** or for the `TABLE/FORMAT` command in the Edit window.

To name a column or row in a GridControlObject:

1. In the Label Template Editor, select the **GridControlObject**.
2. Double-click on the first cell in a column or row.
3. Right-click to access the property editor.
4. Click either the **Cell** or **Column** tab.
5. In the **Row Label** box or **Column Label** box enter a value. You can select from the list or type a custom value. Custom labels can only show up on custom table formats.

To reorder columns or rows:

1. Select a table format from the **Table ID** list.
2. Its rows or columns already appear.
3. In the **Rows** or **Columns** area, select the row or column you want to reorder, then click the up or down arrow buttons to either send that item either up or down in the list. Items higher in the list get displayed before others.

To show/hide columns or rows:

1. Select a table format from the **Table ID** list. Its rows or columns already appear.
2. In the **Rows** or **Columns** area, clear the check box to the left of the row or column name to hide that item in the label. Select the check box to show it in the label.

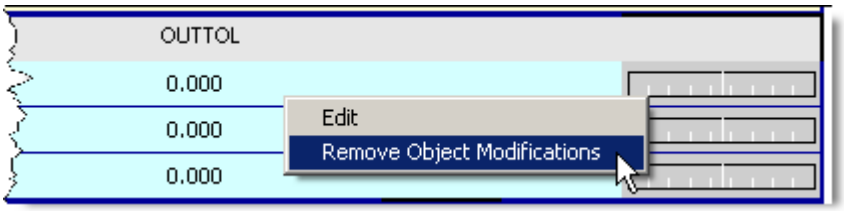
To create a custom table format:

1. In the **New Table Name** box, type a name.
2. Click the **Add Table** button. The custom table appears in the **Table ID** list and remains until you delete it.
3. In the **Add Row** or **Add Column** box, type a name that matches a row or column label name you gave to the **GridControlObject**.
4. Click **Add Row** or **Add Column** to add that item to the table.

5. Continue adding row or columns and deleting or rearranging them as desired until the table format contains the information you want.

Removing your Modifications

To quickly remove a modification, right-click on the desired report object (a TextReportObject, CADReportObject, or Label object), and select the **Remove Object Modifications** button.

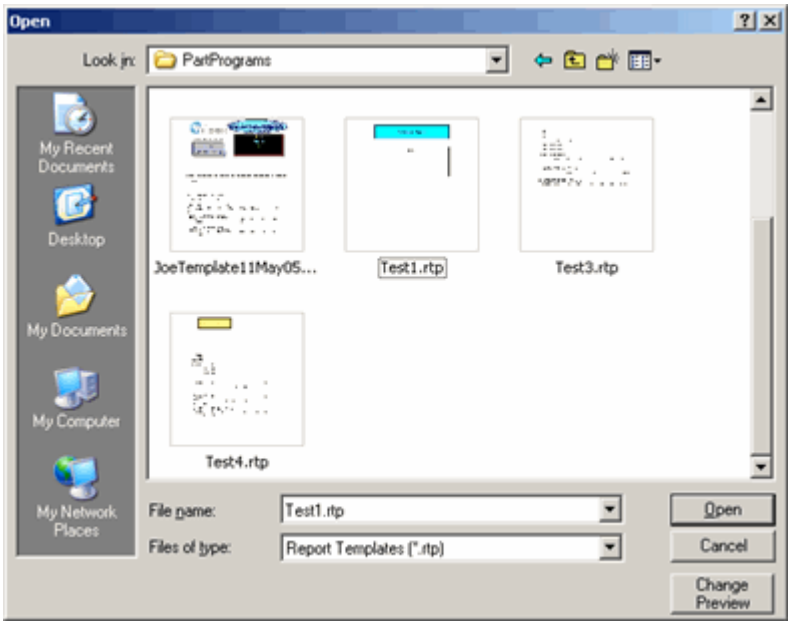


PC-DMIS returns the report object back to its default state.

Changing a Template's Preview Thumbnail Icon

Whenever you create a report or label template, PC-DMIS automatically creates a thumbnail-sized preview of that template based on what appears on the template's initial page or section.

This thumbnail preview appears whenever you attempt to edit a template and access the **Open** dialog box (**File | Reporting | Edit | Report Template** or **Label Template**).



A sample Open dialog box showing several report template preview icons

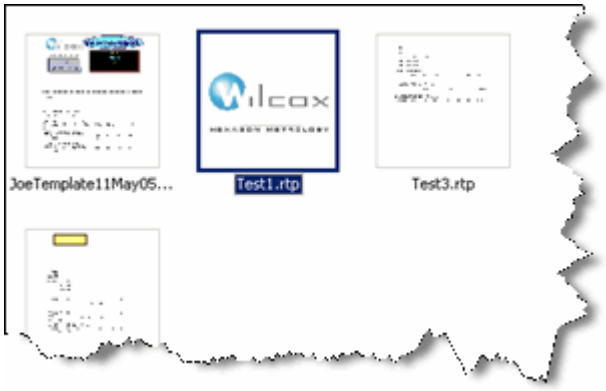
If you want to use a custom image file instead of the default generated preview for a template, you can do this within the **Open** dialog box.

To Change a Template's Preview Thumbnail:

1. Select the template file from the **Open** dialog box.
2. Click the **Change Preview** button to open the **Template Image Selection** dialog box.

Template Image Selection Dialog Box

3. Select the **Select Image from File** option button.
4. Click **OK**. Another **Open** dialog box opens allowing you to select a preexisting image file.
5. Navigate to the directory containing either a .bmp or .jpeg file, select the image file, and click **Open**. PC-DMIS uses the selected image for your preview.



Example of a Test1.rtp file with a bitmap preview image.

Saving a Template or Form to an Earlier Version

You can save your report templates, label templates, or forms so that they are backwards compatible with earlier PC-DMIS versions that also support template reporting.

To do this:

1. Open the desired report or label template inside its respective editing environment (the Report Template editor, the Label Template editor, or the Form Editor).
2. Select the **File | Save As** menu item to open the **Save As** dialog box.

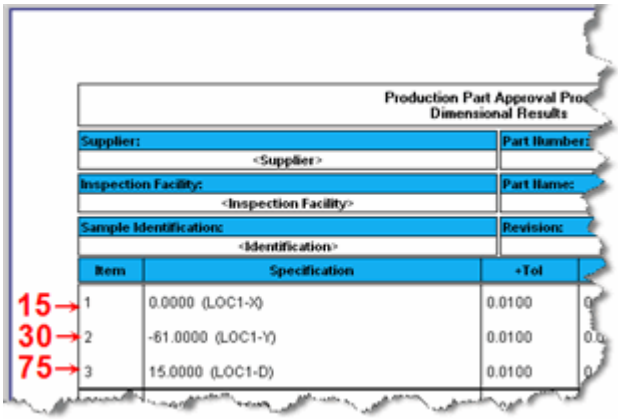
3. Near the bottom of the dialog box, use the list under the **Save as type** list to specify the version. The earliest version you can save these items back to is version 4.2 MR2. Select the desired version and click **Save**.

If you use the **File | Save** menu item, PC-DMIS automatically saves the item to your current running version of PC-DMIS, even if it was saved to an earlier version previously.

Tutorial - Customizing a Report Template

Most of the time, you won't need to create a brand-new report template from scratch, but instead will want to create a new template based on an existing template that ships with PC-DMIS.

For example, suppose that you have ballooned (or numbered) prints, and you want the ballooned numbers to appear in your final report, such as in the **Item** column in the PPAP report (as shown below).



How do you get ballooned data into the PPAP template?

Perhaps feature #15 on the print is the X dimension for a hole, feature #30 is the Y dimension, and feature #75 is the hole's diameter. How do you get this information into the **Item** column in your final report? One way is to configure the PPAP and label templates to pull the information from variables in your measurement routine.

This tutorial shows you how to make copies of the PPAP report and label templates. It then demonstrates how to modify them so that the information that appears under the **Item** heading displays variables taken from your measurement routine, instead of the normal sequential list of numbers that appears in the PPAP report.

This tutorial uses skills learned in the "Creating a Report Template" and "Creating Label Templates" tutorials. You should complete those tutorials first so that you have a proper foundation for this tutorial.

Step 1: Copy the PPAP Report and Label Template

The first thing you need to do is create copies of all the components used to generate the PPAP report. These include both the report template and the underlying label templates.

1. In the Report Template Editor, open the PPAP.RTP report template.
2. Select **File | Save As**.
3. Name the report template "BALLOON_PPAP.RTP".
4. Close the Report Template Editor.
5. In the Label Template Editor, open these templates, select **File | Save As** on each one, and save them with a "BALLOON_" filename prefix as shown in this table:

Existing Label Template	Save As Filename
PPAP_DIMENSION.LBL	BALLOON_PPAP_DIMENSION.LBL
PPAP_DIMENSION_TRUE_POSITION.LBL	BALLOON_PPAP_DIMENSION_TRUE_POSITION.LBL
PPAP_SizeTolerance.LBL	BALLOON_PPAP_SizeTolerance.LBL
PPAP_GEOTOL_SIZE.LBL	BALLOON_PPAP_GEOTOL_SIZE.LBL
PPAP_GEOTOL_SEGMENT1.LBL	BALLOON_PPAP_GEOTOL_SEGMENT1.LBL
PPAP_GEOTOL_SEGMENT2.LBL	BALLOON_PPAP_GEOTOL_SEGMENT2.LBL
PPAP_GEOTOL_SEGMENT3.LBL	BALLOON_PPAP_GEOTOL_SEGMENT3.LBL
PPAP_GEOTOL_SEGMENT4.LBL	BALLOON_PPAP_GEOTOL_SEGMENT4.LBL
PPAP_GEOTOL_SEGMENT5.LBL	BALLOON_PPAP_GEOTOL_SEGMENT5.LBL
PPAP_TRUE_POSITION_INBETWEEN.LBL	BALLOON_PPAP_TRUE_POSITION_INBETWEEN.LBL

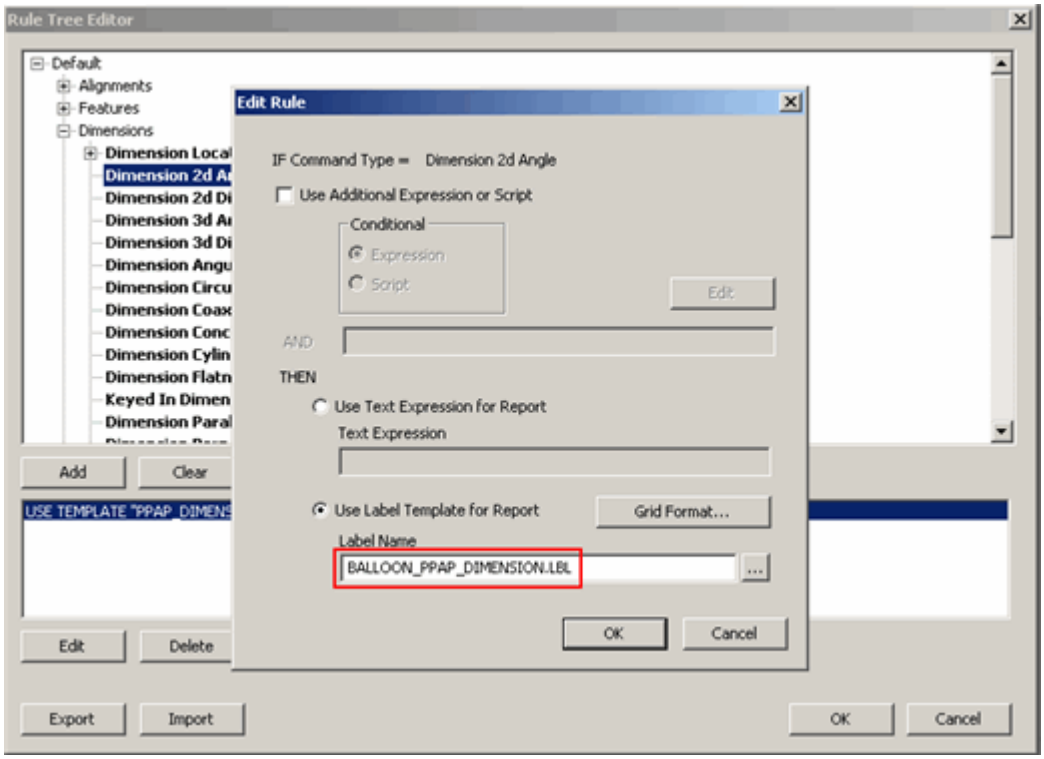
You should now have copies of all the report and label templates used with the PPAP report. You will customize these copies in subsequent steps.

Step 2: Modify Rules in the Report Template

Now that you have created copies of the report and label templates, you need to modify the rules on the new report template so that the newly saved BALLOON label templates are used instead of the usual PPAP labels.

1. In the Report Template Editor, open the BALLOON_PPAP.RTP report template.
2. Access **TextReportObject1** on the template and right-click on it to access the properties for the object.
3. In the **Properties** dialog box, click **Rules** to access the **Rule Tree Editor**.
4. Expand the **Dimensions** list in the **Rule Tree Editor**. You see several dimensions in boldface font indicating that a rule is applied to that dimension type.
5. Select each boldface item in the **Dimensions** list, and modify its existing rule or rules so that the corresponding label templates with the "BALLOON_" prefix are used instead.

Existing Label Templates Used	New Label Templates to Use
PPAP_DIMENSION.LBL	BALLOON_PPAP_DIMENSION.LBL
PPAP_DIMENSION_TRUE_POSITION.LBL	BALLOON_PPAP_DIMENSION_TRUE_POSITION.LBL
PPAP_SizeTolerance.LBL	BALLOON_PPAP_SizeTolerance.LBL
PPAP_GEOTOL_SIZE.LBL	BALLOON_PPAP_GEOTOL_SIZE.LBL
PPAP_GEOTOL_SEGMENT1.LBL	BALLOON_PPAP_GEOTOL_SEGMENT1.LBL
PPAP_GEOTOL_SEGMENT2.LBL	BALLOON_PPAP_GEOTOL_SEGMENT2.LBL
PPAP_GEOTOL_SEGMENT3.LBL	BALLOON_PPAP_GEOTOL_SEGMENT3.LBL
PPAP_GEOTOL_SEGMENT4.LBL	BALLOON_PPAP_GEOTOL_SEGMENT4.LBL
PPAP_GEOTOL_SEGMENT5.LBL	BALLOON_PPAP_GEOTOL_SEGMENT5.LBL
PPAP_TRUE_POSITION_INBETWEEN.LBL	BALLOON_PPAP_TRUE_POSITION_INBETWEEN.LBL



Changing the Label Template

6. Save the report template.

You have modified your existing rules to use the label templates with the "BALLOON_" prefix to format and display your report data.

Step 3: Add ASSIGN Statements into your Measurement Routine

Now that the rules are modified, the next thing you need to do is insert ASSIGN statements into your measurement routine for each ballooned item. Prior to each dimension in your measurement routine, add ASSIGN statements for each characteristic you will display in that dimension. For example, at the beginning of this topic, we suggested that feature #15 on the print was the X dimension for a hole, feature #30 the Y dimension, and feature #75 the hole's diameter. If you had a LOCATION dimension to display the X, Y, and D values (three characteristics), you would insert these three ASSIGN statements prior to your dimension:

```
ASSIGN/V1[1]=15
ASSIGN/V1[2]=30
ASSIGN/V1[3]=75
```

These statements create an array of multiple values for the V1 variable.

Apply similar ASSIGN statements before each dimension in your measurement routine.

- If your dimension displays only one characteristic, then you would only put one ASSIGN/V1[#] before that dimension statement.

- If your dimension displays ten characteristics (for example, a Geometric Tolerance Position), then you would need ten ASSIGN statements for the ten elements in your array:

```
ASSIGN/V1[1]=2
ASSIGN/V1[2]=4
ASSIGN/V1[3]=6
ASSIGN/V1[4]=24
... and so on until ...
ASSIGN/V1[10]=76
```

In most cases, you must use the same variable name for all ASSIGN statements for all dimensions. This tutorial uses the variable V1. The exception is for a Position legacy dimension. For this dimension type, you can use V1[1], V1[2], and other variables for the XYZ axis. However, for diameters of features and datums, and the position callout, you should use unique variables, perhaps V2[1], V2[2], and V3[1] respectively. Also be aware that the variable can only take number characters. Alphabetic characters show only zeros in the report.

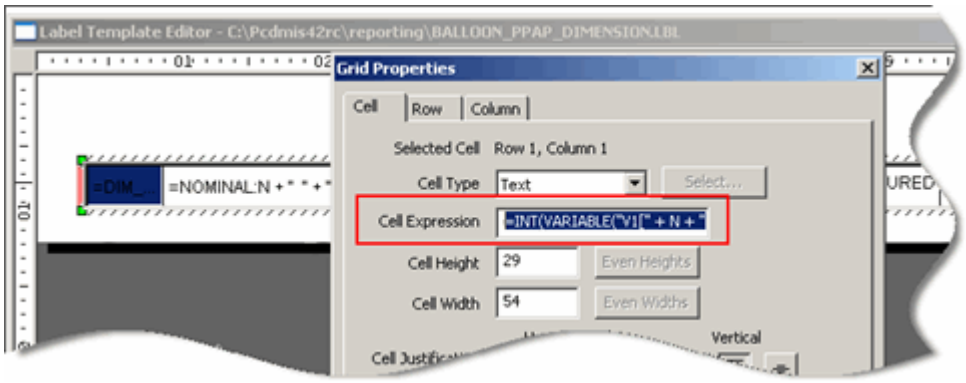
When you've finished adding in the ASSIGN statements, your measurement routine is finished and ready to execute. The only thing left to do is modify your label templates to read in the variable's array of values for each dimension.

Step 4: Modify Label Templates to Read Variables

In this step, you open each applicable label template and configure it so that it reads the array of values for the variable.

1. Inside the Label Template Editor, with the exception of BALLOON_PPAP_DIMENSION_TRUE_POSITION.LBL, open the other label templates with the "BALLOON_" prefix given earlier.
2. Access the GridControlObject and access the contents of the first cell.
3. Modify the expression there to read as follows:

```
= INT ( VARIABLE ("V1["+N+"]",ID:N))
```



4. Save each label template.
5. Finally, open BALLOON_PPAP_DIMENSION_TRUE_POSITION.LBL. Since this template has more than one row in the GridControlObject, you need to specify more than just the V1 variable.
    - *For the XYZ axis*, you can use the V1 variable in your expression:

      ```
      = INT ( VARIABLE ("V1["+N+"]",ID:N))
      ```

    - *For diameters of datums and features*, you need it to use the unique variable used earlier (such as V2):

      ```
      = INT ( VARIABLE ("V2["+N+"]",ID:N))
      ```

    - *For the position callout,* use another unique variable used earlier (such as V3):

      ```
      = INT ( VARIABLE ("V3["+N+"]",ID:N))
      ```

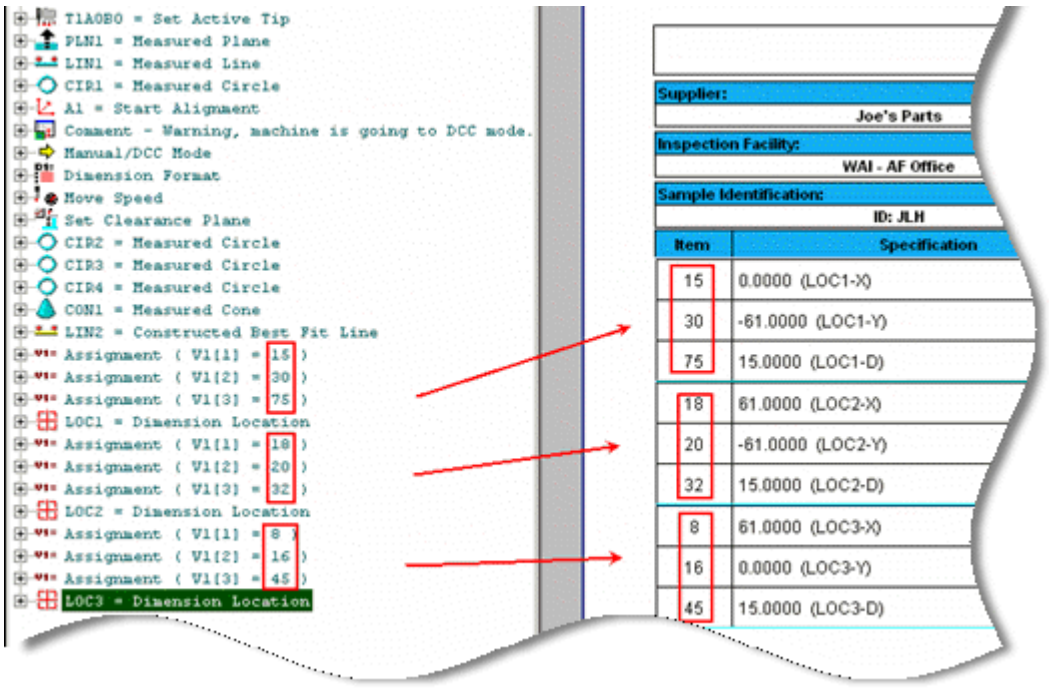
As the report increments through each dimension (using :N), the variable array is called with "V1[" + N + "]", and the Nth element of the V1 array is displayed. You are now ready for the final step of executing and viewing your results.

Step 5: Execute and View Results

In this final step, you define the report template to use in the Report window, execute your measurement routine, and view your final results.

1. Save all your changes made to this point.
2. Access the Report window, and load the BALLOON_PPAP.RPT template.
3. Execute your measurement routine. You may also need to click the **Redraw the Report** icon from the Reporting toolbar after execution.

The Report window shows your final report. It should be similar to this, where the numbered prints inside your PPAP report appear:



Final Report Showing Custom Numbered Items inside of a PPAP Report

Creating Forms

The Form Editor (**File | Reporting | New | Form Report**) lets you build interactive forms and dialog boxes and activate them when a measurement routine gets executed. These interactive forms use the Visual BASIC language. Forms, mixed with a good understanding of the Visual BASIC language and PC-DMIS automation, can give power and flexibility to your measurement routines.

Forms play an active part in reporting because whenever you use a form to control what gets executed, you indirectly control what ends up in your report.

This section, however, does not attempt to cover everything that you can do with forms. Instead, it covers how to use create and use forms in general, and in a reporting sense, how to get information from the user and into a report using standard PC-DMIS commands.

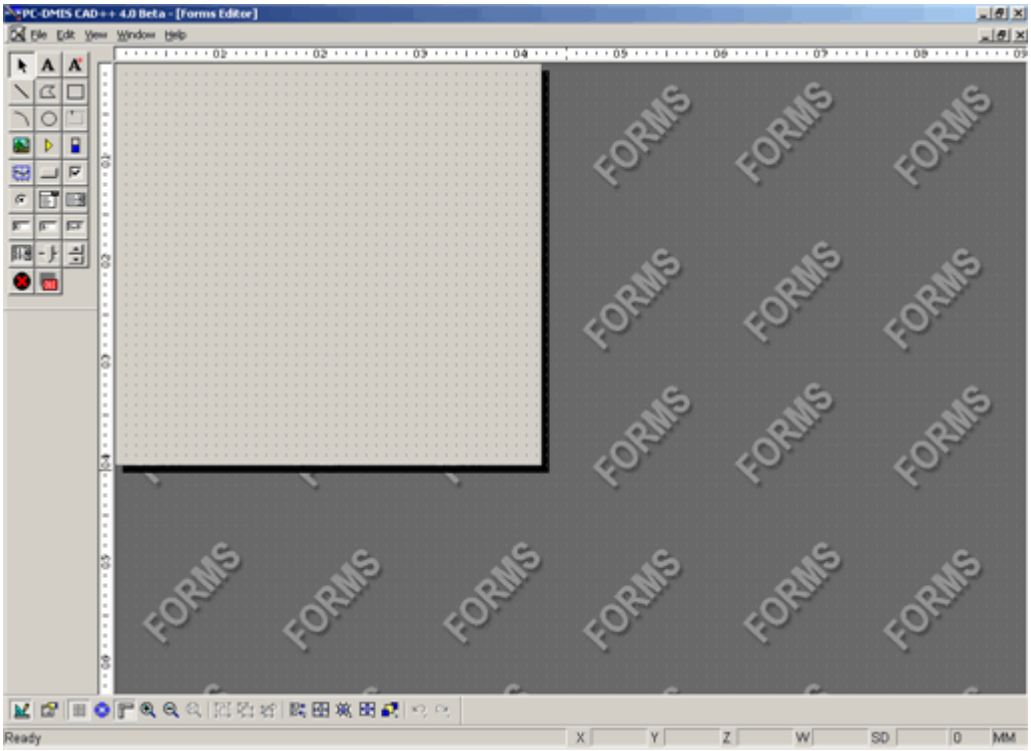
Forms can also be used in a non-reporting role as a way to provide useful instructions to operators. See "Providing Operator Instructions Using OLE Objects with Forms" for some examples on how to do this.

Understanding the Form Editor

To create a new form in the Form Editor, select **File | Reporting | New | Form Report**.

To open an existing form in the Form Editor, select **File | Reporting | Edit | Form Report**.

PC-DMIS displays the Form Editor in a new window.



Form Editor

The Form Editor acts as a work area. It enables you to drag, size, and place various interactive form objects, such as buttons, list boxes, edit boxes, and so forth, and set their properties.

Objects are placed on the gray and sizable area, called a form. If you have worked with HyperView reporting in previous versions of PC-DMIS, you will feel at home with this editor since it essentially functions much the same way and contains many of the same user interface elements.

With the new template approach to reporting, only the Form Editor can use the run-mode capability. The Report and Label template editors do not use it.

> **Run Mode** - This mode only works with the Form Editor. In run mode, you can run (or execute) your form. This enables you to test how it would look or behave in an actual measurement routine execution environment.
>
> **Edit Mode** - This is the default mode for all the reporting editors. It is the only mode for the Report and Label template editors.
>
> To switch between run and edit modes, press CTRL+E.

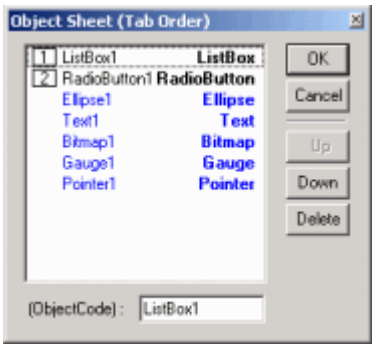
The editor contains the following elements:

- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)

- The Layout Bar (toolbar)
- Object Property Sheets
- Object Sheet

Object Sheet

The Form Editor's **Edit | Layout | Objects** menu item displays the **Object Sheet** dialog box.



Object Sheet Dialog Box

You can use this dialog box to set the tab order for objects and controls typically used on dialog boxes by simply changing the order an object appears in this dialog box.

When you press TAB in run mode, PC-DMIS will jump to the next object defined in this dialog box, giving it the focus. Only some objects are affected by the TAB key. Pressing TAB only works on those objects that have a number to the left of them in the **Object Sheet** dialog box.

Tutorial - Creating Forms

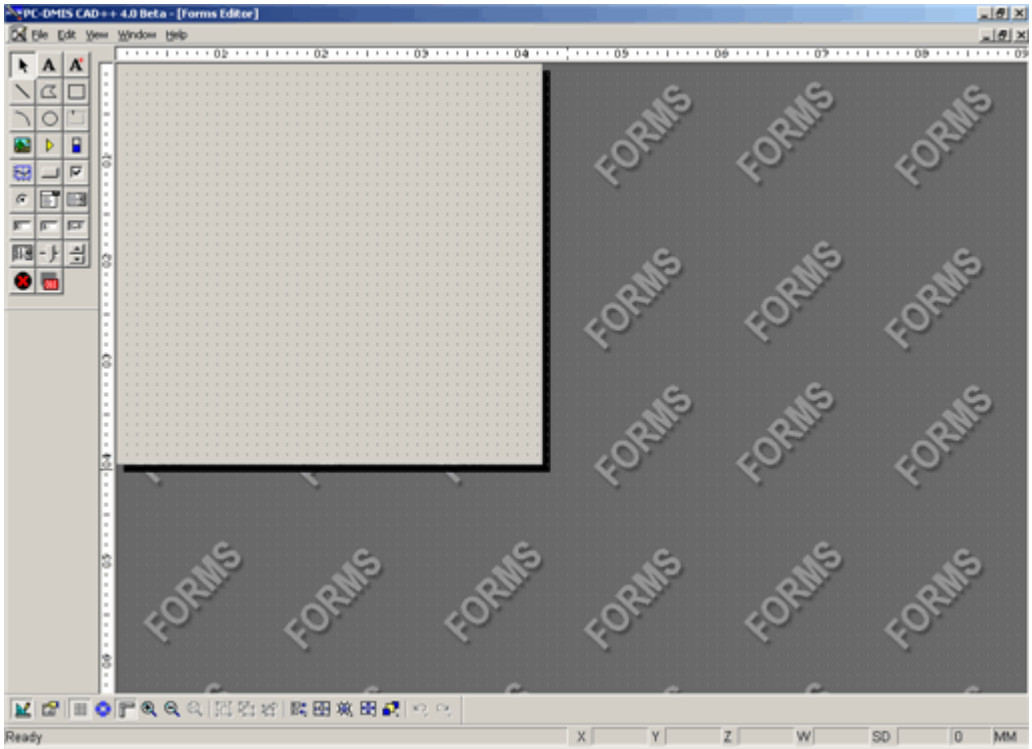
This topic walks you through a basic tutorial that will create a simple form with some interactive dialog box controls that, when executed, allows you to select a user name from a list and then choose to measure only certain features. The information you select will then appear in a final report.

While you can certainly duplicate aspects of this by using certain Edit window commands, this tutorial will give you an idea of what's possible with the Form Editor, some imagination, and a little programming skill.

This tutorial uses files you created in the "Creating a Custom Report Template" and "Creating Label Templates" tutorials. If you haven't already done so, complete those tutorials first.

Step 1: Preparing the Work Environment

1. In this tutorial you will switch between the Form Editor and the Edit window, so first prepare the Edit window.
    - Right-click on your Edit window and select **Docking View** from the shortcut menu to undock it.
    - Select the maximize button so that it fills the screen.
    - Place the Edit window into Command Mode.
2. Select **File | Reporting | New | Form Report** to open the Form Editor.
3. To hide any unused toolbars, right-click on the toolbar area and remove them.
4. To hide any unused PC-DMIS windows, select the open window from the **View** menu. Keep the Edit window open.
5. To maximize the Form Editor, click the *maximize button* in the upper right corner of the editor's window. You should see the words "FORMS" on the background of the editor. Your Form Editor should now look something like this:



Form Editor

When working with the Form Editor, you may find it helpful to hide your usual PC-DMIS toolbars and windows, thereby freeing up some screen space. If you work frequently with this editor, you may want to create a stored screen layout for your template. For information on layouts, see the "Window Layouts Toolbar" topic in the "Using Toolbars" chapter.

Step 2: Create the Form

1. Drag the **TheFrame/TheView's** gray background so that it is five inches wide by six inches tall.

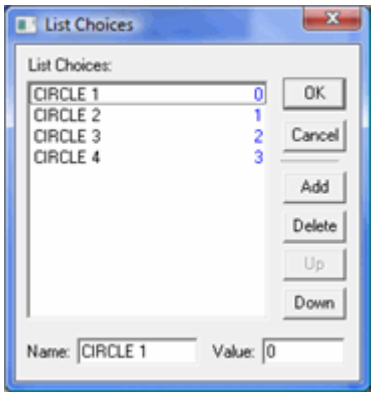
2. Add a **Border** object, and drag the border so that it fits just within the form's area. Set the border's properties so that it has a **LineWidth** value of 3 and a **ForeColor** of dark blue (0.0.128).

3. Add a **Bitmap** object in the top of the form. The **Bitmap Dialog** opens. Use the **Load** button to navigate to a bitmap file of your choice, such as a corporate logo. Size and position the bitmap so that it's within the top 1.5 inches of the form.

4. Add a **RadioButton** object under the Bitmap object. Access the RadioButton's properties, and click on the **ListItems** value. The **List Choices** dialog box opens. This dialog box lets you create a list of option buttons for the object.

    - Select the existing **RadioButton1** item from the **List Choices** box. In the **Name** box, change the name to "CIRCLE 1". The index **Value** should already display 0.

    - Click the **Add** button to add three more items. Then change the newly added items to read "CIRCLE 2", "CIRCLE 3", and "CIRCLE 4". They should have index values of 1, 2, and 3 respectively. The **List Choices** dialog box, should look like this:

    List Choices dialog box showing four options with index values ranging from 0 to 3.

    - Click **OK** when finished.

5. Rename the RadioButton object by changing the **(ObjectCode)** value from "RadioButton1" to "optMeasure".

6. Add a **Frame** object and drag it around your RadioButton object. Change the Frame's **Text** property to read "Select What to Measure".

7. Add an **EditBox** object to the right of the RadioButton object, and rename it from "EditBox1" to "txtMeasure".

8. Add a **Text** object (Text1) under the RadioButton object, and set its **Text** property to read "Select User:".

9. Add a **ComboBox** to the right of the Text object, and rename it from "ComboBox1" to "cboUsers". This control will contain a list of users.

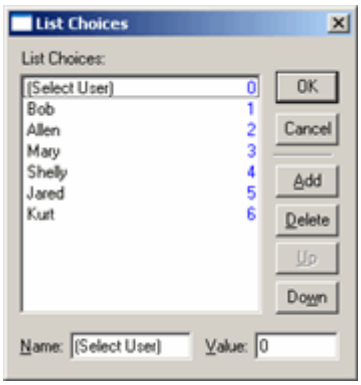
    - Click on the **ListItems** value to add the list of users. This accesses the **List Choices** dialog box again.

    - Click **Add**. PC-DMIS inserts "(None)" under **List Choices**.

    - Change the name of the first item in the list from "(None)" to "[Select a User]", and give it an index **Value** of 0.

    - Continue to use this dialog box as you did before to add five or six user names. This tutorial used "Bob", "Allen", "Mary", "Shelly", "Jared", and "Kurt".

    - Use the **Up** or **Down** buttons as desired to change where an item appears in the list. The dialog box, when finished, should look something like this:



List Choices dialog box showing a list of user names for the ComboBox.

    - Click **OK**. Your ComboBox object now contains those list items.

10. Add an **EditBox** object to the right of the ComboBox, and rename it from "EditBox2" to "txtUser".

11. Add another **Text** object (Text2) under the object that reads "Select User", and set its **Text** property to read "Type a Report Description:".

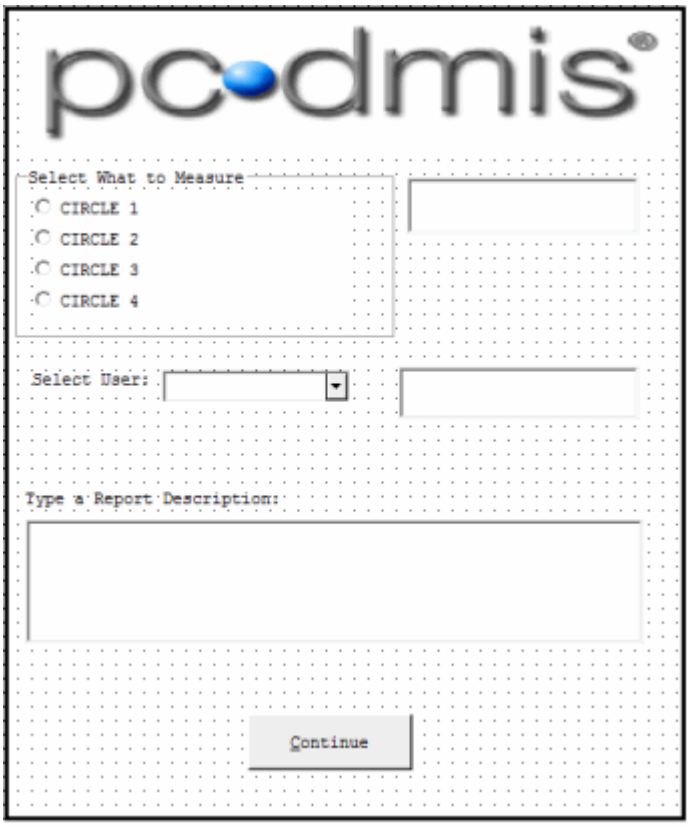
12. Add another **EditBox** object to the right of the above Text object, and rename it from "EditBox3" to "txtDescription".

13. Add a **Button** object to the bottom of the form, and rename it from "Button1" to "cmdContinue". Change its Text property to read "&Continue". The ampersand symbol lets you define whatever character follows it as a shortcut character to be used with Alt. So, pressing Alt + C on your form in Run Mode will act as if you clicked the **Continue** button.

14. Select **File | Save** to save the form. Give it the name "TestForm.Form" and save it in a directory of your choice.

15. Press Ctrl + E to test your form in Run mode. There's no code tied to any of the controls so nothing much happens if you click on anything, but you'll add that next. When finished, press Ctrl + E again to return to Edit mode.

In this step, you have completed the basic form, adding various controls and assigning items to lists. It should look something like this:



Form with the various form controls added

Step 3: Add Code to the RadioButton Control

1. Select the RadioButton object, **optMeasure**, and access its properties.

2. Click on the **EventChange** event. The **VBS Mini Editor** appears. This small code window lets you type Visual BASIC code statements that will run whenever you select a new value from the list of option buttons.

3. In the code window, type this code:

```
Dim intIndex As Integer
intIndex=optMeasure.Index
Select Case intIndex
Case 0
   txtMeasure.Text="Circle 1"
Case 1
   txtMeasure.Text="Circle 2"
Case 2
   txtMeasure.Text="Circle 3"
Case 3
   txtMeasure.Text="Circle 4"
End Select
```

4. Click **OK**. The mini editor closes.
5. Select **File | Save** to save the form.
6. Test your report by pressing Ctrl + E and entering Run mode. Select a feature to measure.

This code runs when you select a feature to measure from the list. It looks at what gets selected from the list of option buttons, and sets the **Text** property of txtMeasure to display a text string of "CIRCLE 1", "CIRCLE 2", and so on. Press Ctrl + E to return to Edit mode.

Step 4: Add Code to the ComboBox Control

1. Select the ComboBox object, **cboUsers**, and access its properties.
2. Click on the **EventChange** event. The **VBS Mini Editor** opens.
3. In this code window, type this code:

```
txtUser.Text=cboUsers.TextValue
```

4. Click **OK**. The mini editor closes.
5. Select **File | Save** to save the form.
6. Press Ctrl + E to enter Run mode to test your report. Select a user.

This code runs when you select a user from the list. It looks at the text value of the selected list item, and sets the **Text** property of txtUser to display a text string of the selected user. Press Ctrl + E to return to Edit mode.

Step 5: Add Code to the Continue button

1. Select the Button object, **cmdContinue**, and access its properties.
2. Click on the **EventClick** event. The **VBS Mini Editor** opens.
3. In this code window, type this code:

```
   If cboUsers.Value>0 And Len(txtDescription.Text)>0
And Len(txtMeasure.Text)>0 Then
      TheView.Cancel
   Else
      MsgBox "Please fill out the entire form before
continuing."
   End If
```

4. Click **OK**. The mini editor closes.
5. Select **File | Save** to save the form.
6. Press Ctrl + E to enter Run mode to test your report. Click the **Continue** button.

This code runs when you click the **Continue** button. It checks to make sure the form is all the way filled out, and if it is, it closes the Form Editor. If not, it displays a message telling the user to fill out what's missing.

- The Len ( ) function tests the length (or the number of characters) for the text string in the edit boxes, making sure something is there.
- The cboUsers.Value > 0 code checks to make sure a user name gets selected from the list.

Step 6: Assign Edit Window Variables to Hold Form PropertiesStep 6: Assign Edit Window Variables to Hold Form Properties

Now that you have defined your form, you need a way to pass data back and forth, between the Edit window and the form.

1. If you haven't already done so, save your form.
2. From the **Window** menu select **Edit Window** to make the Edit window come to the front.
3. Make sure it is in Command Mode.
4. After each Circle feature insert a Location dimension for that circle.

5. Near the top, before the Circle features, define variables that you will assign to work with the form. To do this, type this code in the Edit window *before* the measured Circle features:

```
ASSIGN/STR_DESCRIPTION=""
ASSIGN/STR_USER=""
ASSIGN/STR_MEASURE=""
```

6. Place your cursor immediately after these statements, and select **Insert | Report Command | Form**. An **Insert Form** dialog box opens. Navigate to where you stored "TestForm.FORM", select it, and click **Open**.

7. PC-DMIS inserts a `FORM/FILENAME` command block into the Edit window with a pathway to the selected form file. PC-DMIS runs this form when you mark and execute this command. It then waits until you close the form before it continues execution of the Edit window.

> Notice that this command contains a `PARAM/=` statement, or a "parameter". These parameters let you pass values back and forth between the PC-DMIS Edit window variable and the form control properties.

8. Click just before the left side of the equals sign in the `PARAM/=` statement, and type "TXTDESCRIPTION.TEXT". Click on the right side of the equals sign, and type "STR_DESCRIPTION". Press Enter. Another `PARAM/=` statement appears.

> Remember that **txtDescription** is the name you gave the EditBox object in your form that holds a user-entered description of the report.
>
> During execution, the **Text** property of **txtDescription** initially takes the value of whatever `STR_DESCRIPTION` holds. In this case, it would take an empty string. After you close the form, it passes whatever value the form has back into `STR_DESCRIPTION` .

9. Continue to define parameters in this way for both the `STR_USER` and `STR_MEASURE` variables, tying them to the **txtUser** and **txtMeasure** objects' **Text** properties respectively.

10. Save your changes to the Edit window. When finished, your `FORM/FILENAME` command block, should look something like this.

```
CS7         =FORM/FILENAME=D:\PARTPROGRAMS\TESTFORM.FORM
             PARAM/TXTDESCRIPTION.TEXT=STR_DESCRIPTION
             PARAM/TXTMEASURE.TEXT=STR_MEASURE
             PARAM/TXTUSER.TEXT=STR_USER
             PARAM/=
             ENDFORM/
```

Step 7: Add Edit Window Conditional Code to Control MeasurementStep 7: Add Edit Window Conditional Code to Control Measurement

When you created your form, you created a list of option buttons (using the RadioButton object) to control what exactly gets measured. You now need to add conditional statements into the Edit window too, so that the proper feature gets measured according to what gets selected from the form.

1. In the Edit window, place your cursor right before the first CIR1 feature and press Enter. Your cursor should be in a blank line above the CIR1 feature.
2. Select **Insert | Flow Control Command | Control Pairs | If / End If**. PC-DMIS inserts an IF / END IF conditional statement into the Edit window:

   ```
   IF/0

   END_IF/
   ```

3. Define the condition. Highlight the default value of 0 and type:

   ```
   STR_MEASURE=="Circle 1"
   ```

4. Press the Enter key.
5. Select the entire CIR1 feature and the location dimension following it, and select **Edit | Cut**. Then select **Edit | Paste** to move it onto a blank line after the `IF/ STR_MEASURE == "Circle 1"` line but before the `END_IF/` line. Your first conditional block should look like this:

   ```
   IF/STR_MEASURE=="Circle 1"
      CIR1 feature goes here...
      Location dimension goes here...
   END_IF/
   ```

Remember that `STR_MEASURE` , after execution, will hold the value of the **txtMeasure** object's **Text** property. According to the form code, this is either:

"Circle 1", "Circle 2", "Circle 3", or "Circle 4".

This first line checks the variable value of `STR_MEASURE` and if it matches the string value of "Circle 1" then it measures the CIR1 feature. If not, it skips and goes to whatever follows the END_IF/ statement.

6. Continue repeating the above steps, defining conditional statements for the other circle features as well. When finished, your Edit window code should look something like this:

```
            ASSIGN/STR_DESCRIPTION=""
            ASSIGN/STR_USER=""
            ASSIGN/STR_MEASURE=""
CS7        =FORM/FILENAME=D:
\PARTPROGRAMS\TESTFORM.FORM
            PARAM/TXTDESCRIPTION.TEXT=STR_DESCRIPTIO
N
            PARAM/TXTMEASURE.TEXT=STR_MEASURE
            PARAM/TXTUSER.TEXT=STR_USER
            PARAM/=
            ENDFORM/
            IF/STR_MEASURE=="Circle 1"
CIR1       =FEAT/CIRCLE...
DIM LOC1 LOCATION OF CIRCLE CIR1
            END_IF/
            IF/STR_MEASURE=="Circle 2"
CIR2       =FEAT/CIRCLE...
DIM LOC2 LOCATION OF CIRCLE CIR2
            END_IF/
            IF/STR_MEASURE=="Circle 3"
CIR3       =FEAT/CIRCLE...
DIM LOC3 LOCATION OF CIRCLE CIR3
            END_IF/
            IF/STR_MEASURE=="Circle 4"
CIR4       =FEAT/CIRCLE...
DIM LOC4 LOCATION OF CIRCLE CIR4
            END_IF/
```

Step 8: Add Finishing Touches

Now, you need to apply some finishing touches. First, you need to tell PC-DMIS to send the values of our form to the final report in the Report window with report comments. Then you need to make some objects invisible on the form.

1. In the Edit window, type these commands immediately following the `FORM/FILENAME` command block.

```
COMMENT/REPT,
"User: "+STR_USER
COMMENT/REPT,
"Report Description: "+STR_DESCRIPTION
COMMENT/REPT,
"Measure Routine: "+STR_MEASURE
```

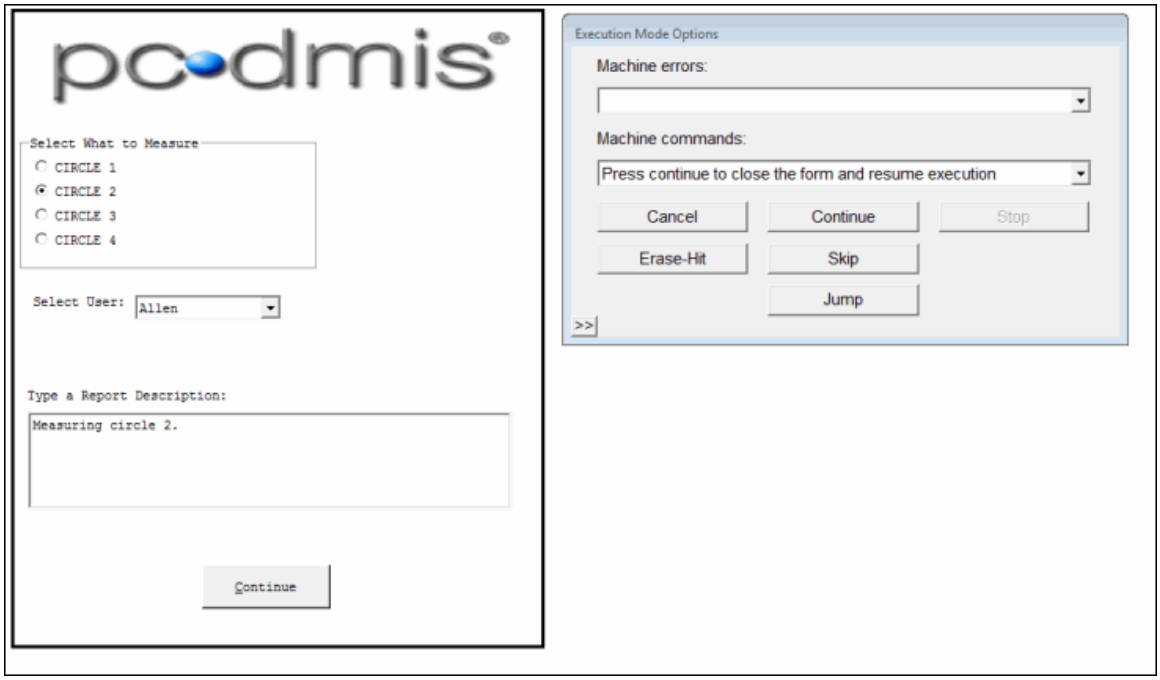
> After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

2. Select **Window | Form Editor** to go back to the Form Editor.
3. Right-click to access the **Properties** dialog box. Select **txtMeasure** from the drop-down list. PC-DMIS selects the object.
4. Click **Advanced** and set the **Visible** property to NO.
5. Select **txtUser** from the drop-down list. PC-DMIS selects the object.
6. Click **Advanced** and set the **Visible** property to NO. Since a user doesn't need to see these values and the only reason we used them was to pass a value back to PC-DMIS, setting this property to NO makes these objects invisible during execution.
7. Save your form.
8. Close the Form editor.

Step 9: Execute the Measurement Routine

1. Select **View | Report Window** and use the **Template Selection Dialog** toolbar icon to set the report to use the default standard report template, TextOnly.rtp.
2. Return to the Edit window. Mark the entire Edit window, save your measurement routine, and then select **File | Execute** to test your measurement routine.

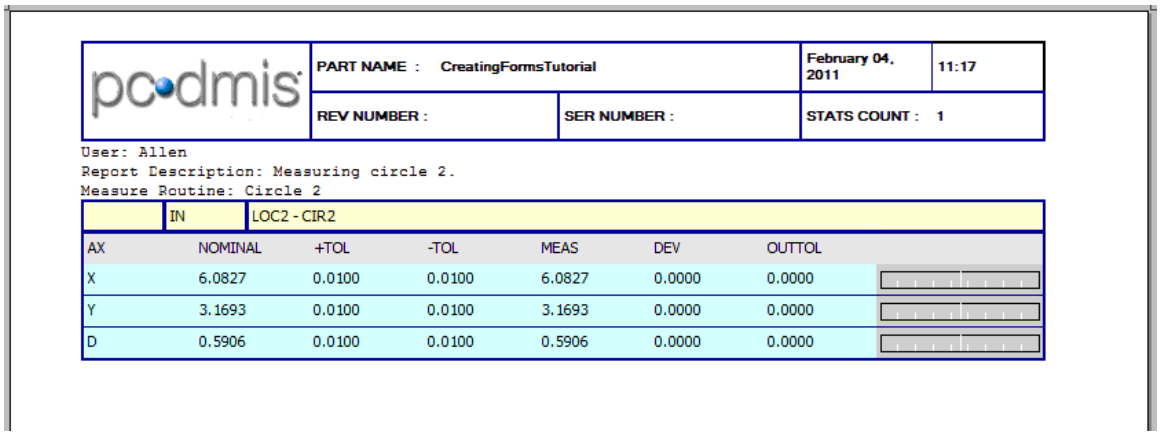
3. When PC-DMIS reaches the `FORM/FILENAME` command, it runs the form and pauses execution until you finish filling it out.



4. Fill out the form and click the **Continue** button. PC-DMIS passes the values from the form back into the PC-DMIS variables.

5. The conditional statements check the value of the `STR_MEASURE` variable and execute the appropriate Circle feature accordingly.

6. PC-DMIS prints the report comments and the measured results for the measured feature to the Report window.



Using Forms with Reports

Forms usually only modify reports indirectly. For example, you may have a form that controls whether or not certain features get executed. In this case, the final report isn't modified directly, but indirectly because reports always show what gets executed anyway.

You can however use the `FORM/FILENAME` command to pass parameters back and forth between the Edit window and a form's objects, and use Comments to cause those parameters to appear in a final report. See the "Creating Forms" tutorial and the "Inserting a FORM Command" topic for examples of this type of forms use.

Additionally, after getting the values from a form into the Edit window, you can also use the `REPORT/TEMPLATE` command to pass parameters from the Edit window into a report template in the same way that you use the `FORM/FILENAME` command.

Select **Insert | Report Command | Template Report** to insert a `REPORT/TEMPLATE` command, then assign parameter values to modify object properties in the report template. See the "Creating Forms" tutorial on how to do this, using the `FORM/FILENAME` command as a guide, but instead of selecting a .FORM filename, select a Report Template (.rtp filename). Also, see the "Embedding HyperView Reports or Report Templates in a Measurement Routine" topic.

Creating Custom Reports

Custom reports provide a flexible and easy-to-use approach to creating reports for your measurement results. This is most useful when you need to generate a report quickly and simply for a specific measurement routine, but you don't need the power available to the template-based approach. Since custom reports do not use any report template, and instead utilize your current measurement routine's data directly, custom reports are generally easier to create and customize, but they lack the power and scope of template reporting.

Advantages of Custom Reporting:

- You can place the data anywhere on the page and in any order.
- You can combine the data from multiple commands as a single element on the report.
- You can generate the reports by unique drag-and-drop method.
- The report editor uses the actual data from the measurement routine, not the dummy data. This makes customizing the report much simpler.

Disadvantages of Custom Reporting:

- You are creating a single report, not a template, and this report is tied to your measurement routine. While you can import the structure of the report to use with other measurement routines, the reusability is not as great as with report templates designed with specific rules.
- It isn't as extensible as template reporting. Suppose you add a new feature or dimension later to your measurement routine. For it to show up, you will need to drag and drop the new item into your report editor.

The following topics provide you with a tutorial that walks you through how to create, view, and print your first custom report. Procedural topics will also be provided for quick access should you need them later.

About the Custom Report Editor

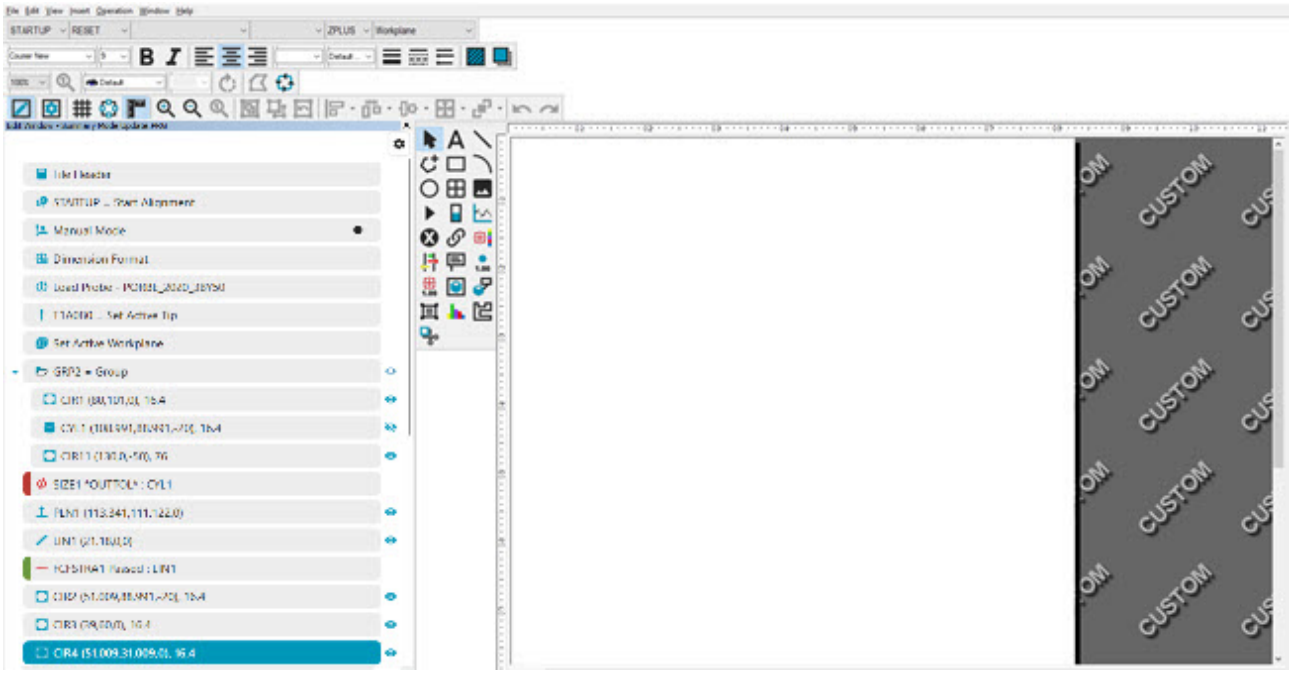
To access the Custom Report Editor, select **File | Reporting | New | Custom Report**. To open an existing custom report for your current measurement routine, select **File | Reporting | Edit | Custom Report**.

You can access a slimmed down version of this editor from the Report window's **Reporting** toolbar. This lets you edit a report without leaving the usual PC-DMIS menu structure.

You can access this editor through either of these options:

- Click **Custom Report selection dialog** ( ) and then click **Blank Report** and then **Open**.
- Click **Edit report** ( ).

PC-DMIS displays the Custom Report Editor with the Edit window placed into Summary Mode next to it.



Custom Report Editor with the Summary Mode to its left

This editor is similar to the Report Template Editor, Label Template Editor, and Form Editor. With the exception of some differences in its **Object Bar**, the user interface is essentially the same.

The custom report is the white and sizable area on top of the "CUSTOM" background. Note that when you use this editor, you do not create a template, but create the actual report that appears in the Report window. When you create a new custom report, this editor opens side by side with the Edit window set in Summary Mode. This allows you to drag the desired objects from Summary Mode onto your Custom Report. See "Creating Custom Reports".

With the Custom Report Editor, you cannot place the Report or Label template editors into Run mode as you did with the HyperView editor. The Custom Report Editor only opens in Edit mode and cannot be switched to Run mode. Run mode functions only with the HyperView editor or the Form Editor.

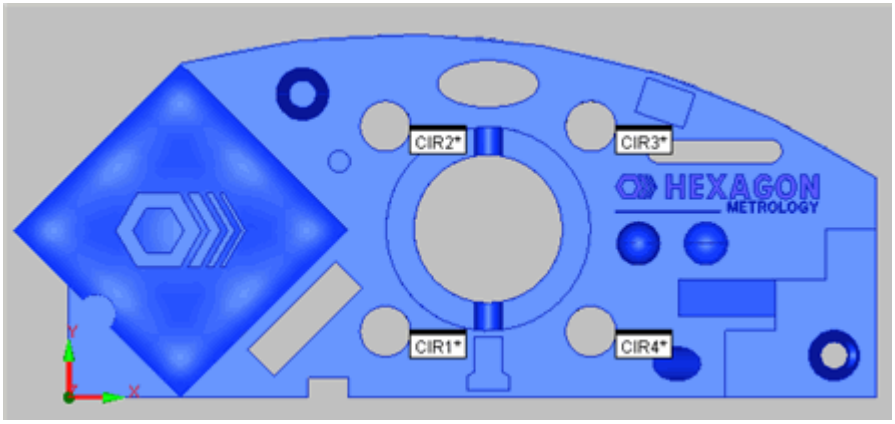
The editor contains the following elements:

- The Menu bar
- The Font Bar (toolbar)
- The Object Bar (toolbar)
- The Layout Bar (toolbar)
- Object Property Sheets

Tutorial - Creating a Custom Report

This topic walks you through a very basic tutorial that will result in you creating a simple custom report. This should give you a basic overview of how to create custom reports within the Custom Report Editor and how they interact with existing label templates so you can later create and use your own custom reports.

Before you begin this tutorial, create a simple measurement routine that has four measured circles on a simple part and four Circularity dimensions, one for each circle. This tutorial uses the Hexagon test block (Hexblock_Wireframe_Surface.igs).



Example showing the four circles.

Step 1: Setting Up the Work Environment

In this tutorial you will use the Edit window in Summary Mode along with the Custom Reporting Editor.

1. Select **File | Reporting | New | Custom Report** from the menu bar to show both the Custom Report Editor and Summary Mode next to it.

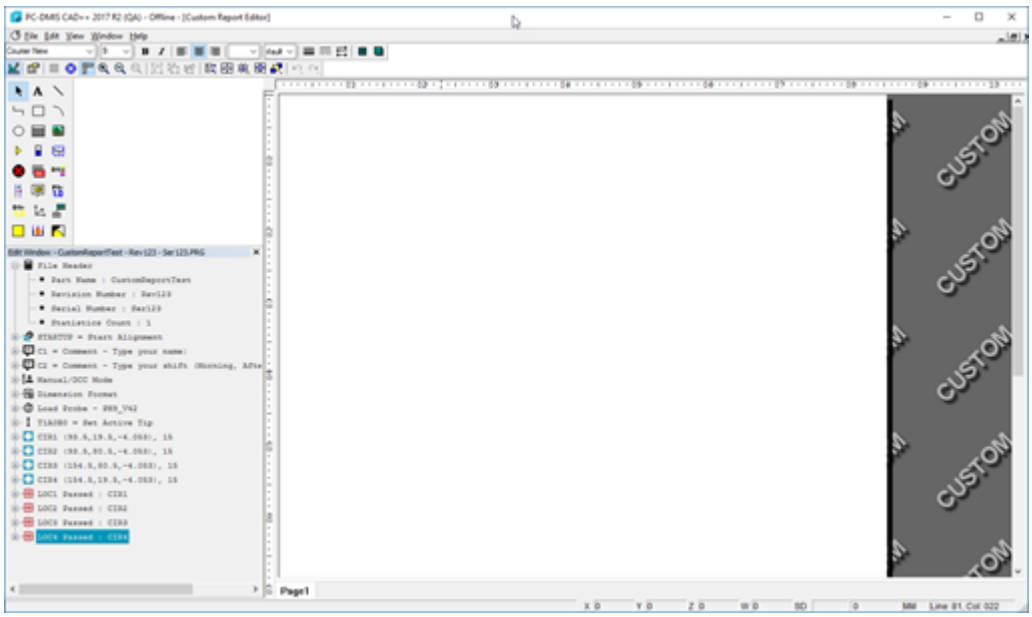
   > You can also create a new report from the **Custom Report Selection dialog** icon ( ) on the Report window's **Reporting** toolbar. From the dialog box that appears, click **Blank Report** and then **Open** to create the blank report.

2. To hide any unused toolbars, right-click on the toolbar area and remove them.

3. To hide any unused PC-DMIS windows, select the open window from the **View** menu. Keep the Edit window open.

4. To maximize the Custom Report Editor, click the maximize button in the upper right corner of the editor's window. You should see the words "CUSTOM" on the background of the editor.

5. Drag the Edit window underneath the Custom Report Editor's **Object Bar**. Your work environment should now look something like this:



When working with the Custom Report Editor, you may find it helpful to hide your usual PC-DMIS toolbars and windows, thereby freeing up some screen space. If you work frequently with this editor, you may want to create a stored screen layout for it. For information on layouts, see the "Window Layouts Toolbar" topic in the "Using Toolbars" chapter.

Step 2: Dragging, Dropping, and Positioning Objects

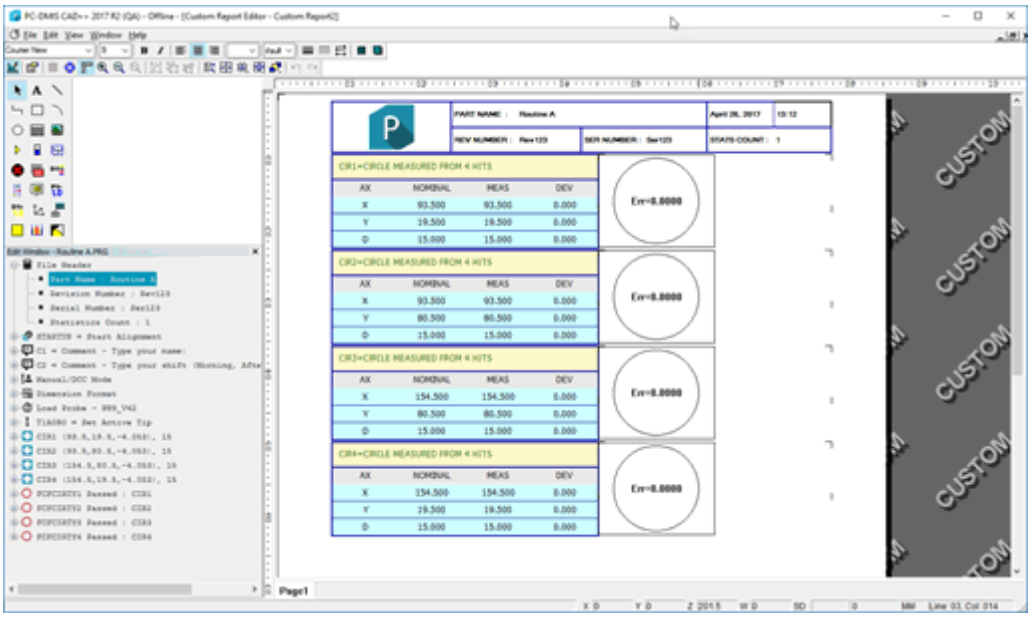
In this step, you will drag the report items into the Custom Report Editor.

1. From the Edit window in Summary Mode, select **File Header**, and drag it out onto the editor. You will notice a transparent image of that item's icon as you drag it out onto the work area.
2. Release the mouse button. PC-DMIS creates a File Header object in the editor.
3. Drag and drop CIR1 and CIR2 onto your report. Don't worry about positioning them properly. For now, just drop them into any empty space on the first page of the report.
4. In the Report Editor, select the **File Header** object already in your report, and drag it to a place near the top of the report and center it horizontally on the page.
5. Next, select the label object for CIR1, and drag it so that the top edge is just under the bottom edge of the File Header object. Try to align their left sides as well.
6. Repeat this step for CIR2.
7. Now, drag the CIR3 feature from the Edit window to the bottom of the label used for CIR2. Notice that as you move your mouse over the various labels already in the editor, green handles appear around the labels. When the blue arrow appears just below CIR2, release the mouse. The feature is dropped into the editor below CIR2, and the label object for it becomes automatically aligned with the object above it.

8. Repeat the above step for CIR4, attaching it just below CIR3.

9. Select **File | Save**. A dialog box will appear, which allows you to save your report. Choose any name and click **Save**.

Your Custom Report Editor should look something like this:



Step 3: Using Multiple Pages

In this step, you will work with multiple pages. You will add two additional pages and modify one page to support a different page size. Then you will rearrange the pages.

1. In the Report Editor, at the bottom of the page, right-click on the **Page1** tab and select **Add Tab**. A new page appears, called **Page2**.

2. Right-click on that page. The **Properties** dialog box appears.

3. Change **Height** to 850, and press the Tab key.

4. Change **Width** to 1100, and press the Tab key. Changing these two properties essentially formats the page for Landscape printing.

5. Create a third page, **Page3**.

6. Right-click on **Page2**, and click **Move Right**. Notice how this moves the **Page2** tab to the right of **Page3**.

7. Right-click on **Page3**, and click **Move Left**. Your pages are now reordered to show **Page3**, followed by **Page1**, followed by **Page2**. In this way, you can rearrange your pages.

8. Save your report.

Your report now has three pages, and their order has been changed.

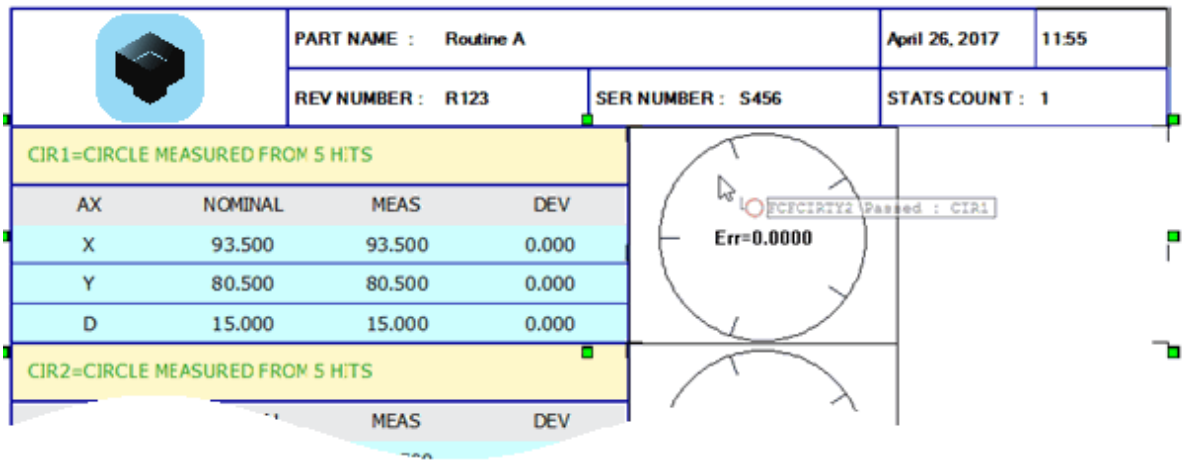
Step 4: Dropping onto Other Objects

This step demonstrates how you can replace existing label objects with new label objects and how to use the **CADReportObject** inside a custom report.

1. In the Custom Report Editor, select the **Page1** tab. To replace any object in your report, drag and drop any other item of a similar type on top of it. For example, you can drop any item that uses labels on top of any existing label in your editor.

2. Select the label in your report for the CIR1 feature. You should already have four Circularity dimensions. If not, create them now, one for each circle feature.

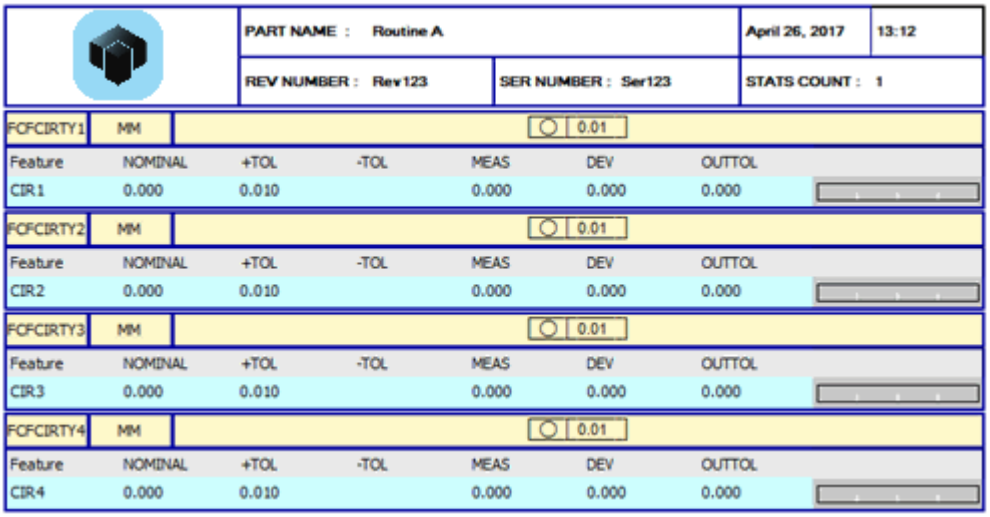
3. Drag the Circularity dimension for the CIR1 feature from your Edit window and drop it on top of the feature label already in your report editor for CIR1, like this:



Example of the label object updated with a new label object.

4. If the updated object moved, reposition it.

5. Drag the other dimensions on top of their respective feature labels in the editor. PC-DMIS updates all the labels accordingly, so that they look something like this:



	PART NAME : Routine A				April 26, 2017	13:12
	REV NUMBER : Rev123		SER NUMBER : Ser123		STATS COUNT : 1	
FCFCIRTY1	MM			0.01		
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CIR1	0.000	0.010		0.000	0.000	0.000
FCFCIRTY2	MM			0.01		
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CIR2	0.000	0.010		0.000	0.000	0.000
FCFCIRTY3	MM			0.01		
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CIR3	0.000	0.010		0.000	0.000	0.000
FCFCIRTY4	MM			0.01		
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
CIR4	0.000	0.010		0.000	0.000	0.000

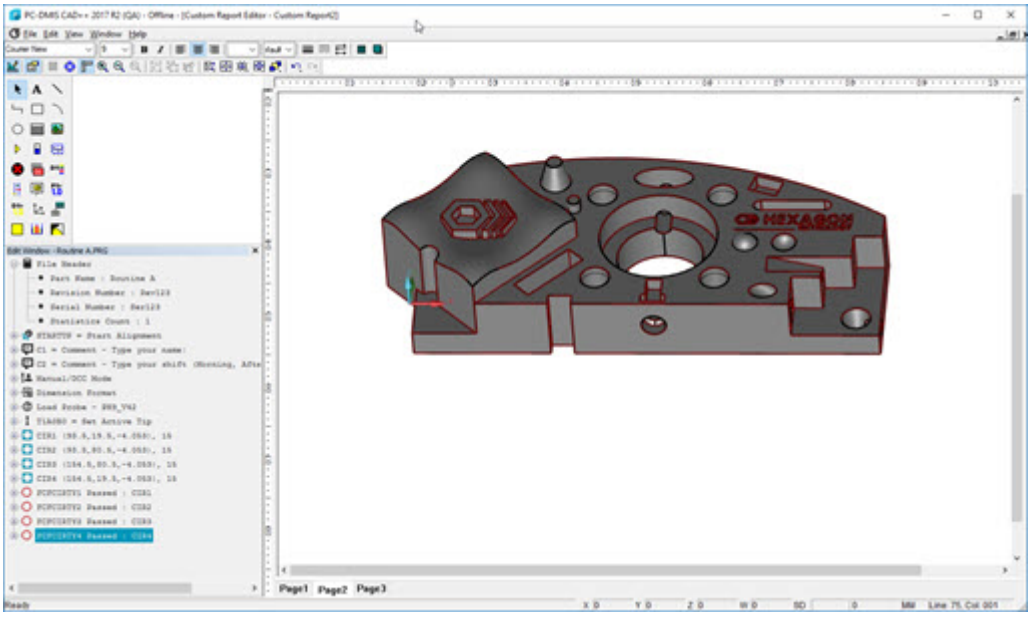
Report Editor showing the four dimension labels.

6. Now, click on the **Page2** tab. Drag a **CADReportObject** from the **Object Bar** onto that page, and size it so it fills the page.

7. Drag the four dimensions one at a time on top of the **CADReportObject**. For this object, notice PC-DMIS does not replace it with a label object for the dimension. Instead, PC-DMIS automatically creates the appropriate labels and leader-lines on top of the **CADReportObject** for the dimensions.

8. Save your report.

Page1 now has dimension labels instead of feature labels, and the **CADReportObject** in **Page2** should look something like this:

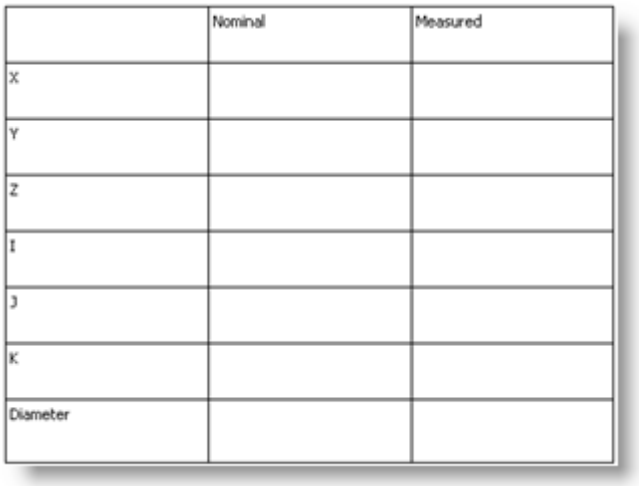


Step 5: Inserting Data Items

In this step, you will insert a **GridControlObject** and get it ready to display values from the Edit window.

1. Click on the **Page3** tab.

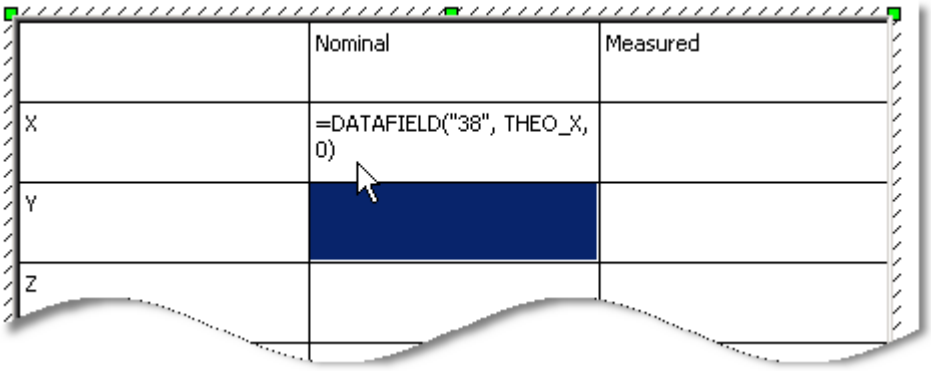
2. Click on the **GridControlObject** icon, and drag the object onto the page.
3. Access the **Properties** dialog box, and set **NumRows** to 8 and **NumCols** to 3.
4. In the first row, the middle cell, double-click, and then type "Nominal". In the right cell, do the same, and type "Measured".
5. Starting with the first row, and first column, and then moving down through the other rows, type "X", "Y", "Z", "I","J" ,"K", and "Diameter". This fills in rows 2 through 8 in column 1. Your GridControlObject should look like this:



A GridControlObject with Static Text

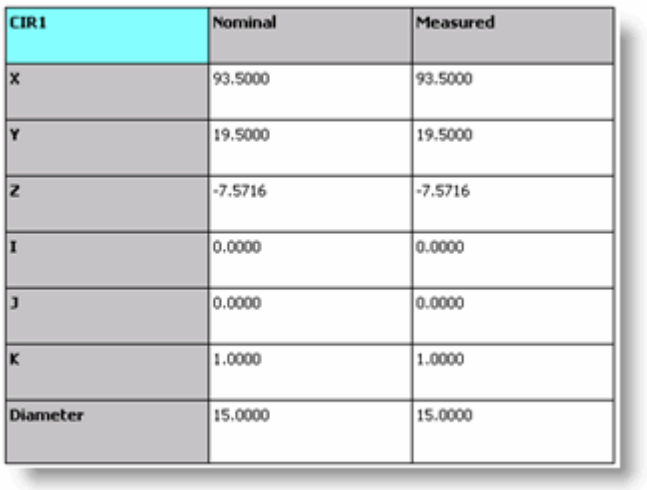
> Remember when typing information into a cell, you must click in another cell or press the Tab key for the value to actually be displayed.

6. Double-click on the item to type the expression in the cell. This cell dynamically displays the data which means the information is not hard coded. If the value changes, PC-DMIS updates the data in your report to match.



Example of the data item's expression.

7. Finally, apply some text and background formatting of your choice to row 1 and column 1, and then save your report. The **GridControlObject** should look similar to this:



CIR1	Nominal	Measured
X	93.5000	93.5000
Y	19.5000	19.5000
Z	-7.5716	-7.5716
I	0.0000	0.0000
J	0.0000	0.0000
K	1.0000	1.0000
Diameter	15.0000	15.0000

A Sample GridControlObject containing data items.

8. Select **File | Close** to close the Custom Report Editor.

   Your values may differ based on the part and circles measured.

Step 6: Viewing, Updating, and Printing your Report

This final step explains how to load your custom report into the Report window, how to view it, how to update a report from a measurement routine that changes, and finally how to print it.

1. Select **View | Report Window** to access the Report window.
2. From the Report window's **Reporting** toolbar, select **Custom Report Selection Dialog** . A dialog box appears and shows all custom reports.
3. Select your report and click **Open**. Your report appears in the Report window.
4. Next, you will update your report. Select **File | Reporting | Edit | Custom Report**. A dialog box appears and shows all the reports you've created for your current measurement routine.
5. Select your report and click **Open**. Your report opens inside the Custom Report Editor.
6. Go ahead and make any change to your report in the editor, and then save your report again.
7. To get the newly-updated report to appear in the Report window, re-execute your measurement routine, or click the **Redraw** icon from the **Reporting** toolbar.
8. Finally, you need to print your report. Select the **File | Printing | Report Window Print Setup** menu item. The **Output Configuration** dialog box appears.
9. Select the **Report** tab. In the **Output Options** section, mark the **Printer** check box to send the report to your printing device.

10. From the Report window's **Reporting** toolbar, click the **Print** icon. PC-DMIS prints your report.

In this step, you've loaded an existing report into the Report window, updated it, and then sent it to your printer.

Congratulations! You've successfully completed the Creating a Custom Report tutorial.

Custom Report Generation

To create a new custom report, follow these steps:

1. Select **File | Reporting | New | Custom Report** to open the **Custom Report Editor** and show the Edit window in Summary Mode.
2. Drag items from the Edit window and drop them into the editor.

> PC-DMIS automatically uses the labels defined by the current **Page** object's rule set to display the objects. If you drop an object and it doesn't have an associated label, an **Open** dialog box appears so that you can choose a label defined for the dropped item.

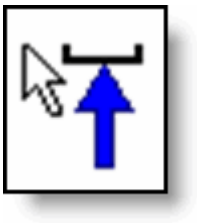
3. Add and configure additional objects from the editor's **Object Bar** as needed.
4. Position report elements as desired.
5. Select **File | Save** to save your report. A dialog box opens to allow you to specify the name of your report.

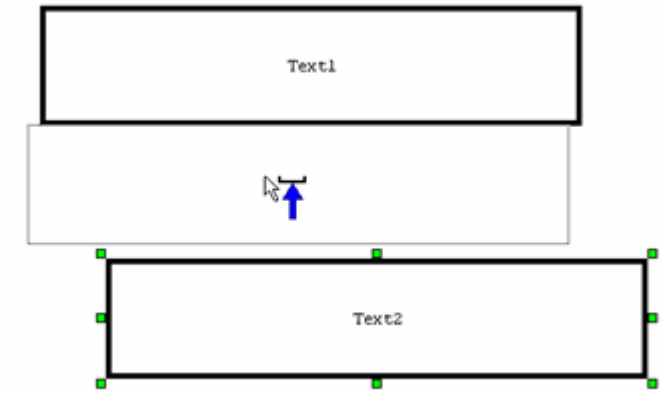
> Another way you can create a new custom report is to use the **Reporting** toolbar. (For information on the Reporting toolbar, see "Reporting Toolbar".)
>
> 1. From Report window's **Reporting** toolbar, click the **Custom Report Selection dialog** button ( ).
> 2. Double-click on **Blank Report** to create a new blank custom report.
> 3. Edit the report as usual.
> 4. Click the **Edit report** button ( ) to close edit mode and follow the prompt to save your report.

Positioning Report Objects

Inside the Custom Report Editor (**File | Reporting | New | Custom Report**), you can position objects by dragging objects where you want them or by using the appropriate alignment icons from the editor's **Layout Bar**.

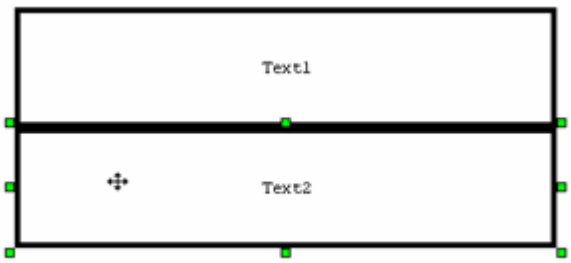
In addition, PC-DMIS provides a useful tool to essentially snap an object below an object above it. To do this, slowly drag an object so that its top and left edges are more or less aligned with another object's bottom and left edges. Your mouse pointer will change to include a small blue arrow: This arrow indicates that the object's left side you are positioning will be aligned with the object's left side above it.





Example showing the object's alignment using the blue arrow.

When this arrow appears, you can release you mouse button, and the object you are dragging will align itself to the other object:



Example showing the aligned objects.

The blue arrow can also help with dropping an object precisely under another object. Simply drag the object over an existing object until green handles appear around the existing object, then drag the mouse slightly under that object until the blue arrow appears.

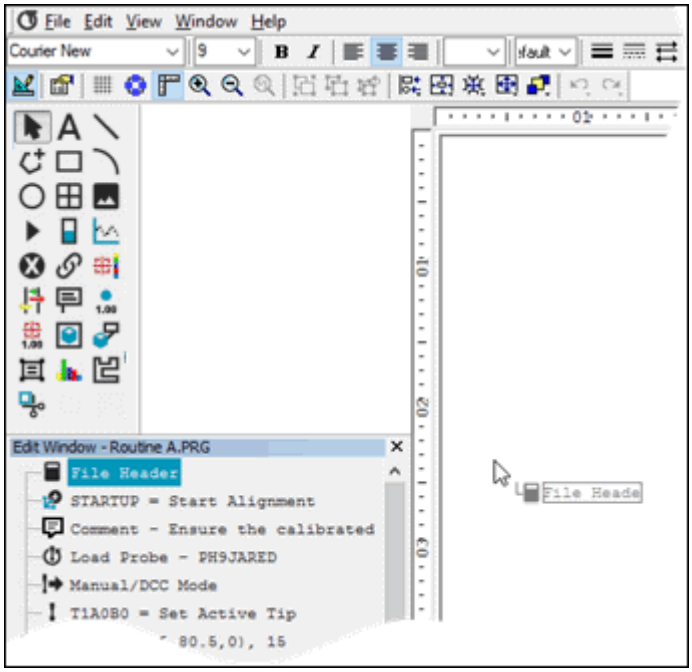
Release the mouse when it appears, and the dropped object will appear aligned to the other object.

This allows you to create a list of objects without white space in between, useful when aligning a list of labels containing your feature or dimension data.

Dragging and Dropping Information into a Custom Report

As the "Custom Report Generation" topic describes, you can drag features and other items from the Edit window in Summary Mode into the Custom Report Editor (**File | Reporting | New | Custom Report**).

You cannot drag the command properties from the Edit window in Summary Mode into the Custom Report Editor.



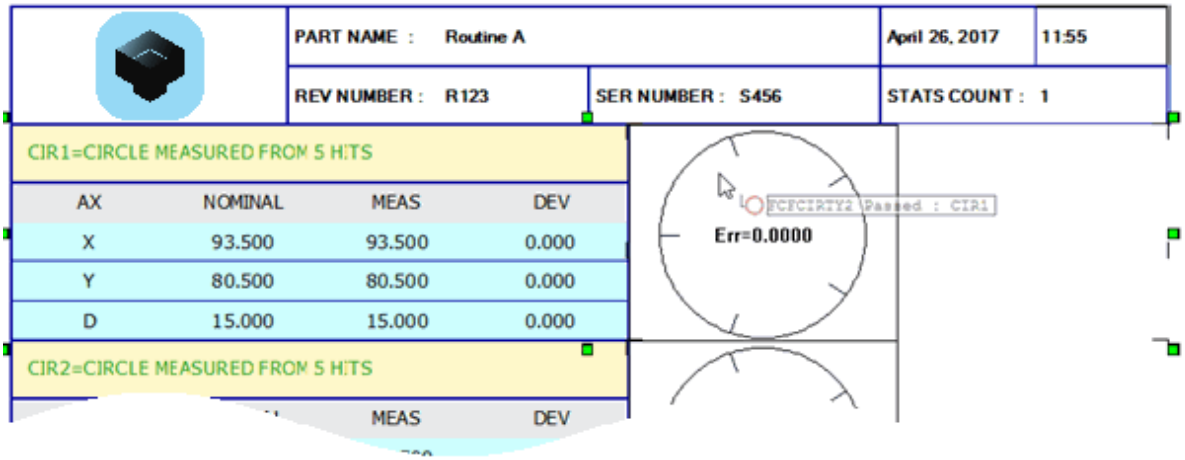
Example of dragging a File Header object onto the editing area (the Page object)

When you drop an item into the editor, the appropriate label for that feature, as defined by the **Page** object's **Rule Tree Editor**, automatically gets created:

Example of a dropped File Header object

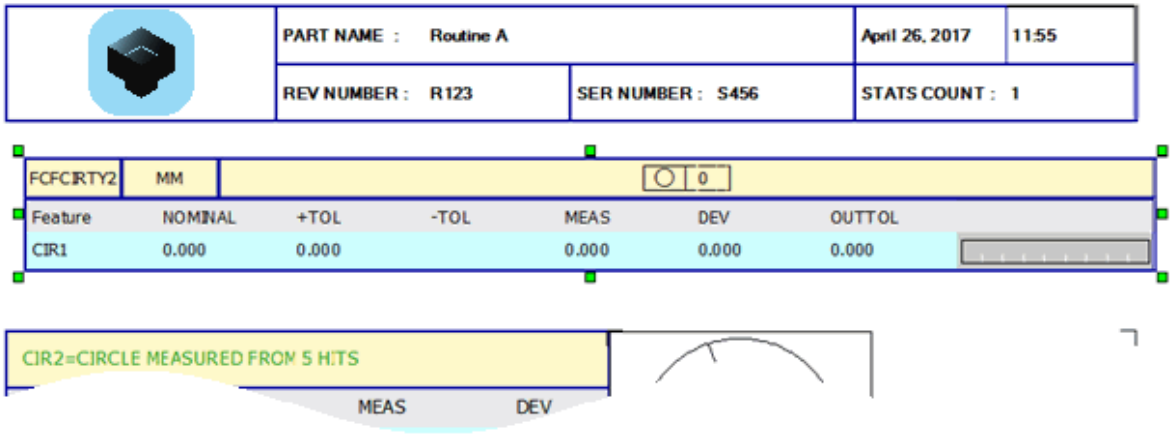
Replacing Information by Dropping on Top of Existing Labels

If you drag an item on top of an existing label, like this:



Example of dropping a Dimension on top of an existing label

PC-DMIS replaces the existing label with the dropped item:



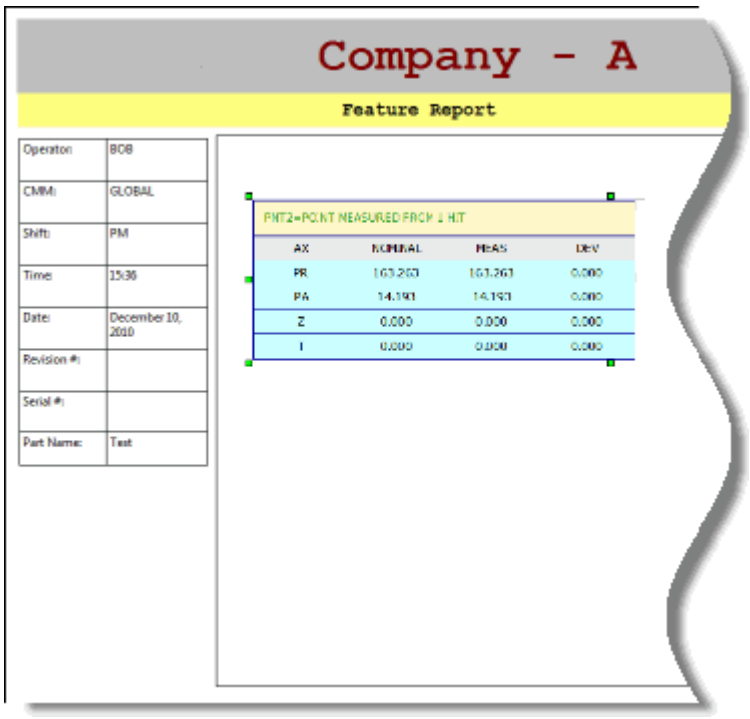
Example of the updated label

Dropping a Floating Label by using the Alt Key

If you drag an item on top of an existing label while you press the Alt key, like this:

Example of Dragging on top of a Large Page-Size Label

PC-DMIS does NOT replace the information in the label with what you dropped. Instead, it floats the new label on top of the existing label.

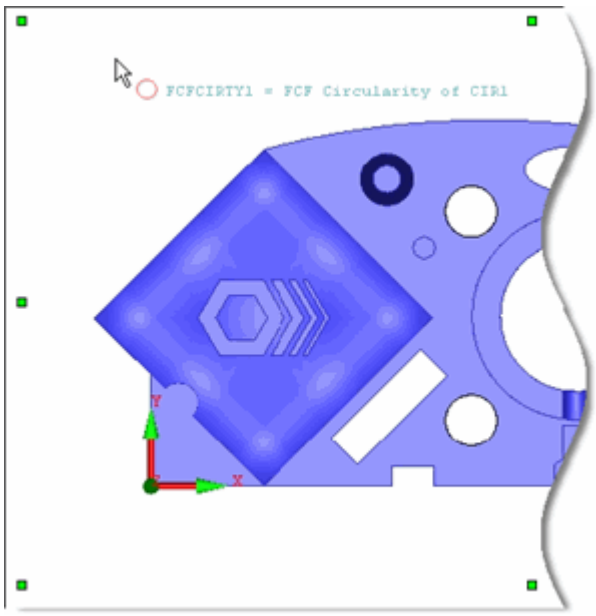


Example of dropping onto a large page-size label with Alt key pressed

You may find this useful if you have a large custom label that covers much of the page, and you want to place new labels on top of that label instead of updating it.

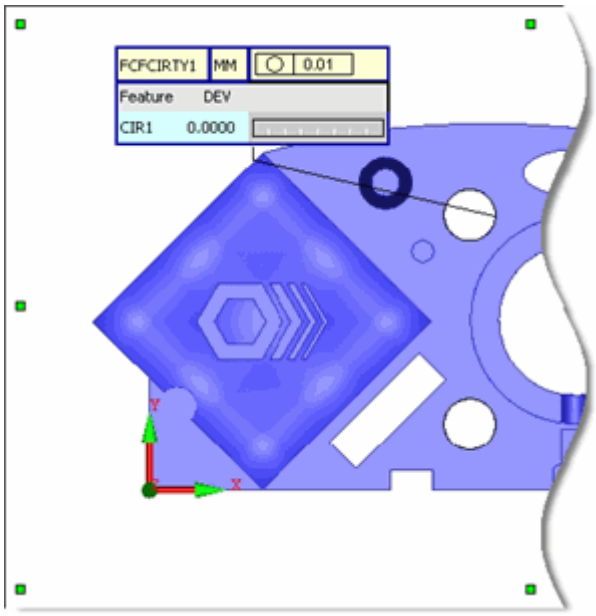
Dropping Feature or Dimension Data on a CADReportObject

If you drag a feature or dimension on top of a **CADReportObject** (CRO), like this:



Example of dropping a Dimension on top of a CRO

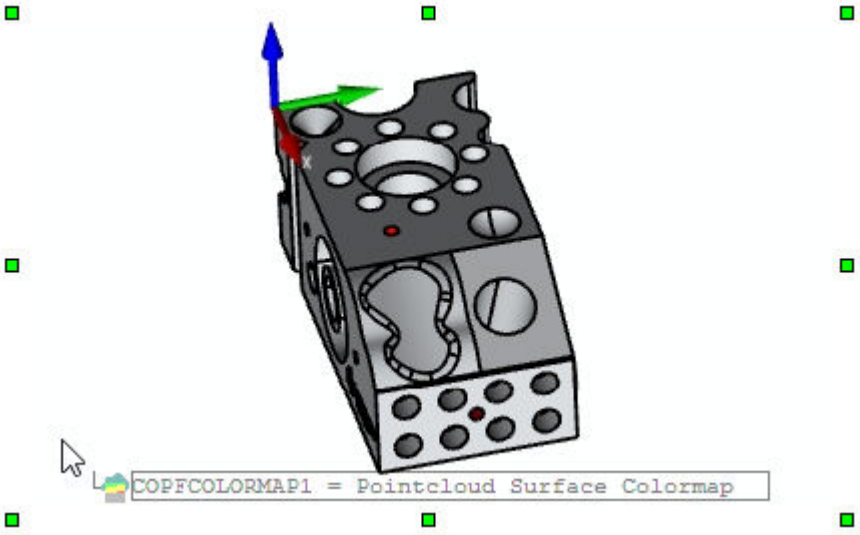
PC-DMIS adds the appropriate label object and leader-line for the dropped object on top of the CRO. The label that PC-DMIS shows depends on the label specified in the CRO's Rule Tree Editor and not the Rule Tree Editor for the **Page** object.



Example of a dropped dimension on top of a CRO

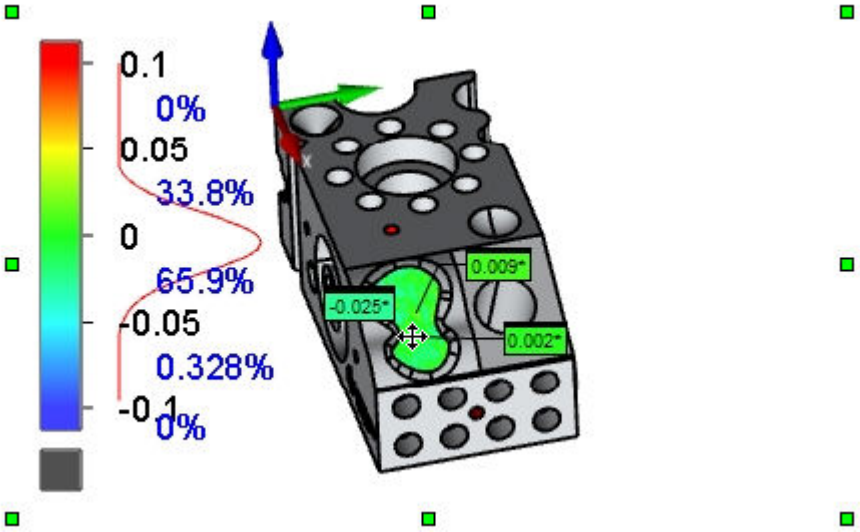
Dropping Colormaps on a CADReportObject

If you drag a colormap on top of a **CadReportObject** (CRO), like this:



Example of dropping a surface colormap onto a CRO

PC-DMIS displays the colormap on the CRO:

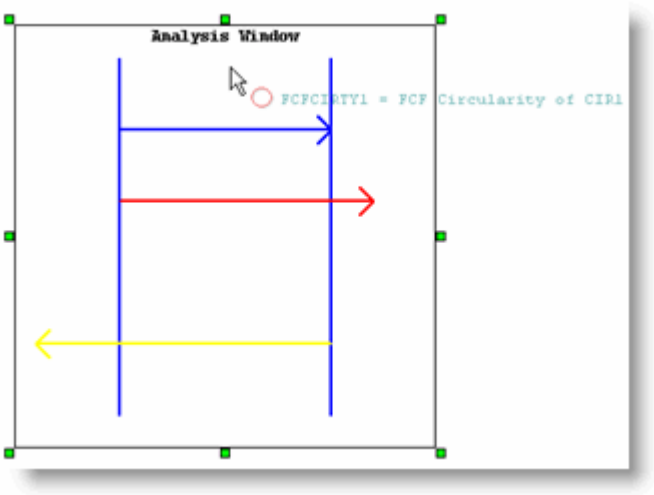


Example of dropping a surface colormap onto a CRO

For more information on using colormaps with the CRO, see "Colormaps and the CadReportObject".

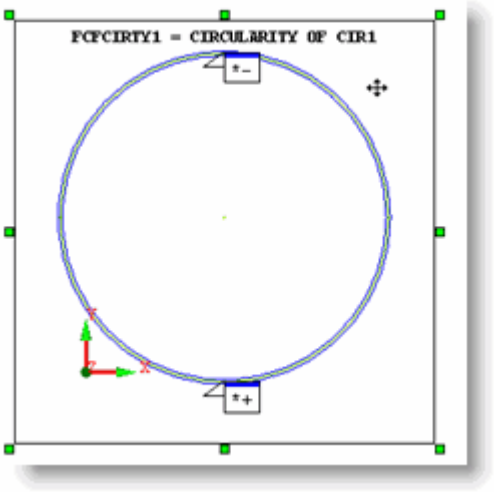
Dropping Data on an Analysis Object

If you drag a dimension on top of a **Analysis** object, like this:



Example of dropping a Dimension on top of an Analysis object

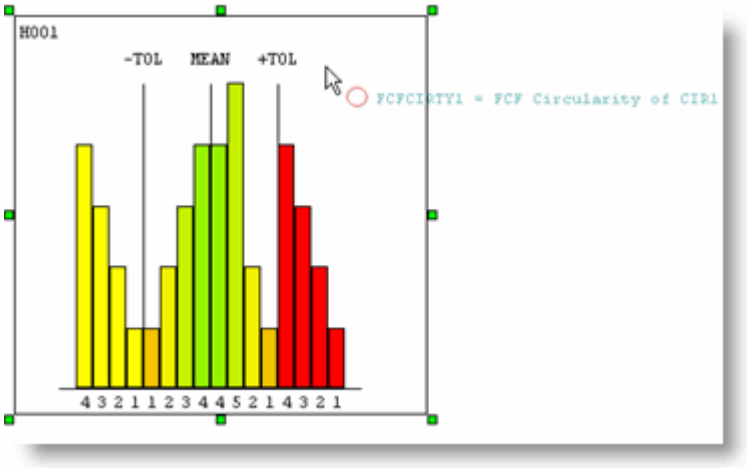
PC-DMIS displays the graphical analysis information for the dropped dimension inside the **Analysis** object.



Example of a dropped Dimension on top of an Analysis object

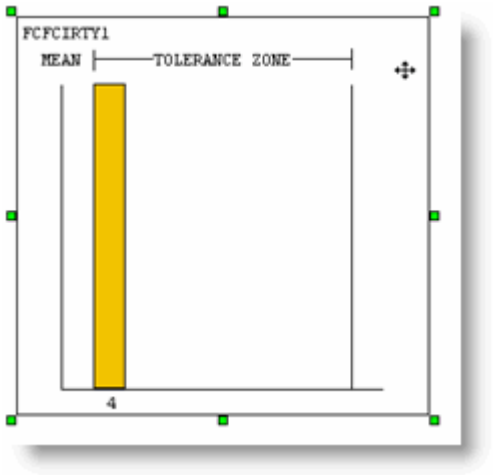
Dropping Data on a DimensionHistogram Object

If you drag a dimension on top of a **DimensionHistogram** object, like this:



Example of dropping a Dimension on top of a DimensionHistogram object

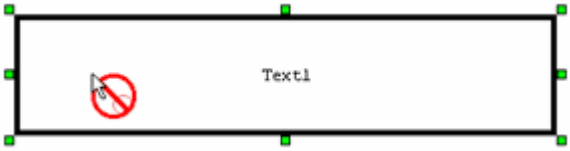
PC-DMIS displays the histogram information for the dropped dimension inside the **DimensionHistogram** object.



Example of a dropped Dimension on top of a DimensionHistogram object

Invalid Drop Areas

If you try to drop an item on top of an invalid object (for example if you drag a feature or dimension on top of a Text object), PC-DMIS changes your pointer to display a red "not allowed" symbol to show you that you cannot drop that item there.



Example of an invalid drop area for a dragged dimension

Using Shift and Ctrl Keys When Dropping

While you perform the drop of a command (such as a feature or dimension), if you hold down the Shift or Ctrl key and release the button, PC-DMIS does the following:

- Shift key - PC-DMIS inserts a CommandTextObject for that item. This lets you display the information for that item in a textual, non-table format.
- Ctrl key - PC-DMIS displays an **Open** dialog box, which enables you to select a different label template for the item.

When performing a drop of a data item, if you hold down the Ctrl key, PC-DMIS displays the evaluated expression for the data item as well as the descriptive string of text from the Summary Mode that precedes it.

Dropping Multiple Items

You can drag multiple items at once from the Edit window in Summary Mode. This is useful if you want to quickly add several items into your report.

- To select an entire list of consecutive items, click the first item, press Shift on the keyboard, and click the last item. All items between them are selected.

- To select or deselect individual items to or from your existing selection, press and hold the Ctrl key while you click on items.

Once you have selected a list of items, drag them into the editor.

Using Rules

Like template reports, custom reports use the **Rule Tree Editor** to determine what label templates should be used in your report. A default set of rules is automatically included, so you only need to change the default rules if you want to load some sort of custom label template.

To work with these rules,

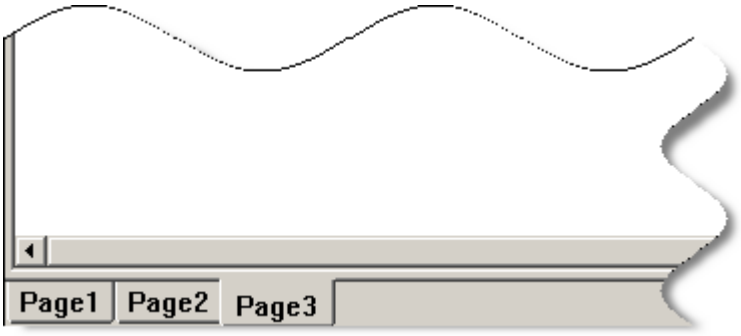
1. Right-click on the editing area (the **Page** object) in the **Custom Report Editor** (**File | Reporting | New | Custom Report**).
2. Select **Properties** from the small popup menu to open the **Properties** dialog box.
3. Click on **Rule...** in the **Rule Tree Editor** property. The **Rule Tree Editor** appears.
4. Modify the rules as desired.

For information on accessing and using the Rule Tree Editor, see "About the Rule Tree Editor".

Working with Multiple Pages

When you create a new custom report using the **Custom Report Editor** (**File |Reporting | New |Custom Report**), the editing area is blank. No objects exist in the editing area except for the editing area itself. The editing area is actually a **Page** object, and its properties can be modified as with any other object. Simply right-click on the object and select **Properties**.

Similar to **Section** tabs in the Report Template Editor, you can create multiple **Page** tabs (**Page** objects) in the **Custom Report Editor**. To do this, right-click on the tab and select **Add Tab** from the popup menu. An additional tab appears at the bottom of the editing area:



Multiple Page Tabs

This will create additional reporting pages that will later appear in the Report window. While similar to **Section** tabs, **Page** tabs reflect a single additional page in your final report, whereas a single **Section** tab may actually end up displaying multiple pages in the final report depending on any defined rules for objects in that section and the length of your measurement routine.

You can size your Page objects to meet different needs. For example, you can modify one page's properties to display its contents in a standard Portrait format and another page's properties to display its information in a Landscape format or to match printer or page settings for a specific locale.

You can easily reorder pages as well. To do this, right-click on a **Page** tab, and select either **Move Right** or **Move Left**. The tabs at the bottom of the Editing area will be reordered accordingly.

From the Report window's preview pane, you can also create and reorder pages. From that pane, you can drag and drop objects to move them from one page to another. For more information, see "Working with the Preview Pane".

Viewing and Printing Custom Reports

Custom reports are viewed and printed using the Report window.

To view your report, follow these steps:

1. Access the Report window (select **View | Report Window**).
2. From the Report window toolbar, select the **Custom Report Selection Dialog** icon . A dialog box appears showing all custom reports.
3. Select your report and click **Open**. Your report appears in the Report window.

To print your report, follow these steps:

1. Define your output using the **File | Printing | Report Window Print Setup** menu item.
2. Either execute your measurement routine or click the **Print** icon from the **Reporting** toolbar.

Editing or Deleting Custom Reports

Custom reports are not stored as a typical file. Instead, they are stored as part of the measurement routine.

Editing a Custom Report

1. In PC-DMIS, select **View | Report Window** to select the Report window.
2. From the Report window, click on the **Custom Report Selection Dialog** icon ( ).
3. Select the report to edit.
4. From the **Reporting** toolbar, click **Edit report** ( ) to place the report in edit mode. When the report is in edit mode, PC-DMIS shows the Edit window in Summary Mode. PC-DMIS also enables or shows the following reporting tools:
   - The **Save** ( ) and **Save As** ( ) buttons on the **Reporting** toolbar
   - The Font bar
   - The Layout bar
   - The Object bar
   - The **View | Report Controls** submenu
5. Use the Edit window and the above reporting tools to edit the report.
6. Click **Save** ( ) or **Save As** ( ) to save your changes.
7. Click **Edit** ( ) again to leave edit mode and restore PC-DMIS to its previous state.

Alternately, you can also edit your custom report through the older **File | Reporting | Edit | Custom Report** menu item.

Deleting a Custom Report

Because custom reports are stored in the measurement routine, you have to delete them within PC-DMIS, not through Windows Explorer.

1. In PC-DMIS, select **View | Report Window** to select the Report window.
2. Make sure the report is not in edit mode.
3. From the Report Window, click on the **Custom Report Selection Dialog** icon ( ).
4. Select the report to delete.
5. Press the Delete key on your keyboard.

Using a Custom Report from another Measurement Routine

You can use a Custom Report from another measurement routine in your current measurement routine to a certain extent.

To do this,

1. Select the **File | Reporting | Edit | Custom Report from Other Measurement Routine** menu item. An **Open** dialog box appears showing all your measurement routines.

2. Select your measurement routine and click **Open**. A **Custom Report** dialog box appears. If a report exists for the selected measurement routine, it will show up in this dialog box.

3. From the dialog box, select the report you want to use and click **Open**. PC-DMIS loads the report in the Custom Report Editor.

If a feature or item is not found in the measurement routine, the label or object will be blank.

About Report Expressions

Report Expressions are special commands that you place inside of report and label template objects to pull specific data from PC-DMIS and place them into those objects. For example, suppose that you want to insert a feature ID into a label template. You would add an object that supports expressions into your report, such as the **GridControlObject**. Then, inside of an Expression Cell of the grid, you would type "=ID".

There are four areas where you can insert report expressions:

- The Rules Tree Editor - in the **Conditional Expression** and **Text Expression** boxes
- The GridControlObject - in its grid cells
- The GridControlObject - in the **Repeat Expression** box in the **Row** tab for repeat expressions
- Property Sheet Values - on edit fields or combo boxes that take text values

Consult those topics for information on where to insert report expression code.

For available expressions, consult "Functions and Operators" and "Using Data Types to Find a Report Expression" for lists of available functions, operators, and data types.

Reporting expressions can also use many normal PC-DMIS expressions. Precede the expressions with the "=" sign when you type them in the cell. Expressions are discussed in the "Using Expressions and Variables" chapter.

Functions and Operators

The following are lists of functions and operators available to the Reporting expression language. These work the same as the PC-DMIS expression language.

The Reporting expression language does not support variables, structures, or functions like the PC-DMIS expression language does. In place of variables, a new type called DATA_TYPE has been added to the language. See "Using Data Types to Find a Report Expression" for more information. Another new difference in the Reporting language is the addition of a set of constants described in "Predefined Constants".

Remember to precede your expression with an equals sign (=). Also, make sure that the command from which you are getting your data supports the expression you want to use.

() Parentheses are used to group expressions and determine evaluation order.

Functions for Report Expressions

Items preceded with an asterisk symbol (*) are unique to the reporting expression language.

`ABS (<expression>)`
This function returns the absolute value of the input value.

`ACOS (<expression>)`
This function returns the arc-cosine for the input value. Input and result are in radians.

`ANGLEBETWEEN (<expression1>, <expression2>)`
This function returns the angle between the two inputs expression1 and expression2, which should be of type vector. Result is in degrees.

`ARRAY (<expression1>,<expression2>, & <expressionN>)`
This function creates an array out of the input values.

`ASIN (<expression>)`
This function returns the arc-sine for the input value. Input and result are in radians.

`ATAN (<expression>)`
This function returns the arc-tangent for the input value. Input and result are in radians.

`AVERAGE (<expression>)`
Same as in PC-DMIS expression language.

`COMMANDDATA (<expression1>,<expression2>)`
This expression calculates or displays information from the feature referenced by the command object from the Report Data event. This takes a single required parameter for expression1. This parameter tells PC-DMIS what feature information to display. A second optional parameter, expression2, is provided for future support but currently isn't used.

You can pass string values through expression1:

- "RMS" - This stands for Root Mean Square. It calculates the Root Mean Square value of the feature.
- "T" - This returns the T value of the feature (it is the same as the T axis on a dimension).

You can also pass expressions, like the ISOUTTOL() function, through expression1:

Suppose you have the following expression taken from the TEXTANDCAD_OOT.RTP report template that checks to see if the out-of-tolerance value is greater than zero:

```
IF "COMMANDDATA("=ISOUTTOL()","")>0" Then
Use Template "Reference_ID.lbl"
```

If so, it uses the Reference_ID.lbl label template. For more information on ISOUTTOL, see the ISOUTTOL expression description below.

`CHR (<expression>)`
This function returns the ASCII character value for the corresponding input value, which should be of type integer.

`* COLOR (<expression1>, <expression2>)`
This function makes the text value of expression1 use one of the four colors as currently defined in the color tree. 1 = Marked Color, 2 = Unmarked Color, 3 = Step Mode, and Color 4 = Error Color. See "Changing a String's Text Color".

`* GetTolColor (expression1, expression2, expression3)`
This function takes three expressions: deviation, plus tolerance, and minus

tolerance. It returns the current tolerance color as type COLORREF based on the deviation.

expression1 is the deviation as a double value, expression2 is the plus tolerance as a double value, and expression3 is the minus deviation as a double value.

You can use this returned color with the color properties of objects in reporting, such as **ForeColor** and **BackColor**, to dynamically change the color of the object to reflect the current tolerance value.

These colors are specified in the **Edit Dimension Colors** dialog box. See "Editing Dimension Colors" in the "Editing the CAD Display" chapter.

`CONCAT (<expression1>, <expression2>, & <expressionN>)`
This function concatenates all of the strings specified in expressions 1 through N together into one string.

`COS (<expression>)`
This function returns the cosine for the input value. Input and result are in radians.

`* COUNT (expression1)`
This function returns the number of instances of the data type specified in expression1 for the current command.

`CROSS (<expression1>, <expression2>)`
This function returns the cross product of expession1 and expression2, both of which should be of type vector.

`* DATAFIELD (<expression1>, <expression2>, <expression3>)`
You use this function only within the Custom Report Editor. This function returns information from a specific data field within a feature, dimension, or command. This function takes three parameters: expression1 is a string representing the unique id or id of the command, expression2 is a string representing the data type, and expression3 is the typeindex. Normally, the typeindex is 0, but in cases where a dtype occurs more than once, it will be 1 or greater.

PC-DMIS automatically creates and uses this expression when you drag and drop items from the Edit window into your custom report.

`DATEVALUE ()`
Same as in PC-DMIS expression language.

`DEG2RAD (<expression>)`
This function converts the input from degrees to radians.

`DELTA (<expression1>, <expression2>, <expression3>)`
This function generates a new point moved along the vector specified in expression2 by the distance specified in expression3 from the point specified in expression1.

`DISTANCEFROMEDGE ()`
This function retrieves the distance of the measured centroid of the command's underlying feature from the closest edge on the CAD.

- If the underlying command is a feature, this function uses the measured feature to return the distance.
- If the underlying command is a dimension, this function uses the first feature in the dimension to return the distance.

You can use this function in the Rule Tree Editor to specify different label templates based on the distances returned.

`DOT (<expression1>, <expression2>)`
This function returns the dot product of expression1 and expression2. The input values should be of type point.

`DOUBLE (<expression>)`
This function converts the input value from its current type to a double. In the case of a point, the function returns the distance of the point from the origin.

`* ELAPSEDTIME ()`
This function returns the amount of time taken for execution.

`ELEMENT (<expression1>, <expression2>, <expression3>)`
Same as in PC-DMIS expression language.

`EQUAL (<expression1>, <expression2>)`
This function tests whether two arrays are identical and returns 1 if they are identical; otherwise, it returns 0.

`<expression1> ^ <expression2>`
Same as in PC-DMIS expression language.

`EXPON (<expression>)`
Same as in PC-DMIS expression language.

`* FILENAME ()`
This function returns the full path and file name of the measurement routine.

`FORMAT (<expression1>, <expression2>)`
Same as in PC-DMIS expression language.

`* GETCOUNT (expression1)`
This function returns a long value of the number of instances that exist for the ENUM_FIELD_TYPES data type, specified in expression1. For example, a location dimension that reports in X,Y,Z, and D would return 4 for the Axis data type.

`GETFEATURESSETID (<expression1>)`

During execution, if the current command is a dimension, this function checks to see if the reference feature for this dimension is in a feature set. If it finds the reference feature in a set, it returns that set's ID as a string value with a ".lbl" extension appended to the end it. If it does not find a feature set command with the reference feature, it returns the default string value supplied in <expression1>. This default value should be a label file name that includes the .lbl extension.

For example, suppose you have the following Location dimension that references a circle named CIR1:

```
DIM LOC1= LOCATION OF CIRCLE CIR1 UNITS=IN ,$
GRAPH=OFF TEXT=OFF MULT=10.00 OUTPUT=BOTH
...
END OF DIMENSION LOC1
```

You can use the GetFeatureSetID function inside a rule to automatically determine the label that PC-DMIS displays for this dimension depending on whether or not CIR1 exists inside of a feature set.

For example, this rule automatically uses the LEGACY_DIMENSION.LBL label if it cannot find a FEAT/SET command that contains CIR1:

```
USE TEMPLATE
"=GetFeatureSetID("LEGACY_DIMENSION.LBL")"
```

If there is a FEAT/SET command, you can then change the ID for that command to match the desired label name you want to use (or change a label file name to match the ID). PC-DMIS then uses that label instead.

In this code here, notice that the FEAT/SET references CIR1. It also has its usual label identification changed to "REFERENCE_ID" and the GetFeatureSetID function returns "REFERENCE_ID.LBL":

```
REFERENCE_ID=FEAT/SET,CARTESIAN
   THEO/<0,0,0>,<0,0,1>
   ACTL/<0,0,0>,<0,0,1>
   CONSTR/SET,BASIC,CIR1,,
```

`GETTEMP (<expression1>)`
This function returns the specified temperature or threshold value. One of these string values used for expression1 determines what this function returns:

- "TEMPP" - Returns the part temperature
- "TEMPX" - Returns the X axis temperature
- "TEMPY" - Returns the Y axis temperature
- "TEMPZ" - Returns the Z axis temperature
- "REF_TEMP" - Returns the temperature compensation reference temperature
- "HIGH_THRESHOLD" - Returns the temperature compensation high threshold
- "LOW_THRESHOLD" - Returns the temperature compensation low threshold

`GETPROGRAMINFO (<string>,<optional string>)`
Same as in PC-DMIS expression language.

`IF (<expression1>, <expression2>, <expression3>)`
If expression1 evaluates to a non-zero value, the function returns the value of expression2; otherwise, the function returns the value of expression3.

`INDEX (<expression1>, <expression2>)`
Same as in PC-DMIS expression language.

`INTEGER (<expression>)`
Same as in PC-DMIS expression language.

`ISMARKEDFORARM (<expression>)`
Use this function in Multiple Arm Mode contexts. This function returns 1 if the command is marked for the arm specified in <expression>; otherwise, it returns 0. This function lets you control what is displayed in the report based on the command's executing arm.

```
=IsMarkedForArm(1)
```

would return 1 if the current command is marked for arm 1, otherwise it returns 0.

```
=IsMarkedForArm(2)
```

would return 1 if the current command is marked for arm 2, otherwise it returns 0.

```
=IsMarkedForArm(1) AND IsMarkedForArm(2)
```

would return 1 if the current command is marked for both arms, otherwise it returns 0.

`*ISOUTTOL()`
This function checks if a command is out of tolerance. This is only used with the COMMANDATA expression as discussed above.

`LEFT (<expression1>, <expression2>)`
Same as in PC-DMIS expression language.

`LEN (<expression>)`
For a string, this function returns the number of characters in the string. For an array, it returns the number of elements in the array.

`LN (<expression>)`
Same as in PC-DMIS expression language.

`* LOADSTR (<expression>)`
This function loads the string using the numerical value from the resource files. A negative numerical value causes the string to be loaded from the string's resources. See "Loading Strings from PC-DMIS" for more information.

`LOG (<expression>)`
Same as in PC-DMIS expression language.

`LOWERCASE (<expression>)`
Same as in PC-DMIS expression language.

`<expression1> < <expression2>`
Same as in PC-DMIS expression language.

`MAX (<expression>)`
Same as in PC-DMIS expression language.

`MAXINDEX ( <expression>)`
Same as in PC-DMIS expression language.

`MAXINDICES ( <expression>)`
Same as in PC-DMIS expression language.

`* MEASSCALE ()`
This function returns the re-scale factor used when measuring.

`MIN (<expression>)`
Same as in PC-DMIS expression language.

`MININDEX (<expression>)`
Same as in PC-DMIS expression language.

`MININDICES (<expression>)`
Same as in PC-DMIS expression language.

`MID (<expression1>, <expression2>, <expression3>)`
Same as in PC-DMIS expression language.

`MPOINT (<expression1>, <expression2>, <expression3>)`
Same as in PC-DMIS expression language.

`* NUMMEAS ()`
This function displays a number value that represents the number of dimensions reported.

`* NUMOUTTOL ()`
This function displays the number of reported out-of-tolerance dimensions.

`ORD (<expression>)`
Same as in PC-DMIS expression language.

`PCDMISUSERHIDDENDATAPATH ()`
Same as in PC-DMIS expression language.

`PCDMISUSERVISIBLEDATAPATH ()`
Same as in PC-DMIS expression language.

`PCDMISSYSTEMHIDDENDATAPATH ()`
Same as in PC-DMIS expression language.

`PCDMISSYSTEMVISIBLEDATAPATH ()`
Same as in PC-DMIS expression language.

`PCDMISSYSTEMREPORTINGPATH ()`
Same as in PC-DMIS expression language.

`PCDMISAPPLICATIONPATH ()`
Same as in PC-DMIS expression language.

`* PARTNAME ()`
This function displays the part name (the same as what is shown in the file header).

`* PAGE ()`
This function displays the current page number.

`* PAGES ()`
This function displays the total number of pages.

`PAGEDIMCOUNT ("ObjectID",Range)`
This function takes two parameters. If the first parameter is empty (nothing inside the quote marks), then this function returns the number of dimensions in the current page with a max deviation less than Range*Tol. The second parameter, Range, is a floating number. If the first parameter references a CadReportObject ID or a TextReportObject ID, then this returns the number of in-tolerance dimensions associated with the specified CadReportObject or TextReportObject.

For example, suppose you want to return the number of out of tolerance dimensions with CadReportObject1. You could use the following code to do this:

```
= TotalPageDimCount ("CadReportObject1") -
PageDimCount ("CadReportObject1",1.0)
```

You can also calculate the number of dimensions that contain a specific number of axes. To do this, append ":N" to the ID, where N is a number that represents the number of axes.

You can type `= PageDimCount ("CadReportObject1:4",1.0)` which return the number of dimensions associated with CadReportObject1 that contains at least four axes and that had the fourth axis within the specified tolerance of 1.0. If you do not specify the number of axes, it returns the number of dimensions associated with CadReportObject1 if all axes were within the specified tolerance of 1.0.

For Custom Reports, you can only use this function with a CADReportObject, because a TextReportObject cannot go on a Custom Report.

`RAD2DEG (<expression>)`
Same as in PC-DMIS expression language.

`REAL (<expression>)`
Same as in PC-DMIS expression language.

`* REGSETTING (<expression1>, <expression2>)`
This function displays a value of a specific entry. This function takes two parameters: expression1 determines the section, and expression2 determines the entry.

`REPORTDIMCOUNT (<"expression1">, <expression2>)`
This function acts like the PageDimCount() function except that instead of displaying the number of out of tolerance dimensions for the current page, it displays the total number of out of tolerance dimensions for the entire report. In addition, for <expression1>, you need to use CadReportObject1 or TextReportObject1 as the ID, or leave it empty (with just the quotes).

For Custom Reports, you can only use this function with a CADReportObject, because a TextReportObject cannot go on a Custom Report.

`REPORT_LABEL_AXIS (<expression1>,<expression2>, etc.)`
This function returns a string delimited by \n that describes each axis that the tolerance command outputs.

`* REPORTVALUE (<expression1>)`
This function displays the value of another object's property. It takes one parameter, shown as expression1. This should be a string value of the object's unique ID followed by a period and then the property's name. For example:
`= REPORTVALUE ("text1.text")`

`* REVNUM ()`
This function displays the revision number (the same as what is shown in the file header).

`* RGB (<expression1>, <expression2>, <expression3>, <expression4>)`
This function colors the string specified in expression1 to the color specified via the RGB values of expression2, expression3, and expression4. See "Changing a String's Text Color".

`RIGHT (<expression1>, <expression2>)`
Same as in PC-DMIS expression language.

`ROUND (<expression1>, <expression2>)`
Same as in PC-DMIS expression language.

`* SECTION ()`
This function displays the current section number.

`* SERNUM ()`
This function displays the serial number (the same as what is shown in the file header).

`SIN (<expression>)`
Same as in PC-DMIS expression language.

`SORTUP (<expression>)`
Same as in PC-DMIS expression language.

`SORTDOWN (<expression>)`
Same as in PC-DMIS expression language.

`SQRT (<expression>)`
Same as in PC-DMIS expression language.

`* STATCOUNT ()`
This function returns the statistics count (the same as in the file header).

`STR (<expression>)`
Same as in PC-DMIS expression language.

`STRING (<expression>)`
Same as in PC-DMIS expression language.

`SUM (<expression>`
Same as in PC-DMIS expression language.

`SYSTIME ()`
Same as in PC-DMIS expression language.

`SYSTEMDATE (<expression>)`
Same as in PC-DMIS expression language.

`SYSTEMTIME (<expression>)`
Same as in PC-DMIS expression language.

`* TOGGLESTR (<expression1>, <expression2>)`
This function pulls the toggle string from the resources base on the resource id value in expression1. The function uses a negative number for cases where the string should come from the strings resources. This function uses the value of expression2 to specify the substring to return. The result is the substring.

`* TOGGLESTRING (DATA_TYPE)`
If the specified data type for the given command is of type toggle string, this function returns the full toggle string.

`* TOGGLEVALUE (DATA_TYPE)`
If the specified data type for the given command is of type toggle string, this function returns the index number (or toggle value) of the toggle string.

`* TOL (<expression1>, <expression2>, <expression3>, <expression4>)`
expression1 is the string to color, expression2 is the test value, expression3 is the maximum in-tolerance value, and expression4 is the minimum in-tolerance value. If the test value is in tolerance (between the values for expression3 and expression4), then the software colors the text black. Otherwise, the software colors the text with

the error color (usually red). You can define the error color in the **Color Editor** dialog box (**Edit | Preferences | Edit Window Colors**).

For example, this code displays the text "In Tolerance" in black:

```
=TOL(" In Tolerance ",100,100.5,99.5)
```

This code displays the text "Out of Tolerance" in the error color (by default, this is red):

```
=TOL(" Out of Tolerance ",99,100.5,99.5)
```

* `TOLF (<expression1>, <expression2>, <expression3>, <expression4>, <expression5>)`
This function provides additional formatting capabilities beyond the TOL expression. The first four parameters are the same as listed in the `TOL` function above. The fifth expression determines the font, size, format, and color for an out-of-tolerance condition. It expects this syntax:

"**f:s:w:c**"

f is the name of the font to use.

s is the size of the font in points.

w is the font's formatting, and it can be done using any of these:

B or BOLD for boldface formatting.

I or ITALIC for italic formatting.

R or REGULAR for regular formatting.

BI, BOLDITALIC, IB, or ITALICBOLD for both bold and italic formatting.

c is an RGB(r,g,b) color formatting where r = red, g = green, and b = blue. Each value can be in the range of 0 to 255.

Each of these items is optional and must be separated with a colon (:) character. For example, to set only the font size, you could have ":14", but to set only the color, you would need ":::RGB(0,0,0)".

This example formats any measured dimension that is out of tolerance with an Arial font, size 12, with a boldface formatting and a red color.

```
=TOLF(DIM_MEASURED:N,DIM_OUTTOL:N,0.0,0.0,"Arial:12:B:RGB(255,0,0)"
```

`TOLEXT (<expression1>, <expression2>, <expression3>, <expression4>, <expression5>)`
This function is similar to the TOL function except you can use ExtendedDTypes.

For example:

```
=TOL(NOMINAL:N,DIM_OUTTOL:N,0.0,0.0)

=TOLEXT(REPORT_LABEL_NOMINAL:N,DIM_OUTTOL:N,"SEG=1",0.0,0.0)
```

Where `"SEG=1"` is the extended portion that PC-DMIS adds to the `REPORT_LABEL_NOMINAL` expression.

`TOTALPAGEDIMCOUNT ("ObjectID")`
This function contains one parameter. If the function is empty (nothing between the quotation marks), it returns the total number of dimensions in the current page. If you type the ID of a CadReportObject or TextReportObject in the parameter, PC-DMIS returns the total number of dimensions associated with that object.

You can use this expression `= TotalPageDimCount ("CadReportObject3") to` return the total number of dimensions associated with CadReportObject3.

You can also calculate the number of dimensions that contain a specific number of axes. To do this, append ":N" to the ID, where N is a number that represents the number of axes.

You can type `= TotalPageDimCount ("CadReportObject1:4")` to return the total number of dimensions associated with CadReportObject1 that contained at least four axes.

For Custom Reports, you can only use this function with a CADReportObject, because a TextReportObject cannot go on a Custom Report.

`TOTALREPORTDIMCOUNT ("ObjectID")`
This function is similar to the TotalPageDimCount() function with these important differences: Instead of returning the number of dimensions for the current page, it returns the total number of dimensions for the entire report. This works if there's a CadReportObject or TextReportObject. If you have an object ID as your parameter, it needs to be named CadReportObject1 or TextReportObject1.

For Custom Reports, you can only use this function with a CADReportObject, because a TextReportObject cannot go on a Custom Report.

`TRACEFIELD (<expression>)`
This function displays the given tracefield's name and value in the report. The value of the expression is a number value that represents the order of the listed tracefields

from top to bottom in your measurement routine. So to display the first listed tracefield, the code would read: `= TRACEFIELD (1)`

`UNIT (<expression>)`
Same as in PC-DMIS expression language.

`USERSTRING (<expression1>,<expression2>)`
This function pulls a custom string from a comma separated value (CSV) text file and displays that string in the label. See "Loading Strings from a Text File" for an example of this function.

`* VARIABLE (<expression1>,<expression2>)`
This function displays the defined variable's value. This function takes one or two parameters. expression1 is the string value that represents the variable's ID. expression2 is an optional ID of another command ID or UID. See "Displaying a Variable's Value" for more information.

`VECX (<expression>)`
Same as in PC-DMIS expression language.

`VECY (<expression>)`
Same as in PC-DMIS expression language.

`VECZ (<expression>)`
Same as in PC-DMIS expression language.

Operators for Report Expressions

`<expression1> == <expression2>`
Evaluates to 1 if expression1 equals expression2. Otherwise, evaluates to 0.

`<expression1> >= <expression2>`
Evaluates to 1 if expression1 is greater than or equal to expression 2. Otherwise, evaluates to 0.

`<expression1> <= <expression2>`
Evaluates to 1 if expression1 is less than or equal to expression 2. Otherwise, evaluates to 0.

`<expression1> > <expression2>`
Evaluates to 1 if expression 1 is greater than expression2. Otherwise, evaluates to 0.

`<expression1> < <expression2>`
Evaluates to 1 if expression 1 is less than expression2. Otherwise, evaluates to 0.

`<expression1> - <expression2>`
Subtracts expression2 from expression1.

`<expression1> / <expression2>`
Divides expression1 by expression2.

`<expression1> % <expression2>`
Returns the remainder from expression1 divided by expression2 if any.

`<expression1> * <expression2>`
Multiplies expression1 by expression2.

`<expression1> <> <expression2>`
Compares expression1 to expression 2. If they don't match, it evaluates to 1. If they do match, it evaluates to 0.

`-<expression>`
The unary minus operator negates the value of the operand in expression.

`!<expression>`
Logical NOT operator. This inverts the value of the variable or expression. If the <expression> evaluates to TRUE, the !<expression> evaluates to FALSE. If the <expression> evaluates to FALSE, the !<expression> evaluates to TRUE.

`<expression1>  AND  <expression2>`
Performs a binary AND operation on two numbers. Otherwise, concatenates strings or numbers together in the case of mixed types.

`<expression1>  OR  <expression2>`
Performs a binary OR operation on two numbers.

`<expression1> + <expression2>`
Adds expression1 and expression2 together.

Some Reporting Expressions Examples

The following topics provide you with a few examples of things you can do with the Reporting expression language:

- Displaying a Variable's Value
- Changing a String's Text Color
- Loading Strings from PC-DMIS
- Loading Strings from a Text File

Use the functions and operators in the "Functions and Operators" topic, to create your own Reporting expressions.

Displaying a Variable's Value

You can use the Variable() function in PC-DMIS's reporting language to display a variable's value in your report. This function has the following syntax:

Variable(<varname>, [<optional command id or uid>])

The first parameter, coerced to a string type, represents the name of the variable. You can use the second optional parameter to solve for the variable's value relative to another command.

For example, suppose you have this code in your measurement routine:

```
ASSIGN/V1=2

F1=FEAT/CIRCLE...

ASSIGN/V1=F1. X

COMMENT/OPER,"Some comment text"
```

For the purposes of this example, assume that the comment has a unique ID or "UID" of 245.

Now consider the following examples with the code above:

`=VARIABLE("V1")` - If there are no other statements setting the value of V1 in the report then the value may be either 0, 2, or the same as the measured centroid x value of feature F1. It all depends on which commands have already executed at the point that the report expression is evaluated and which command is currently being processed for the report.

`=VARIABLE("V1","F1")` - If this is the only feature named "F1" in the measurement routine, then the result of evaluating this expression should be 2 since V1 is assigned to 2 directly above feature F1.

`=VARIABLE("V1",245)` - In this case, the UID is used; the value of this expression on the report should be the same as F1.X.

Individual cells of a **GridControlObject** cannot acquire a variable's value from the measurement routine. In most cases, you will need to use **Text** objects.

Variables and Headers

Normally, you *cannot* display a variable's value inside your report's file header because PC-DMIS evaluates the file header before executing statements in the measurement routine. So, when the file header attempts to reference a variable that hasn't yet been

created, PC-DMIS displays a zero value. However, some options do exist to get information from your measurement routine into your header:

Option 1 - Use the tracefields instead of Variables
Use tracefields instead of variables in your measurement routine to capture the information and then use the =TRACEFIELD() function inside the desired cell in the **GridControlObject** of the header's label template to reference it. Tracefields force the template to re-evaluate the report based on the new information causing the tracefield to appear.

Advantages / Disadvantages

- **Advantages** - Relatively easy to set up.
- **Disadvantages** - You aren't actually using variables. Tracefields are used instead and have their own limitations.

Procedure

Step 1: Open the File_Header.lbl file in PC-DMIS and Modify the **GridControlObject** to hold another row of data.

1. In PC-DMIS access the label template editor (**File | Reporting | Edit | Label Template**) and open File_Header.lbl. It will appear in the label template editor. You will see the information is contained inside an object called a GridControlObject.
2. Select the **GridControlObject** and right-click on it to open the **Properties** dialog box containing the properties that make up the GridControlObject. Change **NumRows** to 3 to add another row and then press the Tab key. Notice that the background isn't tall enough to support the added row.
3. From the drop down list on the **Properties** dialog box, select **TheFrame/ TheView** and change the **Height** property to 100 and press the Tab key to accept the change.

Step 2: Add a report expression into the GridControlObject to accept a tracefield's information

1. Select the GridControlObject again.
2. If the handles (green boxes) do not yet include the newly added row, drag the green handles downward until they do.
3. Double-click to make the GridControlObject "active". This shows all the behind-the-scenes expressions.
4. Select the cell that will hold the tracefield's value and type: =TRACEFIELD(1) and then press the Tab key. This expression tells PC-DMIS to put the data for the first tracefield in that cell. For example, if you

wanted the second tracefield's data, you would use =TRACEFIELD(2). (See the attached image file below.)

5. Play around with the cells and format them as desired. The best way to do this is to select one or more cells within the grid and then right-click on it to open the **Grid Properties** dialog box.
6. Click outside the GridControlObject to deactivate it.
7. Save your changes.

Step 3: Test your changes.

1. Execute your measurement routine.
2. Access the Report window (**View | Report Window**).
3. Click the **Redraw the Report** icon on the Reporting window's toolbar.
4. The tracefield's info should now appear in the header.

Option 2 - Embed the Report Template

Embed the report template into your measurement routine sometime after the ASSIGN statements have been defined, and then send the variable's value to the label template as a parameter. Add additional rows or cells as needed and on top of the added cells, add and size a **Text** object for each variable you want to display. Finally, modify the **Text** property through a parameter of the REPORT/TEMPLATE command, like this:

```
ASSIGN/V1="A String Value to Pass"
CS1=REPORT/
TEMPLATE,FILENAME=TEXTONLY.RTP,AUTOPRINT=NO,Section=
-1
   PARAM/TEXT1.TEXT=V1
   PARAM/=
   ENDREPORT/
```

Because the label template is merged into the report template, you can modify certain parameters on the label template by referencing the report template in the above fashion.

Advantages / Disadvantages

- **Advantages** - Variables will now appear in your final report's header.
- **Disadvantages** - Somewhat difficult to set up since you need to add a **Text** object into your label template in addition to the code to embed the report template into the measurement routine. The main disadvantage to this approach, however, is that PC-DMIS generates the report twice, one after the

other (once from PC-DMIS's default functionality and again from the embedded REPORT/TEMPLATE code block).

Option 3 - Put the header information directly into the report
Instead of using a report template that refers to an external label template for the header, such as File_Header.lbl, recreate the **GridControlObject** from the header's label template directly inside your report template. Add additional rows or cells as needed and on top of the added cells, add and size a **Text** object for each variable you want to display. Then for each **Text** object use the =VARIABLE() function to pull in the variable information. For example, =VARIABLE("V1").

Make these remaining modifications inside the report template:

1. In the **TextReportObject** change the rules so that the header label isn't used.
2. Set these properties for the current section, Section1:

   Command Set = All Commands

   Maximum Number of Pages = 1
3. Add a second section, Section2, and give it a **TextReportObject** as well, again changing its rules so that a header label isn't used.
4. Set these properties for Section2:

   Command Set = Continue From Previous Section

   Maximum Number of Pages = 0 (meaning there is no maximum)

Advantages / Disadvantages

- **Advantages** - Variables will now appear in your final report's header.
- **Disadvantages** - Somewhat difficult to set up since you have to recreate the **GridControlObject** inside the report template, add **Text** objects for each variable, and add an additional report section. Additionally, any editing done to the report, such as turning on "Show Feature" or changing Dimensions to "Out of tolerance Only", needs to be done twice, once for page 1 (first section) and once for the other pages (section 2).

Option 4 - Use a generic feature to force report generation
This option uses an empty Generic feature to force the label template to re-evaluate the report and pull needed variables values into your final report.

In your measurement routine, create an empty Generic feature and give it a descriptive name such as:

```
REPORTHEADER=GENERIC/NONE,DEPENDENT,CARTESIAN,OUT,$
```

Next, modify the header label template, adding additional cells as needed to the **GridControlObject**, and then on top of the added cells, add and size a **Text** object for each variable you want to display. Now set the **Text** property for each **Text** object to use the =VARIABLE() function to pull in the variable information. In this option, however, you need to refer to the generic feature by using the extra parameter in the =VARIABLE() function. For example, =VARIABLE("V1","REPORTHEADER")

Advantages / Disadvantages

- **Advantages** - Probably the most versatile approach. Variables will now appear in your final report's header. You will not need to edit the final report twice as in Option 2.
- **Disadvantages** - Somewhat difficult to set up since you will need to include an empty Generic feature in your measurement routine, and add **Text** objects for each variable inside the label template.

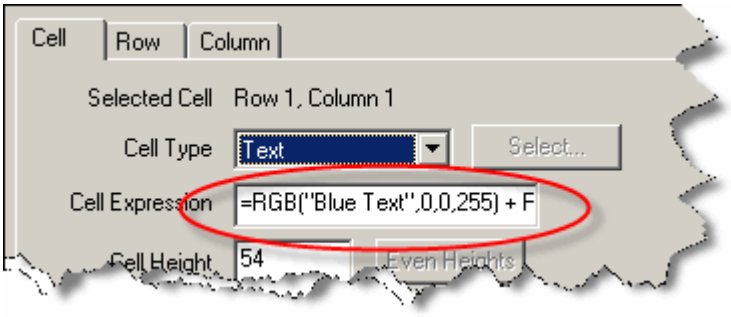
Changing a String's Text Color

Using the RGB Function

The Reporting expression language lets you use a RGB function to define an RGB (Red Green Blue) color value and apply it to a string of characters in expression text. This function takes four parameters, a string parameter, followed by comma separated RGB parameters, like this:

```
=RGB(string,R,G,B)
```

If you type this function in the **Cell Expression** box (or cell) of the GridControlObject, click **OK**, and then click outside of the object, PC-DMIS evaluates the expression and gives the text the specified color value.



Cell Expression box showing the RGB expression

For example, if you type this expression into a cell,

```
=RGB("Blue Text",0,0,255) + RGB(" Black Text",0,0,0) +
RGB(" Yellow Text",255,255,0)
```

then the words would look like this in a template editor:

Displaying a Dimension Color with RGB

For the second parameter in the RGB function, if you use a value that exceeds 255, PC-DMIS interprets that parameter as a COLORREF value instead of a standard 0 - 255 RGB value. In this case, PC-DMIS ignores the second and third parameters (although you do need to include them). You can use this approach to return the color of the dimension's axis if you use the `DIM_RPT_TOLERANCECOLOR1` data type.

For example, suppose you use this expression in a cell of a GridControlObject:

```
=RGB(NOMINAL:N,DIM_RPT_TOLERANCECOLOR1:N,0,0)
```

This grabs the nominal value of the current axis undergoing evaluation and sets the color to that axis's dimension color.

The `:N` behaves as a variable that holds the current dimension axis index so that the expression returns the color value of each axis.

You can also specify a specific axis. For example, suppose you have a dimension with three axes, X, Y, and Z.

- `DIM_RPT_TOLERANCECOLOR1:1` returns the color value for the first axis, X.
- `DIM_RPT_TOLERANCECOLOR1:2` returns the color value for the second axis Y.
- `DIM_RPT_TOLERANCECOLOR1:3` returns the color value for the third axis Z.

If you do not define the optional index (`:N`), it returns the color of the dimension itself.

Using the Color Function

The Color function takes two parameters. The first, a color parameter, is actually a number that represents one of the principle Edit window colors. The second, is the string value to which PC-DMIS applies the color.

```
=COLOR(1, "My Text")
```

The first parameter is a value of 1 to 4 and passes the Edit window color that associated with the following:

1 passes the Marked Color

2 passes the Unmarked Color

3 passes the Step Mode Color

4 passes the Error Color

These colors are defined in the Edit window's **Color Editor** dialog box (**Edit | Preferences | Edit Window Colors**). For information on this dialog box, see "Defining Edit Window Colors" in "Setting Your Preferences" for information.

Loading Strings from PC-DMIS

Similar to how you can change the text color for a cell in the "Changing Expression Text Color" topic, the Reporting expression language lets you pull strings from PC-DMIS's current running language by using this expression:

```
=LOADSTR(<integer expression>)
```

This function takes a single parameter, an integer number that corresponds to the value of a string located in resource.dll or strings.dll.

- A positive number pulls the string from the resource.dll file.
- A negative number pulls the string from the strings.dll file.

If you type this function in the **Cell Expression** box (or cell) of the GridControlObject, click **OK**, and then click outside of the object. PC-DMIS evaluates the expression and returns the string assigned to the specified integer value.

This functionality was added primarily so that the label templates that ship with PC-DMIS use string data from your current language.

Loading Strings from a Text File

With PC-DMIS, you can load custom strings from a comma separated value (CSV) text file and display them inside of a customized label. This is very handy if you need custom strings displayed in a different language. To do this, just localize the csv file and provide it to the operator.

The Syntax

The `USERSTRING` function takes two parameters:

```
=USERSTRING(<expression1>,<expression2>)
```

- The first parameter, Expression 1, is a string pathway that defines the location of the CSV file. This must be inside quotes. If the file cannot be found, due to an incorrect pathway or a lack of read permissions, the function returns, "File cannot be opened".
- The second parameter, Expression 2, is the index number in the CSV file associated with the string of text to display. If the CSV file is accessible, but you specify an index number that doesn't exist in the CSV file, the function returns, "index not found".

The CSV File

The csv file must contain on an individual line: the unique index number, then the comma, and then the string.

To define multi-line strings, use the new line character (\n).

For example, consider this sample CSV file:

```
10,The universe is your playground.
1,Place the part on the table.
3,This is a \nMulti-line string.
...
230,"Bleeding Edge. Cutting Edge."
200,Circle 6001
201,5+5
```

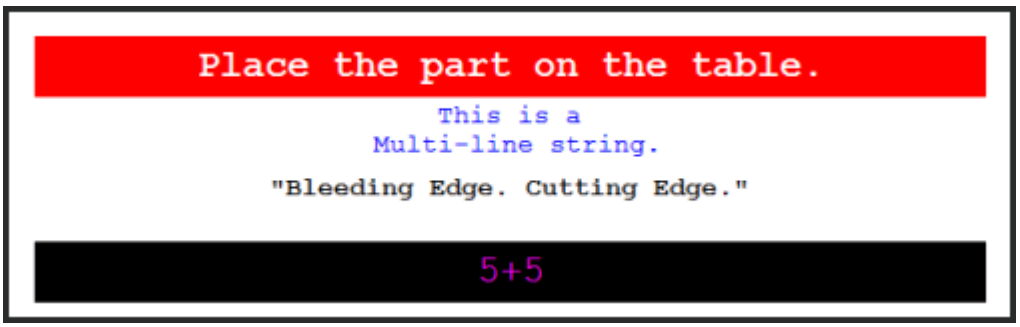
Notice that the numbers preceding each string don't need to be sequential; they only need to be unique. Quotation marks and other characters will come through as typed in the CSV file. Numbers are not mathematically evaluated.

An Example

Suppose the above CSV file was stored in your d:\temp\ directory with a file name of MyStrings.csv. If you wanted to display four different custom strings in a label, your expressions in the cells of the GridControlObject on that label might look like this:

```
=USERSTRING("d:\temp\MyStrings.csv",1)
=USERSTRING("d:\temp\MyStrings.csv",3)
=USERSTRING("d:\temp\MyStrings.csv",230)
=USERSTRING("d:\temp\MyStrings.csv",201)
```

Your label, once evaluated would look like this:



Using Data Types to Find a Report Expression

When using expressions, you usually use expressions that pull data from PC-DMIS. You need to make sure that the command or feature you associate with the label template to do this has the data you are trying to display. Data Types can help you find the correct expression to use.

For example, suppose you create a label template that has a GridControlObject, and in one of the cells you type this expression to display a feature's measured X data:

```
=MEAS_X
```

Now, if you create a report template, add a TextReportObject, and use the **Rule Tree Editor** to associate your label template with PREHIT commands, in the Report window the cell will not display anything. Why? Because the PREHIT command does not have a measured X field. To use the correct expression, you can view data types for different fields in the Edit window's Command Mode.

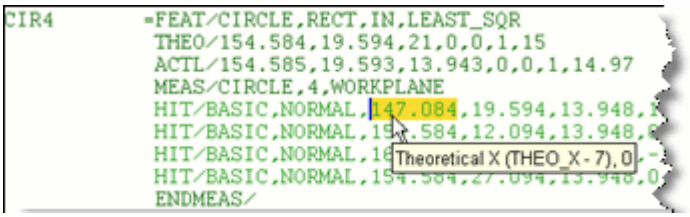
This procedure explains how to turn on and view data types:

1. Access the Edit window.
2. Place the Edit window into Command Mode.
3. Right-click on the Edit window. A shortcut menu appears.

Data Type Information Menu Option

4. Select **Change Popup Display | Data Type Information**.
5. Hover your mouse pointer over a field in a command, and PC-DMIS will display a small, yellow popup that shows that field's data type. The first part of the value in parentheses is the data type and an equivalent expression exists in the list of expressions.

Consider this example:



This data type popup shows that THEO_X is a valid expression for this command. If you typed "=THEO_X" into an appropriate location, PC-DMIS would display this feature's theoretical X value.

Using data types you can ensure that your report and label templates are using expressions supported by that command.

A List of Available Data Types

This list of data types displays the data type names in alphabetical order, their associated type numbers, descriptions, index values, and value strings as appropriate. When you use data types inside VB scripts in template events and rules, you may need to use the data type number as not all script locations accept the enumerated data type value.

Data type names followed by (Deprecated) are no longer used. They remain available so that existing measurement routines and report templates continue to open, but PC-DMIS will remove them in a future release. Do not use them in new report templates, rules, or scripts.

-A-

Number	Data Type Name	Data Type Description	Index	Value String
481	ABOVEBELOW_CONFIG			
409	ADDITIONAL_CHART	For SPC		
153	ALIGN_LIST	Show alignment in alignment list (Yes / No)		
300	ANGLE_COMP_TOGGLE	Toggle field for complement state of angle dimensions		
149	ANGLE_OFFSET	Angle Offset for loops and rotary tables		
373	ANGULARITY_NOM_ANGLE	The reference angle used, not the dimension's nominal		
103	ANGVEC_I	Angle vector i	0	Numeric value
104	ANGVEC_J	Angle vector j	0	Numeric value
105	ANGVEC_K	Angle vector k	0	Numeric value
164	ARROW_MULTIPLIER	Dimension arrow multiplier value		
479	ARTICULATEDARM_TYPE			
234	AUTO_CLEAR_PLANE	Auto clear plane flag		
461	AUTO_ONERROR_TYPE			
533	AUTO_PH9			
219	AUTO_PRINT	Auto print flag for hyper report object		
295	AUTOBEEPING	Turn auto trigger beeping on/off		

52	AUTOFIT_CONSTRAINT	Toggle for type of constraint on best fit alignment		
298	AUTOTOLZONE	Tolerance zone for AUTOTRIGGER		
294	AUTOTRIGGERONOFF	Turn the auto trigger on or off.		
140	AVERAGE_ERROR	Flag for iterative alignments		
749	AXIS_DESCRIPTION			
747	AXIS_MINUS_TOL			
132	AXIS_NOMINAL	Axis		
748	AXIS_NOMINAL (Deprecated)			
746	AXIS_PLUS_TOL			

-B-

Number	Data Type Name	Data Type Description	Index	Value String
51	BF_MATH_TYPE	Math type used to calculate best fit		
50	BOUND_TYPE	Bound / Unbound		
967	BOUDARY_OFFSET	Gets and sets the boundary offset distance during a Void Detection.		
360	BOUNDARY_POINT_X	Boundary Point x		
361	BOUNDARY_POINT_Y	Boundary point y		
362	BOUNDARY_POINT_Z	Boundary point z		
476	BSMETHOD_TYPE			
207	BUFFER_SIZE_TYPE	File i/o buffer size		

-C-

Number	Data Type Name	Data Type Description	Index	Value String
492	CAD_COMP			
237	CAD_TOLERANCE	CAD tolerance for perimeter scans		
471	CALC_STYLE_FILE			
413	CENTER_POINT	For SPC		

478	CENTER_ROTATION_MEAS	Best fit alignments		
477	CENTER_ROTATION_THEO	Best fit alignments		
445	CHART_SUB_TYPE	For SPC		
388	CHART_TYPE	For SPC: charts		
42	CIRC_TYPE	Circular or straight probe motion (circles and cylinders)		
614	CLIP_LEFT_DIST			
604	CLIP_LOW_DIST			
615	CLIP_RIGHT_DIST			
603	CLIP_UP_DIST			
244	COL132_TYPE	On / Off setting for column 132 object		
701	COLUMN_HDR			
296	COLUMN_ID	Sets the column id for a load or unload column command		
245	COMMAND_STRING	For external command object		
189	COMMENT	Comment text		
709	COMMENT_INPUT	Add new type for comment input value		
190	COMMENT_TYPE	Comment type		
724	COMPOSITE			
468	CONE_CONVEX_TYPE			

60	CONE_LENGTH_ANGLE_TYPE	For cones: show length or angle		
39	COORD_TYPE	Coordinate System		
621	COP_BOOLEANTYPE			
618	COP_COLORMAP			
619	COP_COPLEMENT			
616	COP_EXPORTFILETYPE			
543	COP_FILTER			
622	COP_IMPORTFILETYPE			
617	COP_SELECTIONTYPE			
544	COP_SIZE			
545	COP_TYPE			
425	CPOINT_DIAM			
428	CPOINT_F_SCANSPEED			
422	CPOINT_I			
423	CPOINT_J			
424	CPOINT_K			
426	CPOINT_SCAN_CROSS_TOTAL			
427	CPOINT_SCAN_DENSITY			
430	CPOINT_TYPE			
419	CPOINT_X			

420	CPOINT_Y			
421	CPOINT_Z			
433	CREATE_WEIGHTS	For best fit 2D/3D alignments		
65	CURVE_TYPE	For curves - type of curve		

-D-

Number	Data Type Name	Data Type Description	Index	Value String
252	DATA_MEM_PAGES	Database memory pages for statistics object		
250	DATA_READ_LOCK	Database read lock for statistics object		
251	DATA_WRITE_LOCK	Database write lock for statistics object		
895	DATUM1_MMB_SIZE2 (Deprecated)			
731	DATUM1_MODIFIER			
734	DATUM1_MODIFIER2 (Deprecated)			
725	DATUM2 (Deprecated)	This is because DATUM and DATUM2 are used in two different groups in Summary Mode.		
896	DATUM2_MMB_SIZE2 (Deprecated)			
732	DATUM2_MODIFIER			
735	DATUM2_MODIFIER2 (Deprecated)			
897	DATUM3_MMB_SIZE2 (Deprecated)			
733	DATUM3_MODIFIER			

736	DATUM3_MODIFIER2 (Deprecated)			
389	DB_CHART_NAME	For SPC: chart name		
386	DB_QUERY_OP	For SPC		
387	DB_SOURCE_NAME	For SPC		
459	DB_SOURCE_TYPE	For SPC		
539	DELETE_TYPE	FILE/CLOSE, fptr, KEEP | DELETE		
203	DESCRIPTION	Subroutine parameter description		
727	DESCRIPTION2	This is because DESCRIPTION and DESCRIPTION2 are used in two different groups in Summary Mode.		
133	DEST_EXPR	Destination expressions (assign, subroutine, hyper report)		
353	DEV_DIAM	Dev diam value		
280	DEV_PERPEN_CENTERLINE			
350	DEV_X	Dev x value		
351	DEV_Y	Dev y value		
352	DEV_Z	Dev z value		
390	DEVIATION_ANGLE	Deviation angle		
180	DEVIATION_SYMBOLS	Dimension format deviation symbols toggle		
737	DEVPERCENT_NOM			
739	DEVPERCENT2			

199	DIGIT_COUNT	Digit count for dmis file read/ write command		
324	DIM_BONUS	Dimension bonus		
340	DIM_DEVIATION	Dimension deviation		
182	DIM_HEADING	Dimension format heading item		
304	DIM_ID	For Best Fit 2d Alignments		
160	DIM_INFO_LOC	Dimension info object location axis toggle		
159	DIM_INFO_ORDER	Dimension info order toggle		
161	DIM_INFO_TP_LOC	Dimension info object true position location axis toggle		
173	DIM_LENGTH	Dimension length		
754	DIM_LENGTH2 (Deprecated)			
332	DIM_MAX	Dimension max value		
328	DIM_MEASURED	Measured dimension value		
336	DIM_MIN	Dimension min value		
344	DIM_OUTTOL	Dimension out of tolerance		
860	DIM_PLANE_PROJECTION_LENGTH (Deprecated)			
861	DIM_PLANE_PROJECTION_LENGTH2 (Deprecated)			
858	DIM_PLANE_PROJECTION_TYPE (Deprecated)			
859	DIM_PLANE_PROJECTION_TYPE2 (Deprecated)			

862	DIM_PLANE_PROJECTION_WIDTH (Deprecated)			
863	DIM_PLANE_PROJECTION_WIDTH2 (Deprecated)			
703	DIM_RPT_DATUM			
705	DIM_RPT_DEVPERCENT			
704	DIM_RPT_GRAPHIC			
706	DIM_RPT_ISBILATERAL			
702	DIM_RPT_ISDATUM			
707	DIM_RPT_NUMZONES			
917	DIM_RPT_TOLERANCECOLOR1	Gets the color of the dimension as determined by PC-DMIS. This is the primary color.		
918	DIM_RPT_TOLERANCECOLOR2	Gets the secondary color for bilateral dimensions that have a secondary color. If the color is not defined for this dimension, it returns -1 as the value.		
177	DIM_TEXT	Dimension format dim text toggle		
178	DIM_TEXT_OPTIONS	Dimension format dim text options toggle		
510	DISPLAY_ADVANCED_PARAMETERS			
236	DISPLAY_HITS	Scan flag (yes / no) for display of scan hits		
184	DISPLAY_ID	Point info show id toggle		
607	DISPLAY_PROBE_PARAMETERS			

256	DISPLAY_TRACE	Tracefield object yes/no toggle for dialog display		
185	DISPLAY_TYPE	Point info show type toggle		
155	DISTANCE	Distance		
676	DRF_COLUMN_HDR			
1337	DRF_DATUMFIT	Optimized Fit column heading; appears only when a datum feature violates its material boundary		
1338	DRF_DATUMFITVALUE	Optimized Fit value (Interference); appears only when a datum feature violates its material boundary		
681	DRF_ROTATIONX			
682	DRF_ROTATIONY			
683	DRF_ROTATIONZ			
677	DRF_SEGNAME			
678	DRF_SHIFTX			
679	DRF_SHIFTY			
680	DRF_SHIFTZ			
641	DRF_TBLHDR			
291	DTYPE_LEAPFROGFULLPARTIAL	Leapfrog type: Full leapfrog or partial leapfrog		
290	DTYPE_LEAPFROGNUMHITS	Number of hits to be taken during leapfrog procedure		
289	DTYPE_LEAPFROGTYPE	Type of leapfrog being performed		

-E-

Number	Data Type Name	Data Type Description	Index	Value String
99	END_ANG	End angle	0	Radian value
144	END_NUM	End number of loop		
787	END_OFFSET	Ending depth	0	Ending depth value
467	ERROR_LABEL			
202	ERROR_MODE	Error mode for on error flow control command		
201	ERROR_TYPE	Error type for on error flow control command		
292	EXCLUSION_ZONE	Exclusion zone command is on or off		
293	EXECUTE	Is the attached measurement routine executed (Yes / No)		
595	EXPOSURE			

-F-

Number	Data Type Name	Data Type Description	Index	Value String
79	F_AUTOMOVE	Auto move distance		
85	F_BOXLENGTH			

84	F_BOXWIDTH			
88	F_CHECK	The only numeric associated with the CPCDcheck feature		
87	F_CIRCRADIN			
86	F_CIRCRADOUT			
81	F_CORNER_RADIUS			
78	F_DEPTH	Depth of measurements	0	Depth value
82	F_INCREMENT			
80	F_INDENT	Measurement offsets for various features	1	Offset value
243	F_LOCATION	Location value for section scans		
89	F_MAXACCELX	Maximum X acceleration		
90	F_MAXACCELY	Maximum Y acceleration		
91	F_MAXACCELZ	Maximum Z acceleration		
168	F_MINUS_TOL	Dimension minus tol value		
95	F_MOVESPEED	Move speed		
74	F_OFFSET	Offset value		
76	F_PITCH	Auto feature pitch		
167	F_PLUS_TOL	Dimension plus tol value		
97	F_SCANSPEED	Scan speed		
434	F_SIZE	For SPC		

75	F_SPACER	Auto feature spacer	0	Max distance value
77	F_THICKNESS	Sheet metal thickness	0	Thickness value
593	F_THICKNESS_EDGE			
83	F_TOLERANCE			
96	F_TOUCHSPEED	Touch speed		
208	FAIL_ON_EXIST	File i/o fail on exist mode		
867	FCF_RUNOUT_TYPE (Deprecated)			
945	FCF_STANDARD_TYPE (Deprecated)			
303	FEAT_TYPE	Feature Type		
198	FIELD_WIDTH	Field width for dmis file read/write command		
206	FILE_COMMAND_TYPE	File i/o command type		
152	FILE_NAME	File name		
197	FILE_POINTER	File pointer name for file i/o commands		
598	FILTER_NEIGHBOR_NUM			
606	FILTER_TOGGLE			
600	FILTER_TOL_ABOVE			
601	FILTER_TOL_BELOW			
602	FILTER_TOL_RIGHT			

472	FILTER_TYPE	Basic scan filter type		
460	FIND_HOLE_PERCENT	For the check command and find hole check distance		
54	FIND_NOM_AXIS_TYPE	Selects between X, Y, Z, or user defined nominal axis		
233	FIND_NOMS_TYPE	Type for find noms mode		
47	FINDHOLE_TYPE	Use find hole algorithm (Yes / No)		
527	FINDNOMS_BESTFIT			
528	FINDNOMS_ONLYSELECTED			
452	FIT			
465	FIXTURE_TOL			
226	FIXTURE_TYPE	Type for load fixture object		
246	FLY_MODE_TYPE	Fly Mode Off/On toggle		
560	FREQUENCY			

-G-

Number	Data Type Name	Data Type Description	Index	Value String
63	GEN_FEAT_TYPE	Generic feature type		
64	GEN_ALIGN_TYPE	Generic feature alignment type (independent / dependent)		
162	GRAPH_ANALYSIS	Use graphical analysis on dimension (yes / no)		
183	GAP_ONLY_TYPE	ON / OFF toggle		
408	GRID	For SPC		
458	GRAPH_OPTION	Dimension info and point info commands		
708	GDT_SYMBOL			
730	GDT_SYMBOL2			
783	GRAPH_ANALYSIS_POINT_SIZE	Auto Feature cad point size		
784	GRAPH_ANALYSIS_PLUS_TOL	Auto Feature cad point plus tolerance		
785	GRAPH_ANALYSIS_MINUS_TOL	Auto Feature cad point minus tolerance		

-H-

Number	Data Type Name	Data Type Description	Index	Value String
68	HITINT_TYPE	Interior/Exterior measurement flag		
223	HIGH_THRESHOLD	High threshold for temp comp object		
359	HIT_TYPE	Hit type		
407	HISTOGRAM	For SPC		
483	HIGH_ACCURACY			

-I-

Number	Data Type Name	Data Type Description	Index	Value String
2	ID	Object id		
40	INOUT_TYPE	Inside / Outside		
72	INIT_HITS	Initial hits		
138	ITEM_USED	Best fit item used in calculation of alignment		
150	INTERNAL_EXTERNAL	Recall Alignment		
204	INDEX_START	Array index start number		
205	INDEX_END	Array index end number		
354	ITERATE_COLUMNS	Iterate columns setting		
392	IGNOREMOTIONERRORS_TYPE	For ignore motion errors command		
454	IOCHANNEL_NUMBER	For IO channel management commands		
455	IOCHANNEL_PULSE_WIDTH (Deprecated)			
456	IOCHANNEL_PULSE_INTERVAL (Deprecated)			
457	IOCHANNEL_PULSE_DURATION (Deprecated)			
596	INTENSITY			

-J-

No Entries Available

-K-

No Entries Available

-L-

Number	Data Type Name	Data Type Description	Index	Value String
200	LABEL_ID	Label field for flow control objects that reference labels		
729	LEADER_LINE_ID (Deprecated)			
480	LEFTYRIGHTY_CONFIG			
4	LEVEL_REF_ID	Ref id for a level feature of an iterative alignment		
62	LIN_POL_FILT_TYPE	Filter object - linear or polar filter		
782	LINE1_BONUS			
643	LINE1_CALLOUT			
644	LINE1_COLUMN_HDR			
650	LINE1_DEV			
651	LINE1_DEVPERCENT			
750	LINE1_DEVPERCENT_NOM			
752	LINE1_DEVPERCENT2			

645	LINE1_FEATNAME			
652	LINE1_ISBILATERAL			
768	LINE1_MAX			
647	LINE1_MEAS			
769	LINE1_MIN			
649	LINE1_MINUSTOL			
646	LINE1_NOMINAL			
653	LINE1_NUMZONES			
765	LINE1_OUTTOL (Deprecated)			
648	LINE1_PLUSTOL			
642	LINE1_TBLHDR			
751	LINE1_USE2DEVIATIONS			
686	LINE2_AXIS			
658	LINE2_BONUS			
655	LINE2_CALLOUT			
656	LINE2_COLUMN_HDR			

901	LINE2_DATUMA_DOF (Deprecated)			
902	LINE2_DATUMB_DOF (Deprecated)			
903	LINE2_DATUMC_DOF (Deprecated)			
660	LINE2_DATUMSHFT			
662	LINE2_DEV			
663	LINE2_DEVANG			
664	LINE2_DEVPERCENT			
740	LINE2_DEVPERCENT_NOM			
742	LINE2_DEVPERCENT2			
657	LINE2_FEATNAME			
697	LINE2_ISBILATERAL			
695	LINE2_MAX			
688	LINE2_MEAS			
696	LINE2_MIN			
694	LINE2_MINUSTOL			

687	LINE2_NOMINAL			
698	LINE2_NUMZONES			
766	LINE2_OUTTOL			
693	LINE2_PLUSTOL			
654	LINE2_TBLHDR			
659	LINE2_TOL			
661	LINE2_UNUSEDZONE			
741	LINE2_USE2DEVIATIONS			
669	LINE3_BONUS			
666	LINE3_CALLOUT			
667	LINE3_COLUMN_HDR			
904	LINE3_DATUMA_DOF (Deprecated)			
905	LINE3_DATUMB_DOF (Deprecated)			
906	LINE3_DATUMC_DOF (Deprecated)			
671	LINE3_DATUMSHFT			

673	LINE3_DEV			
674	LINE3_DEVANG			
675	LINE3_DEVPERCENT			
743	LINE3_DEVPERCENT_NOM			
745	LINE3_DEVPERCENT2			
668	LINE3_FEATNAME			
699	LINE3_ISBILATERAL			
774	LINE3_MAX			
771	LINE3_MEAS (Deprecated)			
775	LINE3_MIN			
773	LINE3_MINUSTOL (Deprecated)			
770	LINE3_NOMINAL (Deprecated)			
700	LINE3_NUMZONES			
767	LINE3_OUTTOL (Deprecated)			
772	LINE3_PLUSTOL (Deprecated)			

665	LINE3_TBLHDR (Deprecated)			
670	LINE3_TOL			
672	LINE3_UNUSEDZONE			
744	LINE3_USE2DEVIATIONS			
355	LOAD_TYPE	Load setting		
287	LOCATOR_BMP	Feature Locator BMP On/Off		
288	LOCATOR_WAV	Feature Locator WAV On/Off		
210	LOW_FORCE	Low force for option probe object		
224	LOW_THRESHOLD	Low threshold for temp comp object		

-M-

Number	Data Type Name	Data Type Description	Index	Value String
227	MACHINE_TYPE	Machine type for load machine object		
485	MAGNIFICATION			
176	MAN_RETRACT			
94	MANUAL_FINE_PROBING	Manual fine probing		
534	MANUAL_PREPOSITION	Whether manual pre-position needed on vision target		

221	MATERIAL_COEFFICIENT	Material coefficient for temp comp object		
242	MAX_ANGLE	Setting for variable technique on scans		
209	MAX_FORCE	Max force for option probe object		
240	MAX_INCREMENT	Setting for variable technique on scans		
491	MEAN			
569	MEAS_A			
612	MEAS_A2	For CMT characteristic point 1		
30	MEAS_ANGLE	Measured angle		
721	MEAS_AREA			
556	MEAS_DEPTH			
29	MEAS_DIAM	Measured diameter		
584	MEAS_EA			
585	MEAS_EH			
624	MEAS_END_ANG			
583	MEAS_ER			
313	MEAS_EX	Measured endpoint x component		
314	MEAS_EY	Measured endpoint y component		
315	MEAS_EZ	Measured endpoint z component		
552	MEAS_FLUSH			

554	MEAS_GAP			
570	MEAS_H			
613	MEAS_H2	For CMT characteristic point 1		
306	MEAS_HEIGHT	Measured height		
25	MEAS_I	Measured i		
26	MEAS_J	Measured j		
27	MEAS_K	Measured k		
28	MEAS_LENGTH	Measured length		
305	MEAS_MINOR_AXIS	Measured minor axis (ellipse)		
719	MEAS_PERIMETER			
568	MEAS_R			
611	MEAS_R2	For CMT characteristic point 1		
581	MEAS_SA			
582	MEAS_SH			
307	MEAS_SLOTVEC_I	Measured slot vector i - See the "Slot Vector Data Type" note below.		
308	MEAS_SLOTVEC_J	Measured slot vector j - See the "Slot Vector Data Type" note below.		
309	MEAS_SLOTVEC_K	Measured slot vector k - See the "Slot Vector Data Type" note below.		
580	MEAS_SR			
623	MEAS_START_ANG			

310	MEAS_SX	Measured start point x component		
311	MEAS_SY	Measured start point y component		
312	MEAS_SZ	Measured start point z component		
316	MEAS_WIDTH	Measured width		
22	MEAS_X	Hit point measured x	point number	X-Coordinate
396	MEAS_X2	For CMT characteristic point 1		
23	MEAS_Y	Hit point measured y	point number	Y-Coordinate
397	MEAS_Y2	For CMT characteristic point 1		
24	MEAS_Z	Hit point measured z	point number	Z-Coordinate
398	MEAS_Z2	For CMT characteristic point 1		
141	MEASURE_ALL_FEATURES	Flag for iterative alignments		
59	MEASURE_ORDER_TYPE	Measure order for edge feature SURFACE/ EDGE/ BOTH		
66	MEASURED_2D3D_TYPE	Determines plane in which solution will lie or 3D solution		
106	MEASVEC_I	Measure vector i		
107	MEASVEC_J	Measure vector j		
108	MEASVEC_K	Measure vector k		
357	METHOD_TYPE	Scan method type		
100	MIDPOINT_X	Midpoint x		

101	MIDPOINT_Y	Midpoint y		
102	MIDPOINT_Z	Midpoint z		
241	MIN_ANGLE	Setting for variable technique on scans		
239	MIN_INCREMENT	Setting for variable technique on scans		
486	MINOR_WORD_TOGGLE			
58	MODE_TYPE	MANUAL or DCC		
45	MOVE_TYPE	Use auto move (yes / no)		

-N-

Number	Data Type Name	Data Type Description	Index	Value String
429	N_CONTROLPOINTS	Number of control points in a scan		
70	N_HITS	Number of hits per row	0	1 or more
55	N_INIT_HITS_TYPE	Toggle field for fixed number of initial hits		
56	N_PERM_HITS_TYPE	Toggle field for fixed number of perm hits		
71	N_ROWS	Number of rows	0	1 or more
489	N_SIDES	Number of polygon sides		
249	NEW_STATS_DIR	Empty stats directory field in editor for statistics object		
157	NEW_TIP	New tip in move ph9 offset		
0	NINGUNO_TIPO	No type, default value, used with constants		
826	NO_APPROACH_VECTOR_FLIP	Allow the feature vector to be flipped during optimization	0	"YES" or "NO"
482	NOFLIPFLIP_CONFIG			
166	NOMINAL	Dimension nominal value		
321	NOMINAL_COLOR	Vision Nominal Color		
232	NORM_RELEARN	Normal scan mode or re-learn nominals mode		
317	NUM_CONTROL_POINTS	Number of control points		
320	NUM_FIT_POINTS	Number of points used for fitting surface		

356	NUM_ITERATIONS	Number of iterations		
215	NUM_RETURN_DATA	Return data for option probe object		

-O-

Number	Data Type Name	Data Type Description	Index	Value String
61	OFFSET_LINE_METHOD	method for offset lines		
238	OFFSET_TOLERANCE	offset tolerance for perimeter scans		
156	OLD_TIP	old tip in move ph9 offset		
285	ONOFF_TYPE	Initially implemented for use with IGNOREROTAB but can be used with any ON OFF toggle field		
620	OPERTYPE			
1342	OPT_ROTATIONX	Optimized rotation vector, x component, in degrees		
1343	OPT_ROTATIONY	Optimized rotation vector, y component, in degrees		
1344	OPT_ROTATIONZ	Optimized rotation vector, z component, in degrees		
1339	OPT_SHIFTX	Optimized shift in the x direction		
1340	OPT_SHIFTY	Optimized shift in the y direction		

1341	OPT_SHIFTZ	Optimized shift in the z direction		
220	ORIGIN	Origin data point for temp comp object		
6	ORIGIN_REF_ID	Ref id for an origin feature of an iterative alignment		
449	OUTPUT_DMIS_REPORT	DMO reporting		
448	OUTPUT_FEAT_W_DIMENS	DMO reporting		
447	OUTPUT_FEATURE_NOMS	DMO reporting		
165	OUTPUT_TYPE	Mode of output for dimensions		
558	OVERLAP			
559	OVERSCAN			
446	OVERWRITE	DMO reporting		

-P-

Number	Data Type Name	Data Type Description	Index	Value String
374	_PRINT_TO_FILE	Prepended with _ to avoid confusion with Edit window constants		
375	_PRINT_TO_PRINTER	Prepended with _ to avoid confusion with Edit window constants		

191	PART_NAME			
519	PATTERN_TYPE	Auto plane pattern	0	"SQUARE" or "RADIAL"
487	PERCENTAGE			
73	PERM_HITS	Number of Perm (sample) hits	0	0 or more
170	PERP_PARALLEL_TYPE	Dimension perpendicular parallel type		
115	PINVEC_I	Pin vector i		
116	PINVEC_J	Pin vector j		
117	PINVEC_K	Pin vector k		
605	PIXEL_TOGGLE			
186	POINT_INFO_HEADING	Point info heading type		
380	POINTINFO_FILTER_DEVIATION	Toggle field for deviation filter state of point info commands		
381	POINTINFO_FILTER_ DEVIATION_NUMBER	Number field associated with deviation filter state of point info commands		
301	POINTINFO_FILTER_INTERVAL	Toggle field for interval filter state of point info commands		
302	POINTINFO_FILTER_ INTERVAL_NUMBER	Number field associated with interval filter state of point info commands		
382	POINTINFO_FILTER_OUTTOL	Toggle field for out of tolerance filter state of point info commands		
378	POINTINFO_FILTER_WORST	Toggle field for worst filter state of point info commands		

379	POINTINFO_FILTER_ WORST_NUMBER	Number field associated with worst filter state of point info commands		
218	POLAR_VECTOR_COMPENSATION	Polar vector compensation object		
277	POS_REPORT_AXIS_X	Positive reporting axis		
278	POS_REPORT_AXIS_Y	Positive reporting axis		
279	POS_REPORT_AXIS_Z	Positive reporting axis		
462	POS_REPT_DISPLAY_OPTION			
214	POSITIONAL_ACCURACY	Positional accuracy for option probe object		
786	PPAP_INDEX			
399	PPROG	Measurement routine name (used in a database query)		
175	PRECISION	Dimension display precision		
377	PRINT_DELETE_RUNS			
376	PRINT_DRAFTMODE			
213	PROBE_ACCURACY	Probe accuracy for option probe object		
228	PROBE_COMP	Probe comp (on/off) toggle		
299	PROBING_MODE	Probing mode for option probe		
174	PROFILE_FORM_TYPE	Dimension profile form type toggle		
550	PROFILE_TYPE			

521	PROGRAM_GAGE_FEAT_TYPE			
522	PROGRAM_GAGE_TYPE			
118	PUNCHVEC_I	Punch vector i		
119	PUNCHVEC_J	Punch vector j		
120	PUNCHVEC_K	Punch vector k		

-Q-

Number	Data Type Name	Data Type Description	Index	Value String
470	QUERY_SHOW_GRAPHIC_SETTINGS" ? >QUERY_SHOW_GRAPHIC_SETTINGS			

-R-

Number	Data Type Name	Data Type Description	Index	Value String
171	RADIUS_TYPE	Dimension 2D Distance radius type		
196	READ_WRITE	Read / Write toggle		
46	READPOS_TYPE	Read Position (yes / no)		
3	REF_ID	ID of object being referred to		
222	REF_TEMP	Reference temperature for temp comp object		
412	REGR	For SPC		
142	REPIERCE_CAD	Flag for iterative alignments		
383	REPORT_SURFVEC_I	Extended sheet metal vector for DCC Edge		
384	REPORT_SURFVEC_J	Extended sheet metal vector for DCC Edge		
385	REPORT_SURFVEC_K	Extended sheet metal vector for DCC Edge		
121	REPORTVEC_I	Report vector i		
122	REPORTVEC_J	Report vector j		
123	REPORTVEC_K	Report vector k		
188	RET_ONLY_TYPE	Retrolinear only on / off toggle		
216	RETURN_SPEED	Return speed for option probe object		
192	REVISION_NUMBER	File header revision number		
48	RMEAS_TYPE	Use relative measures (yes / no)		

69	RMEASFEATID	Name of relative measure feature		
524	RMEASFEATIDX	Referenced feature (X axis)	0	Feature label
525	RMEASFEATIDY	Referenced feature (Y axis)	0	Feature label
526	RMEASFEATIDZ	Referenced feature (Z axis)	0	Feature label
5	ROTATE_REF_ID	Ref id for a rotate feature of an iterative alignment		
158	ROTATION_TYPE	Move rotary table type		
286	ROW_ID			
639	RPT_DIMENSION_TABLES			

-S-

Number	Data Type Name	Data Type Description	Index	Value String
151	SAVE_ALIGN_CAD_TO_PARTS	Save Align Save Type		
92	SCAN_ACCELERATION	Scan acceleration		
265	SCAN_AXISVEC_I	Axis vector i - for boundary conditions...		
266	SCAN_AXISVEC_J	Axis vector j - for boundary conditions...		
267	SCAN_AXISVEC_K	Axis vector k - for boundary conditions...		
432	SCAN_BNDRY_TYPE			
274	SCAN_CROSS_TOTAL	Number of crossings allowed in boundary condition		

259	SCAN_CUTPLANEVEC_I	Cut plane vector i		
260	SCAN_CUTPLANEVEC_J	Cut plane vector j		
261	SCAN_CUTPLANEVEC_K	Cut plane vector k		
217	SCAN_DENSITY	Scan density for option probe object		
276	SCAN_EDGE_THICK	Edge thickness for edge scans		
268	SCAN_ENDVEC_I	End touch vector i		
269	SCAN_ENDVEC_J	End touch vector j		
270	SCAN_ENDVEC_K	End touch vector k		
271	SCAN_INITDIR_I	Unit direction vector i		
272	SCAN_INITDIR_J	Unit direction vector j		
273	SCAN_INITDIR_K	Unit direction vector k		
262	SCAN_INITVEC_I	Initial touch vector i		
263	SCAN_INITVEC_J	Initial touch vector j		
264	SCAN_INITVEC_K	Initial touch vector k		
93	SCAN_OFFSET_FORCE	Scan offset force		
358	SCAN_TECHNIQUE	Scan technique		
275	SCAN_TIME_INCR	Time increment for manual scans		
540	SCREEN_CAPTURE_AUTO_TIME	Screen capture time (always or on error)		
536	SCREEN_CAPTURE_AUTO_TYPE	Screen capture auto type (which gives more options that basic screen capture)		

503	SCREEN_CAPTURE_QUALITY	Screen capture image color depth		
502	SCREEN_CAPTURE_SCALE	Screen capture image size		
535	SCREEN_CAPTURE_TYPE	Screen capture type (can now do Live Image View as well as CAD)		
57	SEARCHMODE_TYPE	Box or circular search mode for high point feature		
764	SECTION_INDEX			
225	SENSOR_LIST	Sensor list for temp comp object		
193	SERIAL_NUMBER	File header serial number		
494	SHOW_COLUMN			
136	SHOW_DETAILS	Display details for alignments and scans		
179	SHOW_HEADINGS	Dimension format headings toggle		
135	SHOW_IDS	Display id for alignments and loops		
414	SHOW_MORE_SPC_CALCS	For SPC		
723	SHOW_NOMS			
728	SHOW_OPTIONS			
1336	SHOW_OPT_SHIFT	Shows or hides optimized shift information in the report		
187	SHOW_POINT_INFO	Point info / Show point info		
493	SHOW_ROW			
402	SHOW_SPC_CALCS	For SPC		

763	SIMULT_EVAL			
235	SINGLE_POINT	Point mode on dcc scan object		
145	SKIP_NUM	Skip number of loop		
53	SLOT_MIN_MAX_TYPE	Measure slot using 5 (NORM) or 6 (MINMAX) hits		
297	SLOT_NUMBER	Sets the column slot number for a column load and unload		
563	SLOT_TYPE			
109	SLOTVEC_I	Theoretical slot vector i - See the "Slot Vector Data Type" note below.		
110	SLOTVEC_J	Theoretical slot vector j - See the "Slot Vector Data Type" note below.		
111	SLOTVEC_K	Theoretical slot vector k - See the "Slot Vector Data Type" note below.		
43	SNAP_TYPE	Vector features / Surface features		
416	SOLID	For SPC		
403	SPEC_LIMITS	For SPC		
415	SPEC_OFFSET	For SPC		
134	SRC_EXPR	Source expression (assign, subroutine, call sub, basic script, flow control)		
181	STANDARD_DEVIATION	Dimension format standard deviation toggle		
98	START_ANG	Start angle	0	Radian value
466	START_LABEL			

143	START_NUM	Start number for loop		
254	STAT_CALC_TYPE	Calculation type on/off toggle for statistics object		
194	STAT_COUNT	File header statistics count		
253	STAT_NAME_TYPE	Variable name type for statistics object		
1	STATIC_TOGGLE	Used for toggle fields that cannot contain expressions		
391	STATS_DATASOURCE	Data source for statistics object		
453	STATS_DB_TYPE	For stats command database option		
248	STATS_DIR	Stats directory for statistics object		
247	STATS_TYPE	Stats type for statistics object		
405	STDDEV	For SPC		
195	SUB_NAME	Subroutine name		
690	SUMMARY_AXIS			
781	SUMMARY_BONUS			
684	SUMMARY_COLUMN_HDR			
640	SUMMARY_DEV			
685	SUMMARY_FEAT			
779	SUMMARY_MAX			
692	SUMMARY_MEAS			
780	SUMMARY_MIN			

778	SUMMARY_MINUSTOL			
691	SUMMARY_NOMINAL			
776	SUMMARY_OUTTOL			
777	SUMMARY_PLUSTOL			
689	SUMMARY_TBLHDR			
484	SURFACE			
112	SURFVEC_I	Surface vector i		
113	SURFVEC_J	Surface vector j		
114	SURFVEC_K	Surface vector k		
546	SURFVEC_MEAS_I	Surface vector i meas		
547	SURFVEC_MEAS_J	Surface vector j meas		
548	SURFVEC_MEAS_K	Surface vector k meas		

Slot Vector Data Type

Note that SLOTVEC_I, SLOTVEC_J, and SLOTVEC_K are enumerations. They only function within the context of automation.

For an automation example using enumerations, you should use SLOTVEC_I, SLOTVEC_J, and SLOTVEC_K as shown below:

```
   FeatureCommand.PutData(LocSlotVector,FDATA_SLOT_VECTOR,FD
ATA_MEAS,FDATA_PART,"",PLANE_TOP)
   Command.PutText("0",MEAS_SLOTVEC_I,1)
   Command.PutText("1",MEAS_SLOTVEC_J,1)
   Command.PutText("0",MEAS_SLOTVEC_K,1)
```

If you're using pointers or variables directly in the Edit window via the PC-DMIS expression language, you need to use TANGVEC_IJK or ANGVEC_IJK:

- TANGIJK gets the *theoretical* IJK slot vector.

  You can also get the specific values:

  - TANGI gets the I value.
  - TANGJ gets the J value.
  - TANGK gets the K value.

- ANGIJK gets the *measured* IJK slot vector.

  You can also get the specific values:

  - ANGI gets the I value.
  - ANGJ gets the J value.
  - ANGK gets the K value.

For an expression example, to give variables V1 and V2 the theoretical and measured IJK values of the a round slot, you can use the following:

```
ASSIGN/V1=SLTR.TANGIJK
ASSIGN/V2=SLTR.ANGIJK
```

-T-

Number	Data Type Name	Data Type Description	Index	Value String
345	T_VALUE	T value on hits		
572	TARG_A			
590	TARG_EA			
591	TARG_EH			
589	TARG_ER			
516	TARG_EX	Target endpoint x component		

517	TARG_EY	Target endpoint y component		
518	TARG_EZ	Target endpoint z component		
573	TARG_H			
31	TARG_I	Target i - vector		
32	TARG_J	Target j - vector		
33	TARG_K	Target k - vector		
571	TARG_R			
587	TARG_SA			
588	TARG_SH			
586	TARG_SR			
513	TARG_SX	Target start point x component		
514	TARG_SY	Target start point y component		
515	TARG_SZ	Target start point z component		
19	TARG_X	Target x - centroid		
20	TARG_Y	Target y - centroid		
21	TARG_Z	Target z - centroid		
557	TARGET_BLOB_TYPE			
282	TARGET_COLOR	Vision Target Color		

474	TARGET_DIRECTION			
520	TARGET_EDGE_ANGLE			
508	TARGET_EDGE_DENSITY			
712	TARGET_EDGE_EDGEDETECT			
538	TARGET_EDGE_EDGENUM			
537	TARGET_EDGE_EDGESELECT			
717	TARGET_EDGE_GRADIENT			
711	TARGET_EDGE_HEIGHT			
505	TARGET_EDGE_ILLUM			
475	TARGET_EDGE_POLARITY			
504	TARGET_EDGE_SIZE			
507	TARGET_EDGE_STRENGTH			
506	TARGET_EDGE_TOL			
509	TARGET_EDGE_TYPE			
549	TARGET_EDGE_UNDERSCAN			
710	TARGET_EDGE_WIDTH			
715	TARGET_FILTER_AREA			
716	TARGET_FILTER_AREA_SIZE			
713	TARGET_FILTER_CLEAN			

714	TARGET_FILTER_CLEAN_ STRENGTH			
561	TARGET_FILTER_OUTLIER			
562	TARGET_FILTER_OUTLIER_ DISTANCE_THRESHOLD			
599	TARGET_FILTER_OUTLIER_ STD_DEV_THRESHOLD			
523	TARGET_FOCUS			
722	TARGET_SURFACE_ CROSSHAIR_HEIGHT			
722	TARGET_SURFACE_ CROSSHAIR_HEIGHT			
499	TARGET_SURFACE_DURATION			
497	TARGET_SURFACE_HEIGHT			
501	TARGET_SURFACE_HIACC			
490	TARGET_SURFACE_ILLUM			
500	TARGET_SURFACE_MODE			
498	TARGET_SURFACE_RANGE			
511	TARGET_SURFACE_TYPE			
496	TARGET_SURFACE_WIDTH			
564	TARGET_TYPE			
124	TARGSLOT_I	Target slot vector i		
125	TARGSLOT_J	Target slot vector j		

126	TARGSLOT_K	Target slot vector k		
532	TEMPP	Current Part temperature		
529	TEMPX	Current X axis temperature		
530	TEMPY	Current Y axis temperature		
531	TEMPZ	Current Z axis temperature		
163	TEXT_ANALSYIS	Use text analysis on dimension (Yes / No)		
761	TEXTANAL_LABEL_DEV			
758	TEXTANAL_LABEL_MEAS_I			
759	TEXTANAL_LABEL_MEAS_J			
760	TEXTANAL_LABEL_MEAS_K			
755	TEXTANAL_LABEL_MEAS_X			
756	TEXTANAL_LABEL_MEAS_Y			
757	TEXTANAL_LABEL_MEAS_Z			
762	TEXTANAL_LABEL_MINMAX			
566	THEO_A			
609	THEO_A2	For CMT characteristic point 1		
38	THEO_ANGLE	Theoretical angle		
720	THEO_AREA			

555	THEO_DEPTH			
34	THEO_DIAM	Theoretical diameter		
578	THEO_EA			
579	THEO_EH			
284	THEO_END_ANG	Vision Circle Feature		
577	THEO_ER			
13	THEO_EX	Theoretical x - end point		
14	THEO_EY	Theoretical y - end point		
15	THEO_EZ	Theoretical z - end point		
551	THEO_FLUSH			
553	THEO_GAP			
567	THEO_H			
610	THEO_H2	For CMT characteristic point 1		
37	THEO_HEIGHT	Theoretical height		
16	THEO_I	Hit point theoretical i - vector	point number	Numeric value
17	THEO_J	Hit point theoretical j - vector	point number	Numeric value
18	THEO_K	Hit point theoretical k - vector	point number	Numeric value
36	THEO_LENGTH	Theoretical length		

130	THEO_MINOR_AXIS	Theoretical minor access		
718	THEO_PERIMETER			
565	THEO_R			
608	THEO_R2	For CMT characteristic point 1		
575	THEO_SA			
576	THEO_SH			
574	THEO_SR			
283	THEO_START_ANG	Vision Circle Feature		
10	THEO_SX	Theoretical x - start point		
11	THEO_SY	Theoretical y - start point		
12	THEO_SZ	Theoretical z - start point		
35	THEO_WIDTH	Theoretical width		
7	THEO_X	Hit point theoretical x - centroid	point number	X-Coordinate
393	THEO_X2	For CMT characteristic point 1		
8	THEO_Y	Hit point theoretical y - centroid	point number	Y-Coordinate
394	THEO_Y2	For CMT characteristic point 1		
9	THEO_Z	Hit point theoretical z - centroid	point number	Z-Coordinate

395	THEO_Z2	For CMT characteristic point 1		
49	THEOBF_TYPE	Use theoretical values for best fit algorithm (yes / no)		
41	THICKNESS_TYPE	Theoretical or actual thickness	0	"THICKNESS_NONE", "THEO_THICKNESS", or "ACTL_THICKNESS"
594	THICKNESS_TYPE_EDGE	Theoretical or actual thickness		
67	THINNING_TOL	Thinning tolerance for curves		
488	THRESHOLD			
450	TIME_ARG	For SPC		
401	TIME_FILTER	For SPC: filtering based on time stamps		
229	TIP_I	Tip shank vector i component		
230	TIP_J	Tip shank vector j component		
231	TIP_K	Tip shank vector k component		
418	TITLE	For SPC		
349	TOOL_DIAM	Tool diameter		
346	TOOL_X	Tool x value		
347	TOOL_Y	Tool y value		
348	TOOL_Z	Tool z value		

169	TP_MODIFIER	Dimension Position modifier		
726	TP_MODIFIER2	This is because TP_MODIFIER and TP_MODIFIER2 are used in two different groups in Summary Mode.		
400	TRACE_FILTER	For SPC: filtering based on tracefields		
451	TRACE_FILTER_ARG	For SPC		
257	TRACE_NAME	Tracefield name		
258	TRACE_VALUE	Tracefield value		
473	TRACE_VALUE_LIMIT	Trace command character limit for the value field		
255	TRANSFER_DIR	Transfer directory for statistics object		
212	TRIGGER_FORCE	Trigger force for option probe object		
469	TRIGGERPLANE			
463	TRIGGERTOLERANCE			
464	TRIGGERTOLVALUE			
131	TWO_D_THREE_D_TYPE	2d or 3d		

-U-

Number	Data Type Name	Data Type Description	Index	Value String
318	U_HITS	Number of U points for surface		
417	U_L_BOUNDS	For SPC		
406	UCL_LCL	For SPC		
172	UNIT_TYPE	Unit type (inch / mm)		
211	UP_FORCE	Up force for option probe object		
127	UPDATEVEC_I	Update vector i		
128	UPDATEVEC_J	Update vector j		
129	UPDATEVEC_K	Update vector k		
431	USE_3DFILTER			
139	USE_AXIS	Flag for iterative alignments and start TP dimension		
753	USE_AXIS2 (Deprecated)			
968	USE_BOUNDARY_OFFSET	Enables or disables using the boundary offset as a minimum distance from the boundary (the edge) where the hits are automatically placed during a Void Detection. If set to NO, the tip's radius value is the minimum distance.		YES/NO
541	USE_HSSDAT			
542	USE_STARTENDDELAY			
281	USE_THEO			
44	USEPIN_TYPE	Use pin vector or normal vector		

738	USETWODEVIATIONS			

-V-

Number	Data Type Name	Data Type Description	Index	Value String
319	V_HITS	Number of V points for surface		
363	VIDEO_GAIN			
366	VIDEO_LASERLIGHT1			
367	VIDEO_LASERLIGHT2			
365	VIDEO_LEDLIGHT			
370	VIDEO_LSEG			
364	VIDEO_OFFSET			
371	VIDEO_XSEG			
369	VIDEO_YEND			
368	VIDEO_YORIGIN			
372	VIDEO_YSEG			
512	VOID_DETECT	Turn off void detection	0	"YES" or "NO"

-W-

Number	Data Type Name	Data Type Description	Index	Value String
322	WAVE_FILE	Feature Locator Wave File		
137	WEIGHT	Weight value for Best Fit alignments		
154	WORK_PLANE	Workplane		

-X-

Number	Data Type Name	Data Type Description	Index	Value String
146	X_OFFSET	X Offset for loops		

-Y-

Number	Data Type Name	Data Type Description	Index	Value String
147	Y_OFFSET	Y Offset for loops		

-Z-

Number	Data Type Name	Data Type Description	Index	Value String
148	Z_OFFSET	Z Offset for loops		
404	ZONES	For SPC		

Using a Type Index to Display Specific Data

Type indexes let you pull in very specific data into your report. For example, if you had a cylinder feature with eight hits, you could use a type index to grab the second hit's measured X value. To use a type index, simply append a colon and a number to the expression to get that hit or that row of dimension data.

Type Indexes to Display Specific Data

To demonstrate how this works, if you typed:

```
=MEAS_X
```

PC-DMIS would not use a type index (the type index would default to 0) and PC-DMIS would display the measured X value of the feature's center point.

But, if you typed:

```
=MEAS_X:1
```

PC-DMIS would display the measured X value of the first hit.

If you typed:

```
=MEAS_X:2
```

PC-DMIS would retrieve the measured X value of the second hit.

Repeating Rows to Display all Data

Similar to type indexes, In the GridControlObject you can define a row as a "repeating row". With repeating rows, the character "N" evaluates to the current repeat index of the row being drawn. When you define a repeating row, you can also define a repeating row expression that, when evaluated, determines how many times the row repeats.

So, if you wanted to display all of the measured X values for all hits in a feature, you could define a single repeating row with the following expression:

```
=MEAS_X:N
```

You could then set the repeating row expression to "=N_HITS" to get the total number of hits. PC-DMIS would continue to repeat the row with subsequent hit data until the total number of hits was displayed.

Using a Prefix Number to Access Reference Feature Data

A report expression can take an optional prefix number inside curly brackets to indicate from which reference command the data should be taken.

For example, if you create a label template for a Roundness dimension, and you want to show the measured diameter on the label template, you could use this expression:

```
={1}MEAS_DIAM()
```

Notice the {1}. This indicates that the data needs to come from the first reference command (the feature) instead of the command for which the template is being created (the dimension).

If there is more than one reference feature used, the number within the curly braces represents the feature from which PC-DMIS will pull the information. For example, a Distance dimension uses two features to compute the distance. Using {1} would pull information from the first reference feature and {2} would pull it from the second reference feature.

Predefined Constants

The reporting expression language also uses some predefined constants for the type indices used with the DIMENSION_TABLES data type.

```
LINE1_SIZE_TABLE = 1
LINE2_POSITION_TABLE = 2
LINE3_POSITION_TABLE = 3
DATUM_SHIFT_TABLE = 4
SUMMARY_TABLE = 5
LINE2_ORIENTATION_TABLE = 7
LINE3_ORIENTATION_TABLE = 8
LINE2_BASIC_DIMENSION_TABLE = 9
LINE3_BASIC_DIMENSION_TABLE = 10
TRUE_POSITION_TABLE = 11
DIMENSION_TABLE = 12
DIMENSION_WITH_BONUS_TABLE = 13
```

NEWLINE

NEWLINE is used to add a carriage return to a text expression:

This expression,

```
="This is line 1" + " and this is line 2"
```

would appear as,

```
This is line 1 and this is line 2
```

on the report.

The expression,

```
="This is line 1" + NEWLINE + " and this is line 2"
```

would appears as,

```
This is line 1

and this is line 2
```

on the report.

N:

When you use repeating rows in a grid, N evaluates to the current row count in the repeating rows. This is useful when placing expressions that use a data type more than once in a repeating row.

Below is a sample grid that shows hit values in a table. N is used in the expressions for the repeating row. When the expressions for the first instance of the repeating row are evaluated, N equals 1, and so the X, Y, and Z values of the first hit are shown. When the second row is added, N equals 2, and so the X, Y, and Z values of the second hit are shown.

Hit #	Measured X	Measured Y	Measured Z
=N	=MEAS_X:N	=MEAS_Y:N	=MEAS_Z:N

Expressions Used in Standard Templates

In the various standard report and label templates, PC-DMIS uses several reporting expressions to display different pieces of information or to format the display of information. For example, consider the code behind feature.lbl, a standard label template that ships with PC-DMIS.

In this template, there are several reporting expressions (=LOADSTR, =DIM_MEASURED, =NOMINAL and so on) that determine what information PC-DMIS should display in the cells.

Common Expressions

The following list describes the most common expressions used in the various standard templates:

Some of these functions are also discussed in the "Functions and Operators" topic.

- =<Expression>:N - This expression allows you to display repeated information. See "Using a Type Index to Display Specific Data" and "Predefined Constants" for more information on repeating rows.
- =AXIS - This expression displays the current axis name (XAXIS, YAXIS, or ZAXIS).
- =DEVIATION_ANGLE - This expression displays a decimal value that represents the deviation angle in degrees of a Position dimension. PC-DMIS uses this value to draw the radial graphic for the Position dimension templates.
- =DEVPERCENT_NOM - This expression displays a decimal value that represents the percentage of the total tolerance range that marks the 0 deviation (nominal) position on the linear graphic. For a bilateral tolerance where the +/- values are equal, this value would be 50.0 (percent); the center of linear graphic represents 0 deviation. For a non-bilateral tolerance, this value would be 0.0; the leftmost side of linear graphic represents 0 deviation.
- =DEVPERCENT2 - When you set USETWODEVIATIONS to TRUE (see USETWODEVIATIONS below), this expression displays a decimal value representing the percentage of the + tolerance used by the max deviation of a bilateral profile reported as form and location. The percentage of the – tolerance used by the min deviation is given by DIM_RPT_DEVPERCENT.
- =DIM_DEVIATION - This expression displays how much a dimension deviates from the nominal.
- =DIM_OUTTOL - This expression displays the out of tolerance value for a dimension.
- =DIM_MAX - This expression displays the maximum value for the given axis among all points related to the input feature.
- =DIM_MEASURED - This expression displays a dimension's measured value.
- =DIM_MIN - This expression displays the minimum value for the given axis among all points related to the input feature.
- =DIM_RPT_DEVPERCENT - This expression displays a decimal value representing the percentage of the total tolerance range that marks the deviation of this dimension.

- =DIM_RPT_GRAPHIC - This expression displays an integer value representing the type of graphic control PC-DMIS should display for the dimension.

  0 - Report no graphic control.

  1 - Report the linear graphic control.

  2 - Report the radial graphic control.

- =DIM_RPT_ISBILATERAL - This expression determines whether or not the linear graphic represents a bilateral tolerance (+/- tolerance value) such as the size of a hole or profile reported as form and location. Examples of tolerances that are not bilateral (single tolerance value) are form tolerances such as circularity, flatness, straightness, and profile reported as form only. The value can be "0" (FALSE) or "1" (TRUE).

- =DIM_RPT_NUMZONES - This expression displays a decimal value indicating the number of tolerance zones (from 0 deviation to the maximum allowed deviation) to be drawn on the linear graphic. You can set this in the **Edit Dimension Colors** dialog box (select **Edit | Graphic Display Window | Dimension Colors**). You can assign each zone a unique color to show by the color of the linear graphic how much of the tolerance was used by the deviation of the dimension.

- =F_PLUS_TOL - This expression displays a feature's plus tolerance.

- =F_MINUS_TOL - This expression displays a feature's minus tolerance.

- =ID - This expression displays the feature or dimension ID.

- =LOADSTR - This expression loads a string from a table of stored strings within PC-DMIS. See "Loading Strings from PC-DMIS" for more information.

- =NOMINAL - This expression displays the nominal data for a feature.

- =Page() - This expression displays the current page number of the Report window.

- =Pages() - This expression displays the total number of pages of the Report window.

- =TOL - This function colors any text or expressions.

- =UNIT_TYPE - This expression displays the unit of measurement for the feature or dimension.

- =USETWODEVIATIONS - This expression determines whether or not the legacy dimension label template's linear graphic uses two deviation values. A bilateral profile reported as form and location displays how much of the + tolerance was used by the max deviation and how much of the – tolerance was used by the min deviation. The value can be "0" (FALSE) or "1" (TRUE).

Combined Expressions

You can also combine expressions or use them together with other expressions, as shown in this code string taken from the Legacy_Dimension_Cad.lbl label template:

```
=TOL(DIM_DEVIATION:N,DIM_OUTTOL:N,0.0,0.0)
```

This uses the =TOL function to display the dimension's deviation value and give it the Error color (usually red).

You can find another example in the standard report templates in PC-DMIS versions 4.2 and higher. The **Text** object at the bottom of those templates uses this combined expression in the **Text** property to display the current page number along with the total number of pages:

```
=page() + " Of " + pages()
```

Page six of a 10-page report would read as follows at the bottom of the page in the Report window:

6 Of 10

Using PC-DMIS ActiveX Controls

This series of topics provides an example of how to set up the properties of various PC-DMIS ActiveX controls and use them in a label template to display dimensional information.

- Adding an ActiveX Control
- Passing Information to an ActiveX Control
- Some PC-DMIS ActiveX Controls

The topics focus on the unique ActiveX properties for each control.

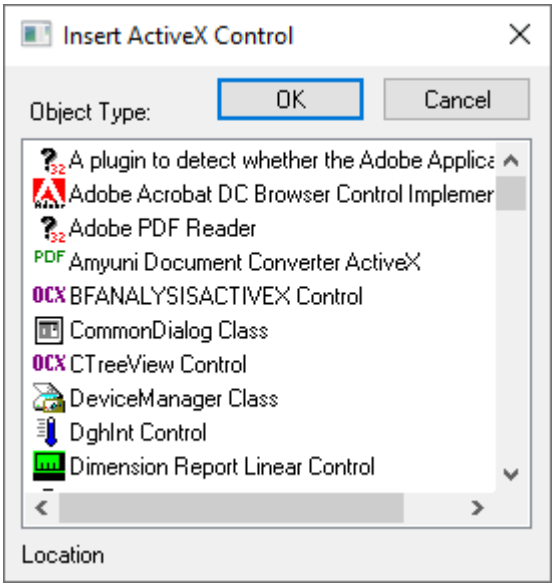
Adding an ActiveX Control

You can add an ActiveX control to a label template in the Label Template Editor in two ways:

- **ActiveX Object from Object Bar** - Click the **ActiveX** icon in the editor's Object Bar to add an ActiveX control directly to the label template. You can then define the area where the control will populate. (See the "ActiveX Object" topic.)

- **ActiveX Object from GridControlObject** - In a GridControlObject, double-click on the object to select it, and then right-click the object to show the **Grid Properties** dialog box. From that dialog box, set the **Cell Type** to **ActiveX**. This activates the **Select** button. Click **Select**. (For information, see the "GridControlObject" topic.)

Regardless of what method you use, PC-DMIS displays the **Insert ActiveX Control** dialog box.



Insert ActiveX Control Dialog Box

Select the desired control from the list and click **OK**.

This dialog box lists all the controls on your system, not just those controls added by PC-DMIS. If you want to use a third party control in a label template, you would add it in the same way.

Passing Information to an ActiveX Control

For all of the PC-DMIS ActiveX controls described here, the software passes information into the control using the EventReportData event. PC-DMIS encapsulates the specific dimensional information into an object called ReportData (see the "ReportData Object" topic in the PC-DMIS 2026.2 Automation Object Library documentation for additional information). For example, if you look at the Rule Tree Editor for the TextOnly.rtp report template, you will find for the Dimension Location item that one of the rules is:

Use Template "legacy_dimension.lbl"

The Legacy_dimension.lbl label template receives the dimensional information about the location dimension via the ReportData object. The information is now in the label but it has not yet been passed to the ActiveX control.

To understand how this information gets passed:

1. Open the legacy_dimension.lbl label template in the Label Template Editor. You will see that it contains a GridControlObject named **ActiveX12**.
2. Right-click to display the **Properties** dialog box for the object.
3. Double-click on one of the cells in this grid control to put it in edit mode.
4. Click the cell in the lower right corner. This cell uses the **Dimension Report Linear** control. Notice that the **Properties** dialog box has four categories of properties (**Standard**, **Advanced**, **Events**, **ActiveX**). Normally, objects only have three categories (**Standard**, **Advanced**, **Events**). This fourth category, **ActiveX**, is unique to ActiveX controls and is the area we will be concentrating on in the following sections.
5. Expand the **Events** section.
6. Access the **EventReportData** property. The VBA Mini Editor appears. Notice that the editor contains this line of code:

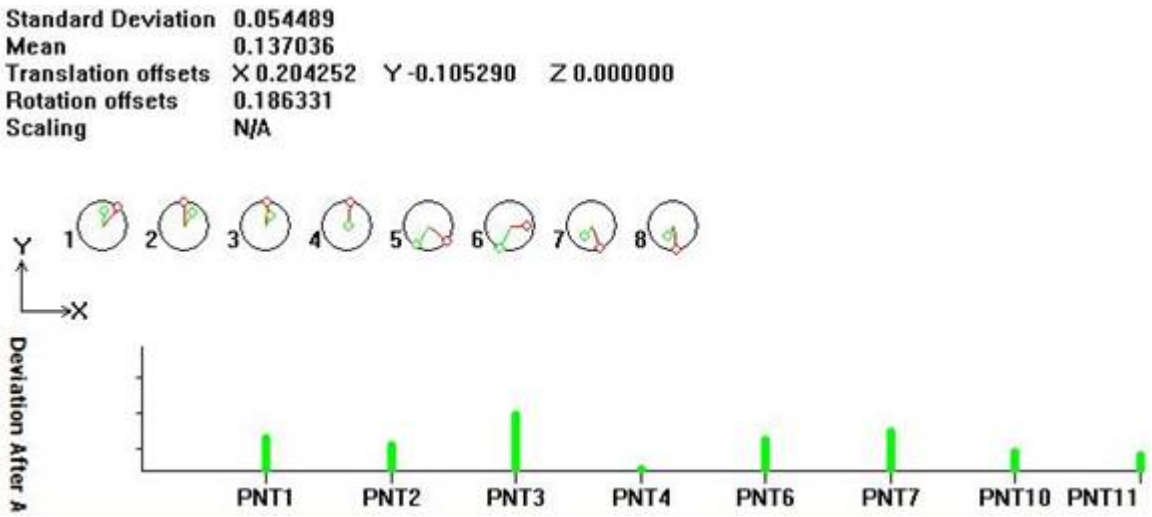
```
This.X.EventReportData ReportData
```

This code is the mechanism that passes ReportData (the dimensional information) to the ActiveX control. Therefore, whenever you add an ActiveX control to a label template, you will need to set the EventReportData property to do the same thing by using that line of code.

Some PC-DMIS ActiveX Controls

The following sections describe some PC-DMIS reporting ActiveX controls and their properties. Note that the DimAnalysisActiveX Control is not discussed here. Although it appears in the ActiveX list, it is used internally by the Analysis object in the Label Template Editor.

BFAnalysisActiveX



Property	Description	Example
AfterColor	The color of the last bar.	tbd [defaults to 65280 (Green)]
BeforeColor	The color of the first bar.	tbd [defaults to 255 (Blue)]
NumberFeatures	The number of features shown on the X axis.	tbd

`AfterColor` and `BeforeColor` take long values representing a color.

They use the following formula:

(Red * 256 * 256) + (Green * 256) + Blue.

So Red would be 16711680; that is (255 * 256 * 256) + (0 * 256) + 0; Green would be 65280; that is (0 * 256 * 256) + (255 * 256) + 0; Blue would be 255; that is, (0 * 256 * 256) + (0 * 256) + 255.

Example Label Template: BFANALYSISACTIVE1 in Best_Fit_Analysis.lbl.

Dimension Report Linear

Property	Description	Example
DeviationNominalPercent	See "Expressions Used in Standard Templates" for a description of the example expression.	=DEVPERCENT_NOM:N
DeviationPercent	See "Expressions Used in Standard Templates" for a description of the example expression.	=DIM_RPT_DEVPERCENT:N
DeviationPercent2	See "Expressions Used in Standard Templates" for a description of the example expression.	=DEVPERCENT2:N
Is Bilateral	See "Expressions Used in Standard Templates" for a description of the example expression.	=DIM_RPT_ISBILATERAL:N
UseTwoDeviations:	See "Expressions Used in Standard Templates" for a description of the example expression.	=USETWODEVIATIONS:N

Example Label Template: Bottom right cell in Legacy_Dimension.lbl.

Dimension Report Linear2

Property	Description	Example
Deviation:	See "Expressions Used in Standard Templates" for a description of the example expression.	=DIM_DEVIATION:N
Negative Tolerance:	See "Expressions Used in Standard Templates" for a description of the example expression.	=F_PLUS_TOL:N
Positive Tolerance:	See "Expressions Used in Standard Templates" for a description of the example expression.	=F_MINUS_TOL:N

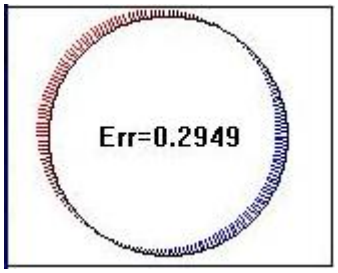
Example Label Template: Not currently used in any standard label template.

Dimension Report Radial

Property	Description	Example
Cylindrical Deviation	See "Expressions Used in Standard Templates" for a description of the example expression.	=DIM_RPT_DEVPERCENT:101
Cylindrical Deviation Angle:	See "Expressions Used in Standard Templates" for a description of the example expression.	=DEVIATION_ANGLE:101

Example Label Template: Bottom right cell in Legacy_Dimension_True_Position.lbl.

FeatureAnalysisActiveX



There are no properties to set up for this control. It uses the data passed to it in ReportData.

Example Label Template: FEATUREANALYSISA1 in Feature.lbl.

DataFileFormatControl

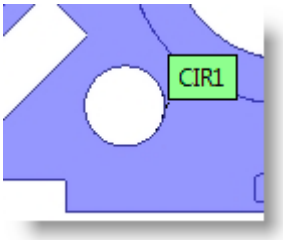
Property	Description	Example
DataFileName	The .dat file (logo.dat, elogo.dat, or header.dat) to interpret. If you use a .dat file outside of the PC-DMIS installation directory, be sure to include the full path to the file.	If you typed logo.dat, PC-DMIS would interpret the keywords inside that .dat file displaying the specified bitmap, date, time, and so forth according to the .dat file keywords in the file.

Example Label Template: None. See the "Reporting Using .DAT File Keywords" topic.

Tutorial - Using Scripting to Enhance your Report

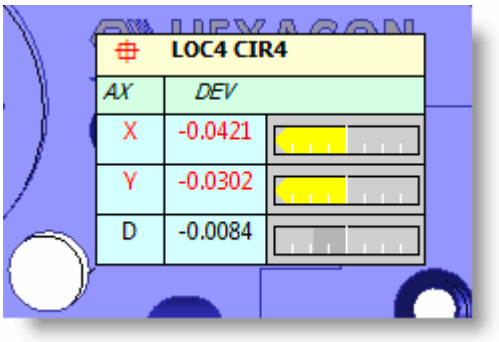
This tutorial demonstrates some of the capability available in PC-DMIS Reporting when you couple it with the power of scripting.

This tutorial shows you how to create a report template that displays a CADOnly type report, except that if the dimension is in-tolerance, a label template similar to Reference_ID.lbl is used.



A Sample in-tolerance feature.

If it is out-of-tolerance, PC-DMIS uses the Legacy_Dimension_CAD.lbl label template. Additionally, the Reference_ID.lbl-like label uses the dimension color for its background color.



A Sample out-of-tolerance feature.

This tutorial is not a complete solution since it does not cover needed modifications to labels used by Geometric Tolerance commands. For this to be a complete solution, you may want to perform similar steps to GEO_TOL_LABEL.lbl so that your Geometric Tolerance commands follow the same pattern.

Before you start the tutorial, you may want to review the "Summary of Tutorial Steps" and "Things to Know" topics below.

Summary of Tutorial Steps

1. Copy CADONLY.RTP to CADONLYREF_ID.RTP.
   CADONLYREF_ID.RTP will be the name of your new report template.

2. Copy REFERENCE_ID.LBL to REFERENCE_ID_COLOR.LBL.
   REFERENCE_ID_COLOR.LBL will be your new label template used for in-tolerance dimensions.

3. Add VB Script to REFERENCE_ID_COLOR.LBL.
   Using the Label Template Editor, you will add VB script code to the new label template, REFERENCE_ID_COLOR.LBL, to set the background color based on the dimension color.

4. Import the CAD2.RUL rules into CADONLYREF_ID.RTP.
   In this step, you will import rules contained in the .RUL file into your new report template, CADONLYREF_ID.RTP. The .RUL file contains the set of rules used by the CAD portion of the TextAndCAD report, and it uses REFERENCE_ID.LBL to display the dimension. You will modify rules that uses REFERENCE_ID.LBL to use new rules and the new label template, REFERENCE_ID_COLOR.LBL.

5. Add VB Script Code to CADONLYREF_ID.RTP
   Using the Report Template Editor, you will add script code to CADONLYREF_ID.RTP at each place in the rule tree that references REFERENCE_ID.LBL to check to see if

the dimension is in tolerance. If it is in tolerance, the code will cause it to use REFERENCE_ID_COLOR.LBL. If it is not, the code will cause it to use LEGACY_DIMENSION_CAD.LBL.

> If you are creating a new FCF_LABEL.LBL, e.g. FCF_LABEL_COLOR.LBL, at this time you would also change the Feature Control Frame rule to use FCF_LABEL_COLOR.LBL instead of FCF_LABEL.LBL if it is in-tolerance and use another label, like LINE2.LBL, if it is out-of-tolerance.

6. Modify the Rules to Use the VB Script Code for Remaining Dimensions.
   At this point the new rules will only be established for Dimension Location. In this step you will use the Copy and Paste functions inside the Rule Tree Editor so that other dimension types can use the same rules.

7. Test CADONLYREF_ID.RTP in the Report Window.
   In this step you select the CADONLYREF_ID.RTP template and test it inside the Report window. By the end of this tutorial you will have a report template that causes in-tolerance dimensions to only display the feature name with a background dimension color that matches a color from the dimension color key at the bottom of the page. Out-of-tolerance dimensions will use the LEGACY_DMENSION_CAD.LBL label.

Things to Know

- The "ReportData Object" topic in the PC-DMIS 2026.2 Automation Object Library documentation provides a description of methods for this object. This example uses several ReportData methods.
- To get Dtype information, right click on a dimension in the Edit Window. At the bottom of the displayed menu is the item "Change Pop-up Display”. Select that and then select "Data Type Information”. Now when you hover the cursor over an item in the Edit Window, a help tip displays Dtype information. For example, if you hover over an OUTTOL value, the tooltip description displays "Out tol (DIM_OUTTOL – 344), 0”. The Dtype is DIM_OUTTOL and the Dtype number is 344. This example uses the Dtype numbers. Future versions of PC-DMIS will also allow the use of the Dtype itself.
- The VBS Mini-Editor accepts the MsgBox statement. You may want to use this statement to help debug your script. Information that you display via the MsgBox statement will be displayed during report generation.
- The VBS Mini-Editor is a "mini" editor. If you are familiar with Visual Basic / Visual Studio, note that this editor and the BASIC language used do not have all of the capabilities of the larger BASIC systems.

Step 1: Copy CADONLY.RTP to CADONLYREF_ID.RTP

In this step, CADONLYREF_ID.RTP will be the name of your new report template.

Inside Windows Explorer, create a copy of the CADONLY.RTP file in your Reporting sub-directory, and rename the copy to CADONLYREF_ID.RTP.

You can find the Reporting folder at this location. This folder contains all the templates that the Report window uses:

C:\Users\Public\Documents\Hexagon\PC-DMIS\<version>\Reporting

<version> represents the PC-DMIS version. If you want to customize existing templates or create new ones, do so in that directory.

If you need to restore any modified template to its factory settings, you can copy it from the DefaultReportingTemplateBackup directory located within the PC-DMIS installation directory and paste it the above Reporting directory.

Do not modify the templates in the DefaultReportingTemplateBackup directory.

Step 2: Copy REFERENCE_ID.LBL to REFERENCE_ID_COLOR.LBL

In this step, REFERENCE_ID_COLOR.LBL will be your new label template used for in-tolerance dimensions.

From the same Reporting subdirectory, create a copy of REFERENCE_ID.LBL, and rename the copy to REFERENCE_ID_COLOR.LBL.

Step 3: Add VB Script to REFERENCE_ID_COLOR.LBL

In this step, you will use the Label Template Editor to add VB script code to the new label template, REFERENCE_ID_COLOR.LBL, to set the background color based on the dimension color.

1. Select **File | Reporting | Edit | Label Template**, and open REFERENCE_ID_COLOR.LBL. The label opens. It contains a GridControlObject with one row and one column. If you double-click on the cell, you will see an expression inside which reads:

   ```
   =REF_ID:1
   ```

   This means the label will display the ID of the reference feature.

2. Right-click on the editor to display the **Properties** dialog box.

3. Select **GridControlObject1** from the list of objects.
4. In the **Properties** dialog box, expand **Events**.
5. Click [None] to the right of the **EventReportData** property. The **VBS Mini-Editor** appears. You may need to expand the **VBS Mini Editor** to be wider so that you can see the entire Sub statement toward the top.

   You can see that the ReportData object is passed in. This is where PC-DMIS extracts information about passed in dimensions. For additional information on the ReportData object, see the "ReportData Object" topic in the PC-DMIS 2026.2 Automation Object Library documentation. Also, if you need to reference the GridControlObject, you can use "This" as a shortcut to the object.

6. Insert this code into the **VBS Mini-Editor**:

```
   Dim I As Integer
   Dim MaxIndex As Integer
   Dim MaxDev As Double
   Dim CurrentDev As Variant
   Dim Dev As Variant
   Dim PTol As Variant
   Dim MTol As Variant
   Dim iReturn As Boolean   ' Initialize Max deviation
and Max Index
   MaxDev = 0.0
   MaxIndex = 1   ' Get the number of axes for this
dimension
   ' 132 is the Dtype AXIS
   Count = ReportData.GetCount(132)   ' Loop through to
find the largest deviation
   ' When the loop is complete, MaxIndex is the index to
the
   ' largest deviation
   For I = 1 to Count
      ' 340 is the Dtype DIM_DEVIATION
      CurrentDev = ABS(ReportData.GetValue(340, I))
      If CurrentDev > MaxDev Then
         MaxDev = CurrentDev
         MaxIndex = I
      End If
   Next I   ' Using MaxIndex, acquire that axes
deviation, + Tol and - Tol
   ' 167 is Dtype F_PLUS_TOL; 168 is Dtype F_MINUS_TOL
   Dev = ReportData.GetValue(340, MaxIndex)
   PTol = ReportData.GetValue(167, MaxIndex)
   MTol = ReportData.GetValue(168, MaxIndex)   ' Use
this information to adjust the background color of the
grid cell
   iReturn = This.SetCellBackgroundColor(0, 0,
ReportData.GetTolColor(Dev,PTol,MTol))
```

Explanation of Code:

This code uses the numeric values for Dtypes. The variable MaxDev is initialized to zero and when a larger deviation is found, it is placed in MaxDev. After the For/Next loop finishes, MaxDev holds the maximum deviation for this dimension and MaxIndex holds the index to that axis. The statement `ReportData.GetCount` gets the number of axes for this dimension. The For/Next loop loops through each axis.

`ReportData.GetTolColor` gets the tolerance color for the specified deviation and tolerance values. We use MaxIndex to get the deviation and tolerance values for the axis with the largest deviation. `GetTolColor` returns the color values as a Long.

We use `"This"` to reference the `GridControlObject1` object. `SetCellBackgroundColor` sets the background color for a specific cell. Arguments 1 and 2 are the cell's row and column numbers. The first cell in a row and column of a grid control is row / column zero. The third argument is the color that you wish to be the background color for this cell. It is also a Long value.

7. Once you have added the code, click **OK**. The Mini-Editor checks for any syntax errors. If you have an error in your code, PC-DMIS displays a message at this time. If there aren't any errors, the **VBS Mini-Editor** closes.
8. Select **File | Save** to save your new copy of REFERENCE_ID_COLOR.LBL.
9. Select **File | Close**. This closes the Label Template Editor.

Step 4: Import the CAD2.RUL rules into CADONLYREF_ID.RTP

In this step, you will import rules contained in the .RUL file into your new report template, CADONLYREF_ID.RTP. The .RUL file contains the set of rules used by the CAD portion of the TextAndCAD report, and it uses REFERENCE_ID.LBL to display the dimension. You will modify rules that uses REFERENCE_ID.LBL to use new rules and the new label template, REFERENCE_ID_COLOR.LBL.

1. Select **File | Reporting | Edit | Report Template**, and open CADONLYREF_ID.RTP.
2. Display the **Properties** dialog box.
3. Select the **CadReportObject1** object from the list of objects.
4. In the **Properties** dialog box, expand the **Standard** list and click **Rules** next to the **Rule Tree** property. The **Rule Tree Editor** opens.
5. Click the **Import** button, select CAD2.RUL, and then click **Open**. Your report template now includes the same rules that are used in the CAD portion of the TextAndCAD report template.

6. In the **Rule Tree Editor**, click on **Dimension Location**, and select the rule that says **Use Template "Reference_ID.lbl"**.
7. Click **Edit**. The **Edit Rule** dialog box appears.
8. Change the **Label Name** from "Reference_ID.lbl" to "Reference_ID_Color.lbl". You can do this by typing or by clicking the **...** button and navigating to this file.

Step 5: Add VB Script Code to CADONLYREF_ID.RTP

In this step, using the **Report Template Editor**, you will add script code to CADONLYREF_ID.RTP at each place in the rule tree that references REFERENCE_ID.LBL to check to see if the dimension is in tolerance. If it is in tolerance, the code will cause it to use REFERENCE_ID_COLOR.LBL. If it is not, the code will cause it to use LEGACY_DIMENSION_CAD.LBL.

1. While still In the **Edit Rule** dialog box, click **Use Additional Expression or Script**, select the **Script** option, and then click the **Edit** button. The **VBS Mini-Editor** appears.
2. Insert the following code into the VBS Mini-Editor:

```
DimCount As Integer
DimCurrentOutTol As Variant
DimI As Integer

' iRetVal = 1 if In-Tolerance;iRetVal = 0 if Out-of-
Tolerance

iRetVal= 1

'132 = Dtype AXIS

Count= ReportData.GetCount(132)

' Loop through each axis to see if OutTol is non-zero

ForI = 1 to Count

            ' 344 = Dtype DIM_OUTTOL
            CurrentOutTol = ReportData.GetValue(344, I)
            If CurrentOutTol > 0 Then
              iRetVal = 0
              Exit For
            End If
NextI
```

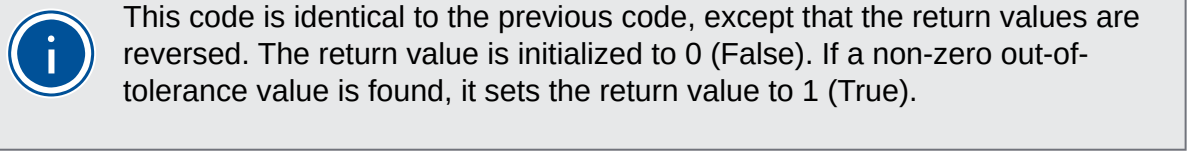
Explanation of Code:

This code uses the numeric values for Dtypes. It loops through each axis of this dimension and looks at the out-of-tolerance value. The return value is initialized to 1 (True). If a non-zero out-of-tolerance value is found, it sets the return value to 0 (False).

3. Once you have added the code, click **OK**. The Mini-Editor checks for any syntax errors. If you have an error in your code, PC-DMIS displays a message at this time. If there aren't any errors, the **VBS Mini-Editor** closes.
4. Click the **Add** button to define a new rule. The **Edit Rule** dialog box appears.
5. In the **Label Name** box, select or type LEGACY_DIMENSION_CAD.LBL.
6. Again, click **Use Additional Expression or Script**, select the **Script** option, and then click the **Edit** button. The **VBS Mini-Editor** appears.
7. In the **VBS Mini-Editor**, insert the following code:

```
Dim Count As Integer
Dim CurrentOutTol As Variant
Dim I As Integer

' iRetVal = 1 if In-Tolerance; iRetVal = 0 if Out-of-
Tolerance

iRetVal = 0

' 132 = Dtype AXIS

Count = ReportData.GetCount(132)

' Loop through each axis to see if OutTol is non-zero

For I = 1 to Count

            ' 344 = Dtype DIM_OUTTOL
            CurrentOutTol = ReportData.GetValue(344, I)
            If CurrentOutTol > 0 Then
              iRetVal = 1
              Exit For
            End If

Next I
```

Explanation of Code:



8. Once you have added the code, click **OK**. The **VBS Mini-Editor** checks for any syntax errors. If you have an error in your code, PC-DMIS displays a message at this time. If there aren't any errors, the **VBS Mini-Editor** closes.

9. Click **OK** to close the **Edit Rule dialog box.**

Step 6: Modify the Rules to Use the VB Script Code for Remaining Dimensions

In this step you will use the **Copy** and **Paste** functions inside the **Rule Tree Editor** so that the other legacy dimension types can use the same rules.

At this point, the **Rule Tree Editor** is open and the **Dimension Location** is selected.

1. Click and highlight both rules for the selected dimension.

2. Click the **Copy** button.

3. Go through each legacy dimension in the rule tree, and check to see if it has this rule: Use Template "Reference_Id.lbl".

4. If it does, select the rule, and click the **Paste** button. This adds the copied rules to the selected dimension type.

5. Once you have modified all the dimension types to use the new rules, click **OK** to close the **Rule Tree Editor**.

6. Select **File | Save** to save your new copy of CADONLYREF_ID.RTP.

7. Select **File | Close**. This closes the Label Template Editor.

Step 7: Test CADONLYREF_ID.RTP in the Report Window

In this step you select the CADONLYREF_ID.RTP template and test it inside the Report window.

1. Ensure that your measurement routine uses legacy dimensions.

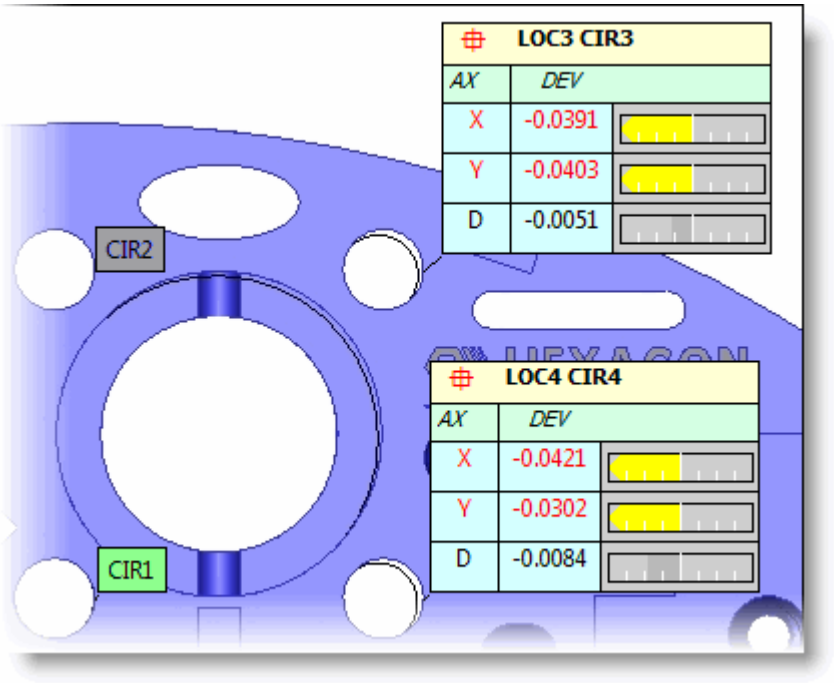
2. Execute your measurement routine.

3. Display the Report Window by selecting **View | Report Window**.

4. From the **Reporting** toolbar, click on the **Template Selection Dialog** icon . The **Report Templates** dialog box appears.
5. Click the **Add** button.
6. From the **Open** dialog box, select CADONLYREF_ID.RTP, and click **Open**. The report template is added into the **Report Templates** dialog box.
7. From the **Report Templates** dialog box, select CADONLYREF_ID.RTP again, and click **Open**.
8. PC-DMIS loads the report template. The Report window will contain a report of your measurement routine results using the newly created CADONLYREF_ID.RTP report template.

Notice that in-tolerance dimensions only display the feature name and that the background color is the dimension color and matches a color on the dimension color key at the bottom of the page. Out-of-tolerance dimensions use the LEGACY_DIMENSION_CAD.LBL label.

The following screen capture demonstrates what such a report might look like:



A sample report showing in-tolerance features (CIR1 and CIR2) and out-of-tolerance features (CIR3 and CIR4).

Congratulations, you've completed the tutorial!

Working with PDF 3D Files

PC-DMIS can print your report or export your CAD model as a PDF 3D file.

- To print your report, see "Printing the Report Window" in the "Using Basic File Options" chapter.
- To export your CAD model, see "Exporting to a PDF 3D File".

Supported Items and Known Limitations

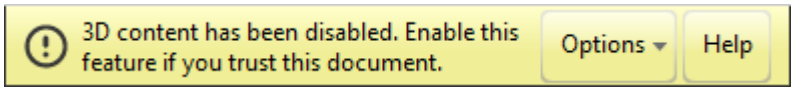
The PDF 3D file supports the Analysis command, Snapshot (live) command, CADReportObject, and the AnalysisWindowObject.

This file format does not support Legacy HyperReport objects or the ability to append to the file.

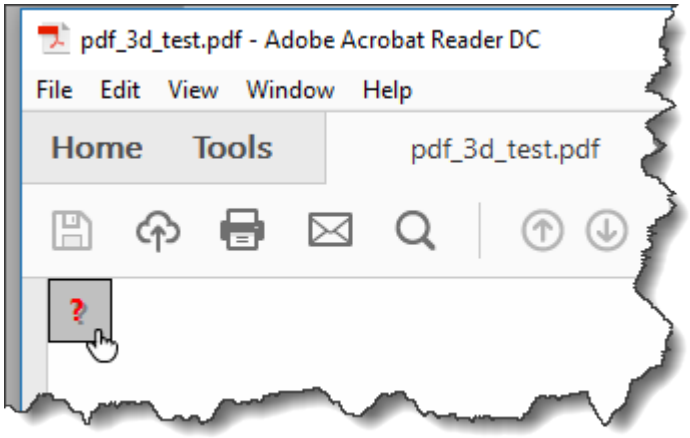
Viewing the CAD Model in the File

Once the PDF 3D file is generated, you can use any PDF viewer that is capable of supporting CAD data to view and interact with the CAD model. You can perform pan, zoom, rotate, and part-selection operations.

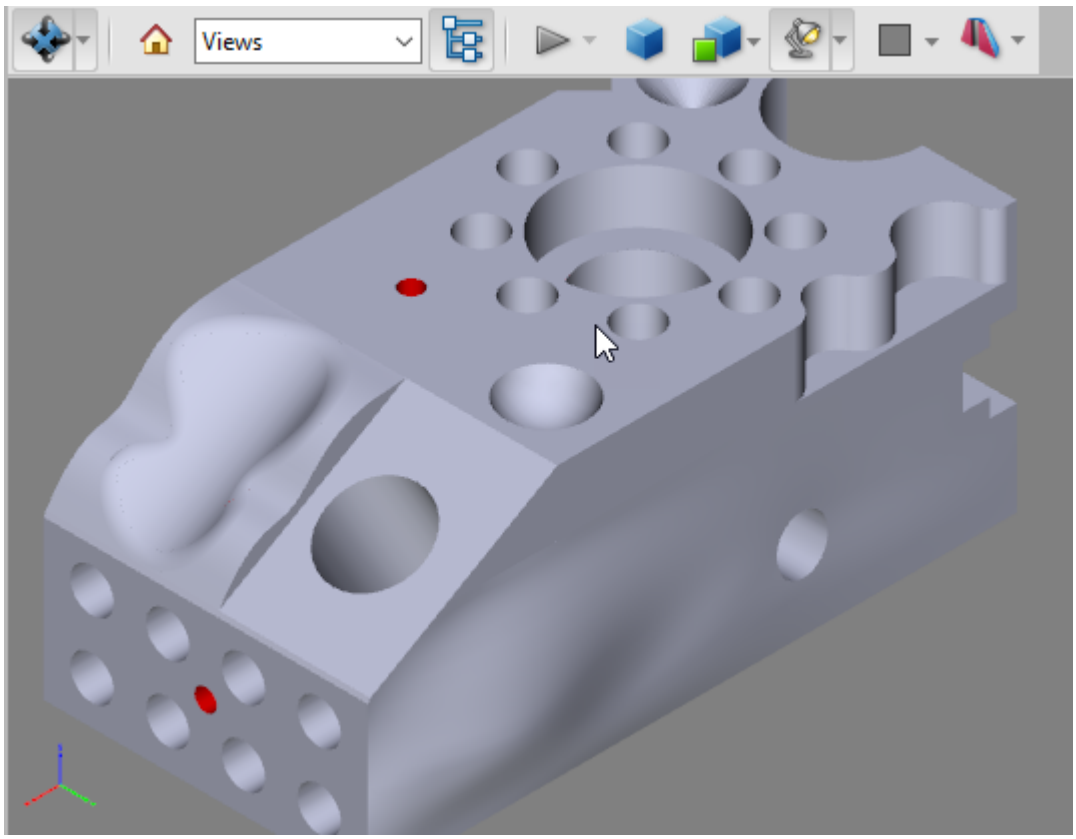
1. Open the exported PDF file inside of an Adobe viewer that supports 3D content. If you have not enabled 3D content, you must first trust the file.



2. Use the **Options** button to trust the file.
3. Once you trust the file, click the red question mark icon to open the CAD model.



4. With the CAD model visible, hover your pointer over the CAD model. Use the buttons on the toolbar to interact with the model.



Additional Information

For information on the different operations you can perform with a PDF 3D file, consult this Adobe documentation:

https://helpx.adobe.com/acrobat/using/displaying-3d-models-pdfs.html

QIF Report

The topics in this section include:

About QIF Report

Creating a QIF Report

Creating a QIF Report from a STEP File

About QIF Report

Quality Information Framework (QIF) is an ANSI standard. PC-DMIS now supports measurement data output in a QIF format report if the routine was created by importing a QIF MBD (Model-Based Definition) file.

For more information on QIF, refer to the QIF website.

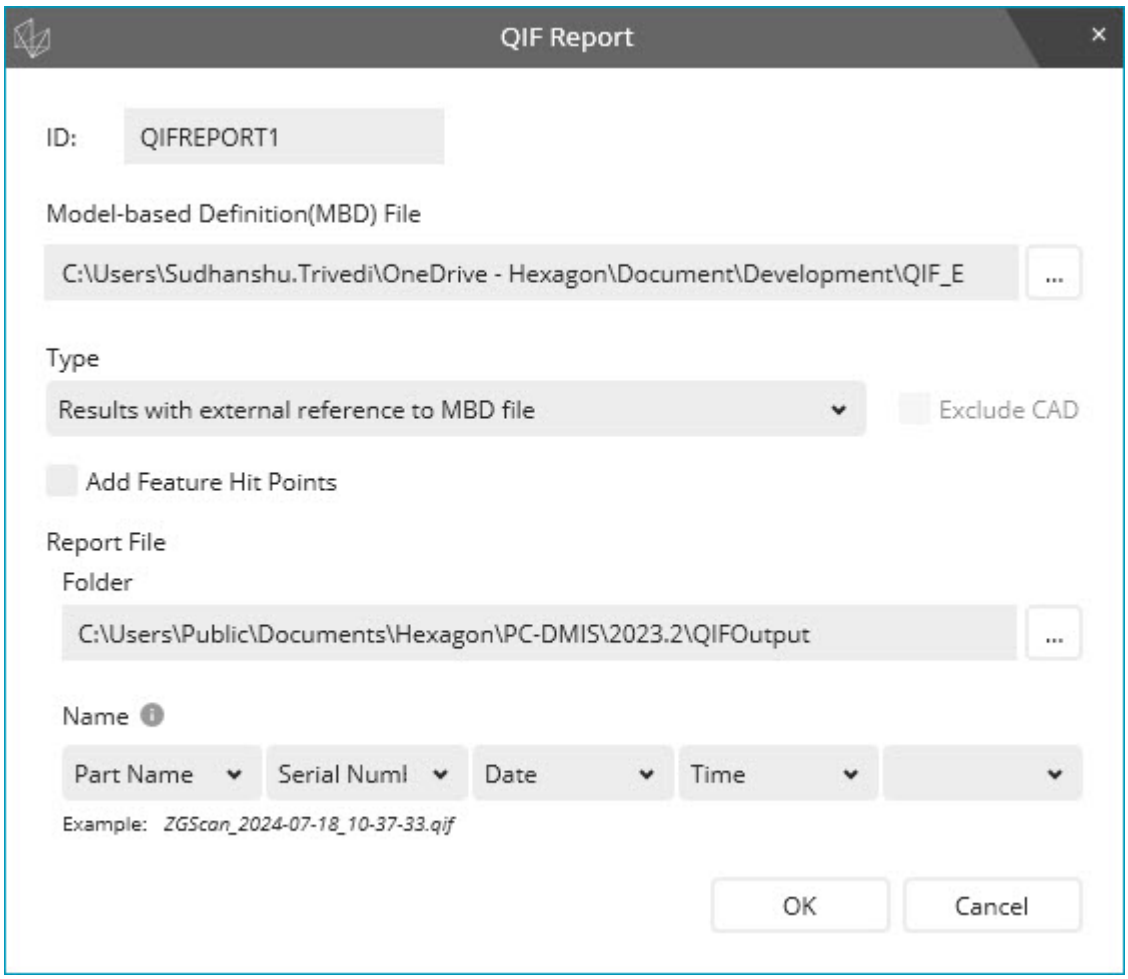
To output the results in QIF format, you must create a measurement routine by importing a CAD model using a QIF MBD file.

Creating a QIF Report

To create a QIF report, follow these steps:

1. Create a new measurement routine.
2. Import the CAD model using QIF MBD file. The CAD has the required information, such as feature and characteristic IDs. The QIF MBD file supports PMI data.
3. Define the measurement features using GD&T Selection Mode (from CAD). For information on how to do this, see the “Using GD&T Selection Mode (from CAD)” topic.

   When you create features and characteristics this way, PC-DMIS stores Feature ID, Characteristic ID, and other information required to output QIF results.
4. After you create the measurement routine, insert the QIF Report command. To do this, select the **Insert | Report Commands | QIF Report** menu option to open the **QIF Report** dialog box.

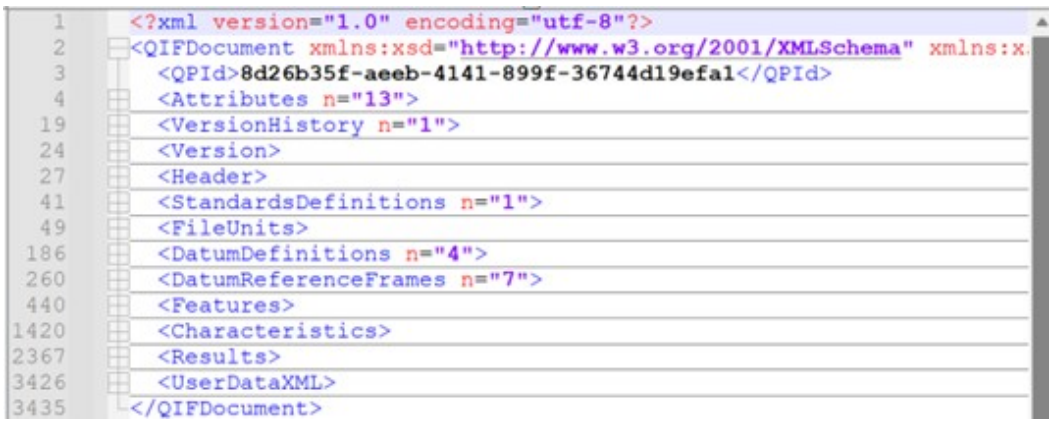


Complete these options:

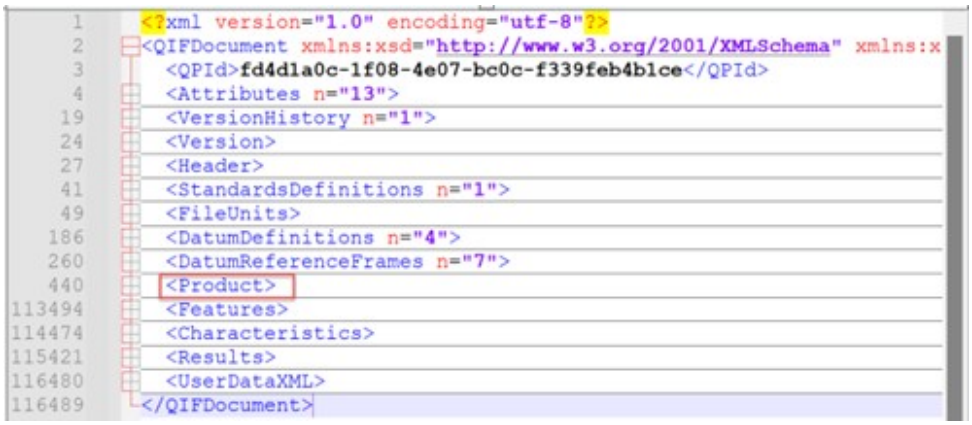
- **ID** - ID of the QIF Report command.
- **Model-based Definition (MBD) File** – This is the location of the .qif file that PC-DMIS uses to obtain information for the output file.

  Ensure that the .qif file is available at this location during execution. QIF Report needs this file.

- **Type** - You can output the QIF Report file in these formats:
  - **Results with MBD data** - In this format, the result file includes the MBD data. This file has the advantage of containing all information in one file. However, adding information to the result file that is available in the MBD file increases the size of the result file.



- **Exclude CAD** - Use this check box to include or exclude the CAD data when you select the **Results with MBD data** option. PC-DMIS stores the CAD data in the `Product` node of the QIF MBD file. This option is enabled only if you select the **Results with MBD data** option.



  - **Results with external references to MBD file** - In this format, the result file does not contain the MBD data. It has a node to define the information on external QIF references. This file type is of minimum size as it includes only the results data.

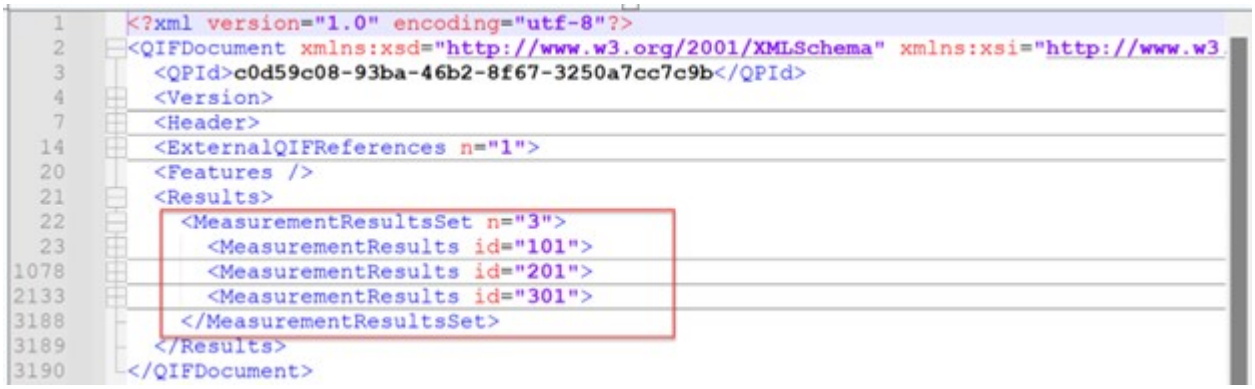
- **Add Feature Hit Points** - If you select this check box, PC-DMIS adds the Hit point data to the result file. This option is available for all output file types.

- **Report File**
  - **Folder** - Use this box to type the path to store the QIF result file, or click the Browser (...) button and navigate to the location where you want to store it.
  - **Name** - Use the drop-down list box to define a name rule for the result file. The options in the drop-down list are **Part Name**, **Serial Number**, **Revision Number**, **Date**, and **Time**.

5. To create a QIF Report, click **OK**.

Results of Multiple Executions in One QIF Report File

You can store multiple results in a single QIF Report file. To get a report of multiple executions in one file, create a name rule to generate report files with the same name. If PC-DMIS finds an existing report file with the same name, it only adds the result, and the previous data remains the same.



Creating a QIF Report from a STEP File

To create a QIF report, follow these steps:

Prerequisites: A STEP file that contains UUIDs for the features and characteristics.

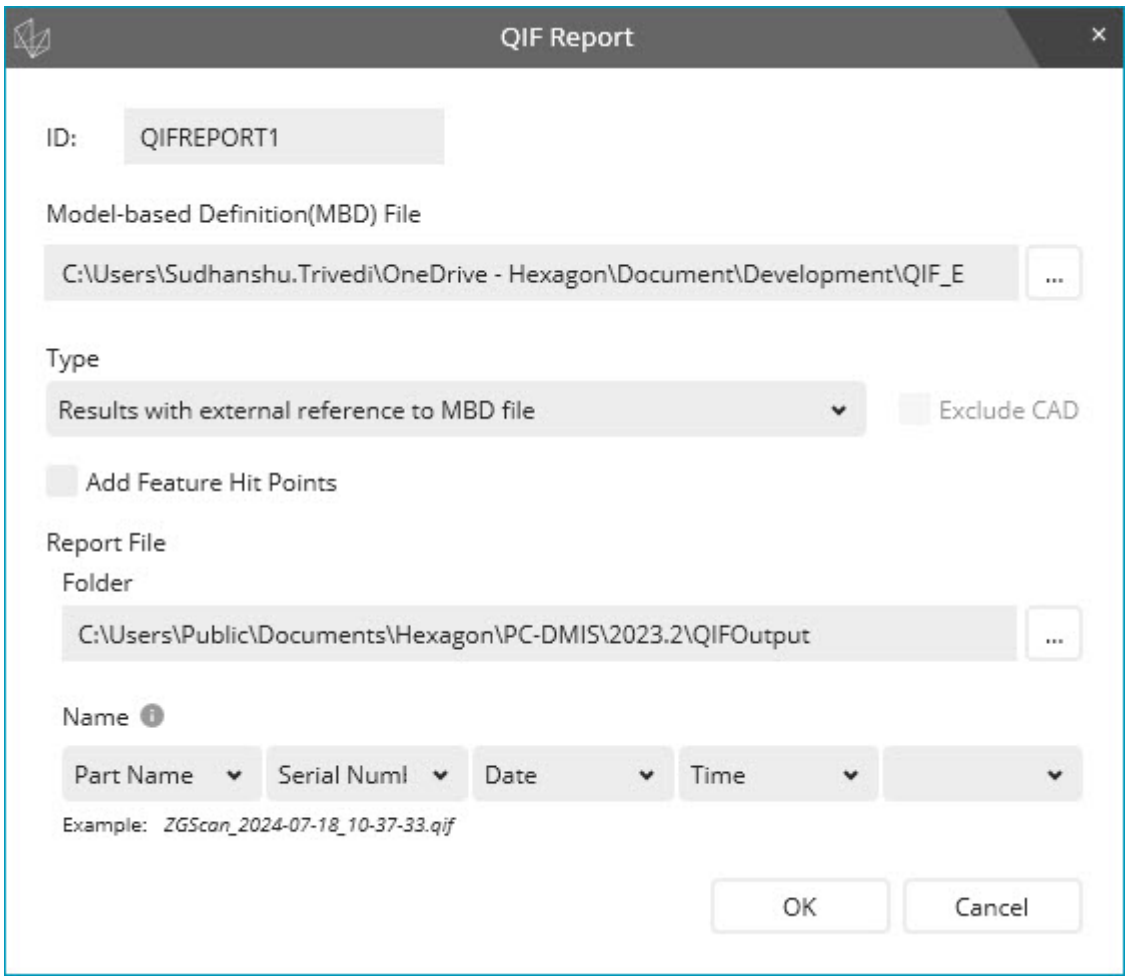
1. Create a new measurement routine.

2. Import the STEP file that contains UUIDs for the features and characteristics. For information on how to import a STEP file, see the "Importing a STEP File" topic in the PC-DMIS Core documentation.

   When you import a STEP file that contains UUIDs into PC-DMIS, PC-DMIS automatically generates and imports a QIF Plan containing the QIF Bill of Characteristics in the background. This associates the digital thread IDs with the corresponding STEP features and characteristics.

3. Define the measurement features using GD&T Selection Mode (from CAD). For information on how to do this, see the "Using GD&T Selection Mode (from CAD)" topic.

4. After you create the measurement routine, insert the QIF Report command. To do this, select the **Insert | Report Commands | QIF Report** menu option to open the **QIF Report** dialog box.

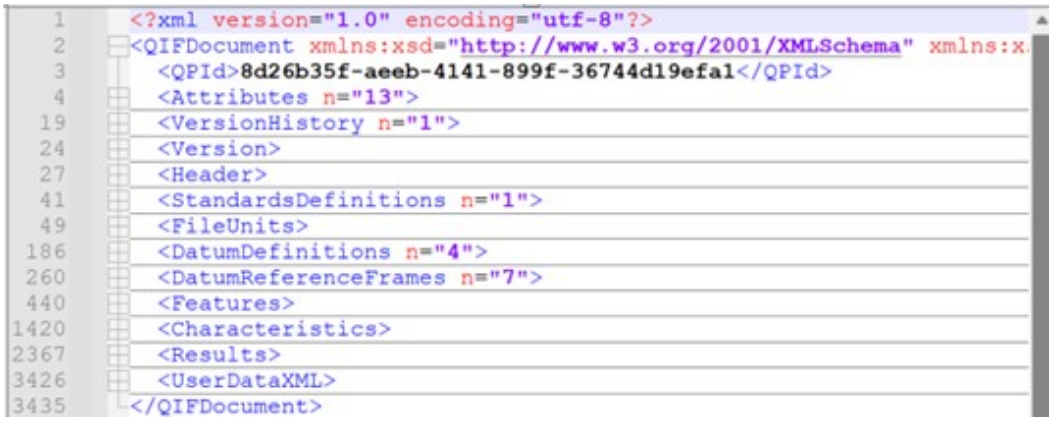


   Complete these options:

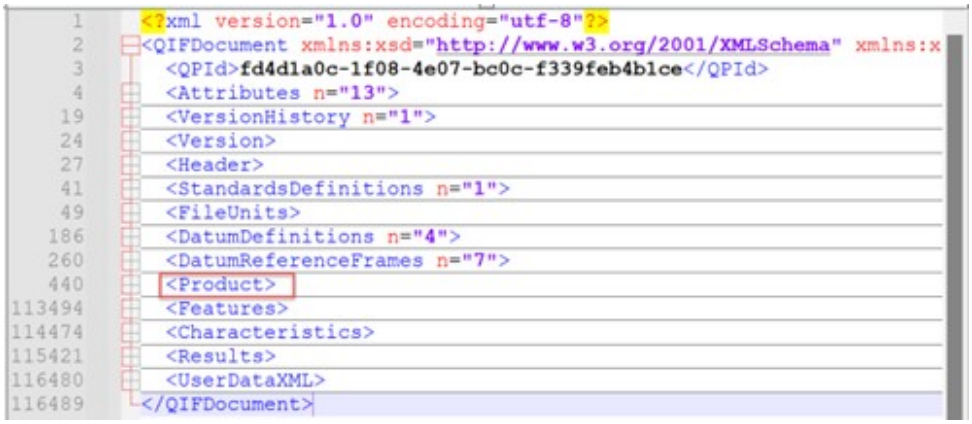
   - **ID** - ID of the QIF Report command.
   - **Model-based Definition (MBD) File** – This is the location of the .qif file that PC-DMIS uses to obtain information for the output file.

Ensure that the .qif file is available at this location during execution. QIF Report needs this file.

- **Type** - You can output the QIF Report file in these formats:
  - **Results with MBD data** - In this format, the result file includes the MBD data. This file has the advantage of containing all information in one file. However, adding information to the result file that is available in the MBD file increases the size of the result file.



    - **Exclude CAD** - Use this check box to include or exclude the CAD data when you select the **Results with MBD data** option. PC-DMIS stores the CAD data in the `Product` node of the QIF MBD file. This option is enabled only if you select the **Results with MBD data** option.



  - **Results with external references to MBD file** - In this format, the result file does not contain the MBD data. It has a node to define the information on external QIF references. This file type is of minimum size as it includes only the results data.

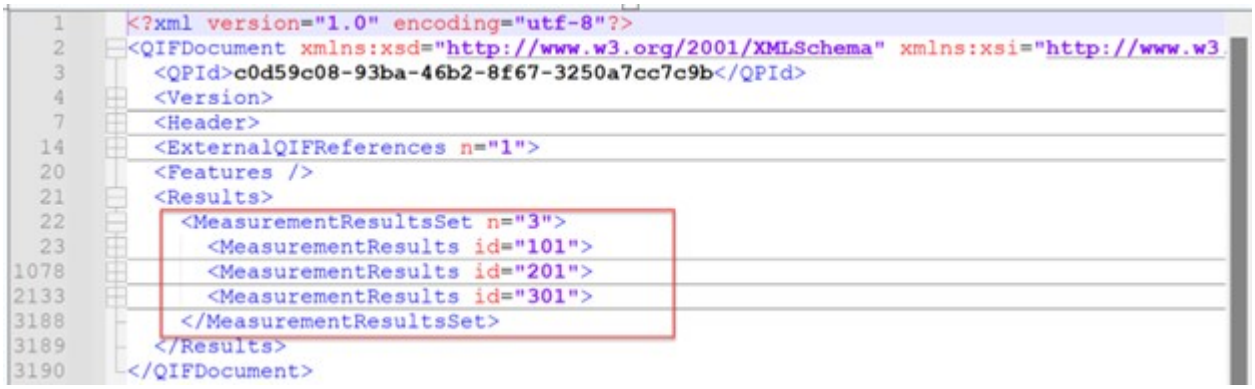
- **Add Feature Hit Points** - If you select this check box, PC-DMIS adds the Hit point data to the result file. This option is available for all output file types.

- **Report File**
  - **Folder** - Use this box to type the path to store the QIF result file, or click the Browser (...) button and navigate to the location where you want to store it.
  - **Name** - Use the drop-down list box to define a name rule for the result file. The options in the drop-down list are **Part Name**, **Serial Number**, **Revision Number**, **Date**, and **Time**.

5. To create a QIF Report, click **OK**.

Results of Multiple Executions in One QIF Report File

You can store multiple results in a single QIF Report file. To get a report of multiple executions in one file, create a name rule to generate report files with the same name. If PC-DMIS finds an existing report file with the same name, it only adds the result, and the previous data remains the same.


<?xml version="1.0" encoding="utf-8"?>
<QIFDocument xmlns:xsd="http://www.w3.org/2001/XMLSchema" xmlns:xsi="http://www.w3
<QPId>c0d59c08-93ba-46b2-8f67-3250a7cc7c9b</QPId>
<Version>
<Header>
<ExternalQIFReferences n="1">
<Features />
<Results>
<MeasurementResultsSet n="3">
<MeasurementResults id="101">
<MeasurementResults id="201">
<MeasurementResults id="301">
</MeasurementResultsSet>
</Results>
</QIFDocument>

Inserting Report Commands

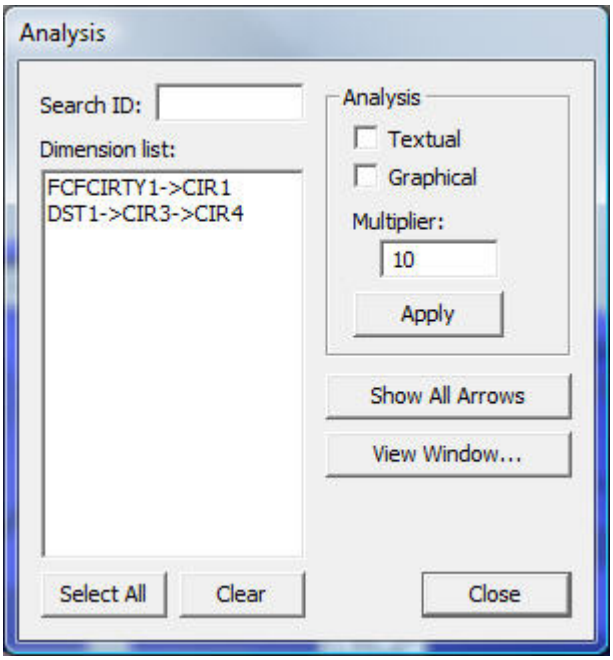
Inserting Report Commands: Introduction

PC-DMIS allows you to insert a variety of commands into the Edit window that affect how and what information gets displayed in reports. These commands, inserted by selecting menu options from the **Insert | Report Command** sub menu, allow the programmer to configure, view, print, and otherwise customize the appearance of the Edit window.

The following topics are covered in this chapter.

- Inserting Commands Related to the Analysis Window
- Inserting Dimension Info Boxes
- Inserting Point Info Boxes
- Inserting Programmer Comments
- Embedding Reports or Report Templates in a Measurement Routine
- Inserting External Objects
- Inserting a Print Command
- Inserting a Form Feed Command
- Working with Viewsets
- Inserting a FORM Command
- Inserting Screen Captures
- Inserting Snapshots

Inserting Commands Related to the Analysis Window

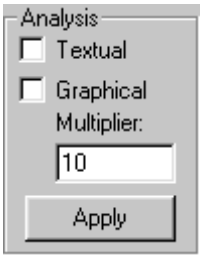


Analysis Dialog Box

The **Insert | Report Command | Analysis** menu option brings up the **Analysis** dialog box. This dialog box sends the dimensional printout to an **Analysis** window in a format suitable for close examination. From the Analysis window, you can insert an `ANALYSISVIEW` command into the Edit window to include the analysis on the report. For information on the Analysis window, see the "View Window" topic.

If this dialog box is open, you can click on a feature in the Graphic Display window to automatically highlight dimensions using that feature in the **Dimension List** box.

Analysis area



The **Analysis** area of this dialog box allows you to choose to view the dimension(s) selected from the **Dimension List** box in either a graphical or a textual readout.

By default, the check boxes in the **Analysis** area are not selected. When you select either the **Textual** or **Graphical** check box and click **Apply**, PC-DMIS will show the selected dimensions in a textual format or a graphical image. It will not display any CAD data.

It is helpful to have either the Graphical or Textual Analysis option on when having a problem with a feature whose cause needs to be determined.

Suppose you are inspecting a plane for flatness, and you find it is out of tolerance. Using the Graphical Analysis option allows you to examine the plane and determine if a specific area is bad or if the entire surface is irregular.

Graphical - If you select the Graphical check box, PC-DMIS stores the dimension information in a format that you can view graphically.

Textual - If you select the Textual check box, PC-DMIS stores the dimension information in a format that you can view textually.

Multiplier - The value in the Multiplier box is a scaling factor that magnifies the arrow and tolerance zone for the graphical analysis mode. For example, if you enter a value of 2.0, PC-DMIS scales the arrow two times the graphical image. You can use this option for viewing purposes only. It is not reflected in the text printout.

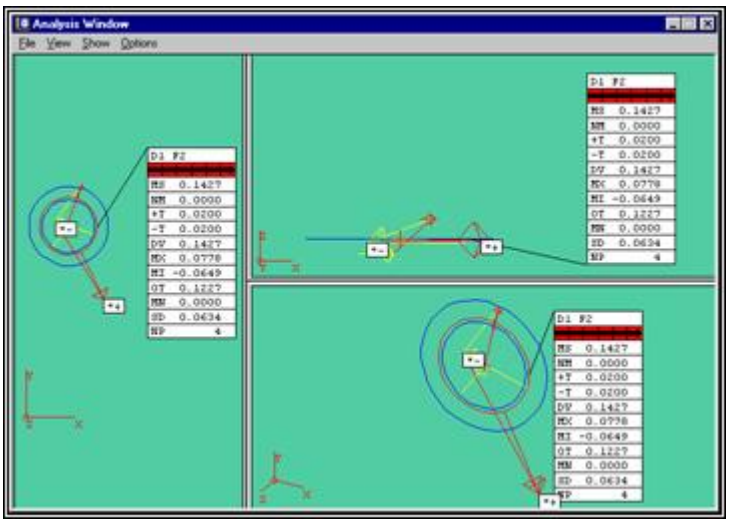
Show All Arrows

The **Show All Arrows** button shows the graphical analysis arrows used to mark deviation. These graphical analysis arrows can be hidden arrow by arrow. To do this, press CTRL + SHIFT, and click the arrows to hide. Clicking the **Show All Arrows** button re-displays the arrows. The graphical analysis arrows will then be reset for display purposes.

View Window (Analysis Window)

From the **Analysis** dialog box, the **View Window** button displays the Analysis window.

You can use the Analysis window's menu bar to alter the graphical image, capture the current graphics screen, or edit the selected dimension colors.



Analysis Window

To access this window:

1. Select **Insert | Report Command | Analysis** menu option to show the **Analysis** dialog box.
2. From the **Analysis** dialog box:
    - Select the dimension that needs to be examined in the Analysis window.
    - In the **Multiplier** box, type a value to magnify the arrow and tolerance zone.
3. Click **View window** button to open the Analysis window.

The menu bar on the Analysis Window provides these menus:

- **File**
- **View**
- **Show**
- **Options**

In order to determine the initial view of the graphical analysis in the Analysis Window, PC-DMIS uses either the scan cut vector for profile dimensions of scans or the current workplane for other dimensions. This view can be altered later.

File Menu

The *Analysis window's* **File** menu allows you to send the contents of the Graphics Analysis window directly to the printer for printing purposes.

View Menu

The *Analysis window's* **View** menu allows you to temporarily alter the graphics displayed for the selected dimension. By rotating, scaling, or changing the view, you can easily access the necessary dimensions.

This option is especially useful when viewing the profile error of a set of points.

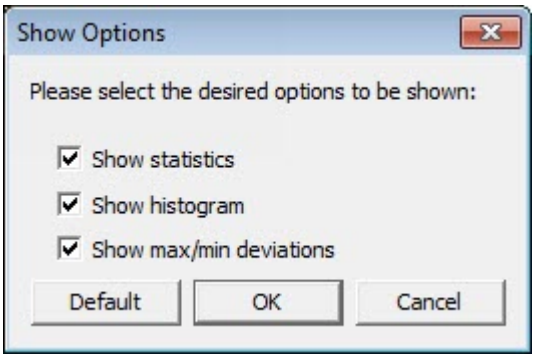
The **View | Change** menu option lets you create new views from a CAD file. This function is useful when a 2D IGES file is used that contains several views of a part in one plane and the views need to be associated with their correct orientation in the Graphic Display window. See "Setting Up the Screen View" in the "Editing the CAD Display chapter for more information on manipulating views.

The **View | Scale To Fit** menu option re-draws the part image to fit entirely within the Graphic Display window. This function is useful whenever the image becomes too large or small.

The **View | Rotate** menu option displays the **Rotate** dialog box, allowing you to rotate the part image in three dimensions.

The topic "Rotating the Drawing" in the "Editing the CAD Display" chapter describes rotating in further detail.

Show Menu



Show Options Dialog Box

The *Analysis window's* **Show | Show Options** menu command displays the **Show Options** dialog box. This dialog box allows you to choose what to display in the Analysis window.

The *Analysis window's* **Show | Show Dimensions Stats** brings up the **Edit Dimension Info** dialog box. This dialog box allows you to edit in depth the highlighted dimension. See

"Common Dimension Dialog Box Options" in the "Using Legacy Dimensions" chapter and "Inserting Dimension Info Boxes" in this chapter for more information.

Show Statistics - Displays various statistics in the Analysis window. If the **Show Statistics** check box is not selected, dimensional information text will not be displayed (for any dimensions). See the **Edit Dimension Info** menu option to change the format of the statistics displayed in the Analysis window.

Show Histogram - Displays a histogram of the deviations for each point. Histograms can be moved to new locations within the Analysis window.

To move to a new location within the Analysis window:

1. Click on the desired histogram.
2. Drag the cursor to the desired location.
3. Release the mouse button.

Show Max/Min Deviations - Displays the maximum and minimum deviations indicated with a '*+' or '*-' symbol.

Options Menu

The *Analysis window's* **Options** menu allows you to capture the current graphics screen for later use. It also provides access to the **Edit Color** dialog box and sizing of the Analysis window.

Create/Save Analysis View Command

The *Analysis window's* **Options | Create Analysis View Command** menu option uses the dimensions, views, and rotations used in the Analysis window and inserts that information into an `ANALYSISVIEW` command into the Edit window. When PC-DMIS executes the `ANALYSISVIEW` command, the following occurs:

- PC-DMIS opens the Analysis window momentarily and then takes a screen capture of its contents.
- PC-DMIS then displays this screen capture in the Report window. It automatically sizes the screen capture to fit the Report window. It is inserted into the Report window's underlying TextReportObject. For information on the TextReport object, see the "TextReportObject" topic in the "Reporting Measurement Results: Overview" chapter. It does not affect the AnalysisWindow object.

The Edit window command line for this option reads:

```
ANALYSISVIEW/D1,D2,D3,,
```

D1,D2,D3 = list of dimensions to be displayed.

There is no limit to the number of dimension parameters that can be used.

You can also type " `ANALYSISVIEW` " in the Edit window to creates an `ANALYSISVIEW` command. Any dimensions that you select for analysis become parameters for this command.

Save Graphics to Report

The image will remain on the clipboard until another screen is captured, or the measurement routine is closed.

The *Analysis window's* **Options | Save Graphics To Report** menu option captures the view in the Analysis window and stores it in a `DISPLAY/METAFILE` command. When executed this in turn sends the capture into the inspection report.

PC-DMIS does not have the ability to edit or format the captured image. The graphics will be displayed at the cursor's location.

This `DISPLAY/METAFILE` command differs from the `DISPLAY/METAFILE` command generated from the main menu in these ways:

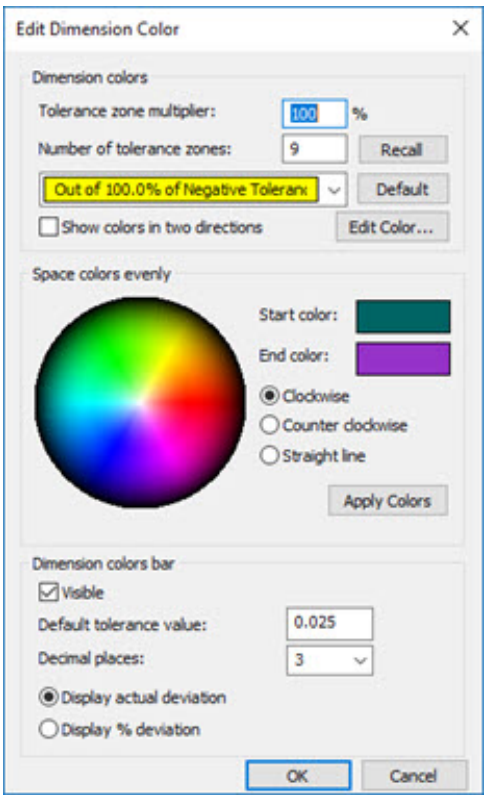
- It takes the screen capture of the view in the Analysis window when you select the menu item, not the Graphic Display window.
- The screen capture does not get retaken during execution, it is a static image.

See "Create Analysis View Command" for creating a screen capture that automatically updates itself upon execution.

See "Screen Captures" in the "Using the Edit Window" chapter for additional information on the `DISPLAY/METAFILE` command.

Edit Dimension Colors

The *Analysis window's* **Options | Edit Dimension Colors** menu option allows you to set the color for a specified tolerance range. Select this option to display the **Edit Dimension Colors** dialog box.

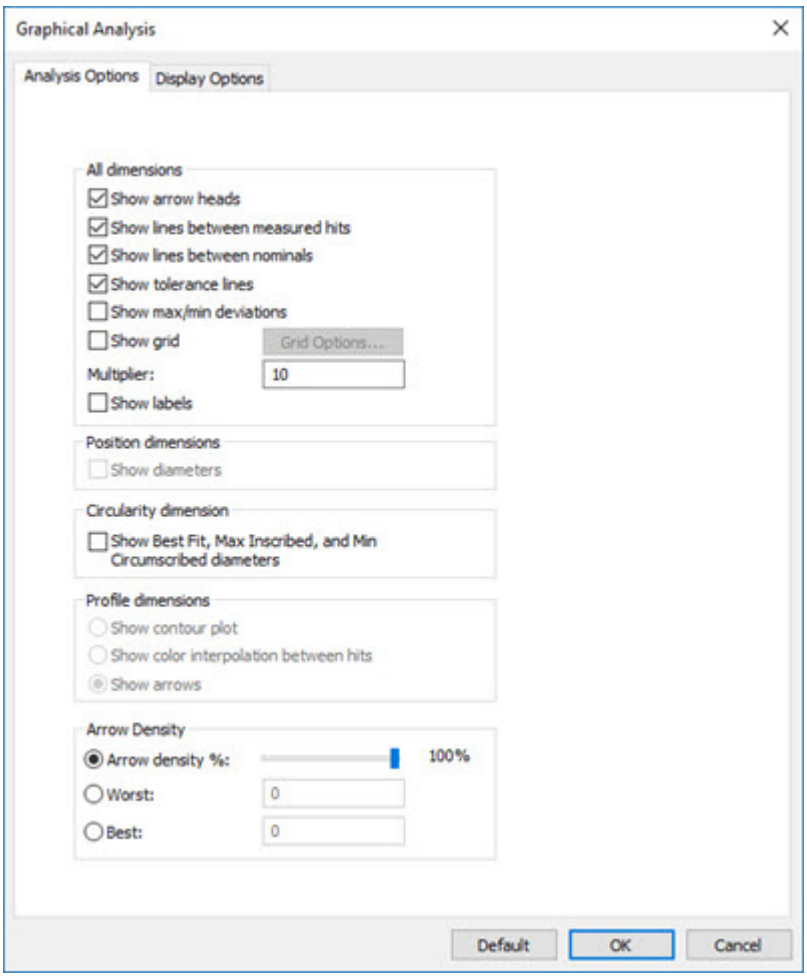


Edit Dimension Colors Dialog Box

For instructions on how to use this dialog box to change dimension colors for a tolerance, see the "Editing Dimension Colors" topic in the "Editing the CAD Display Options" chapter.

Analysis Options

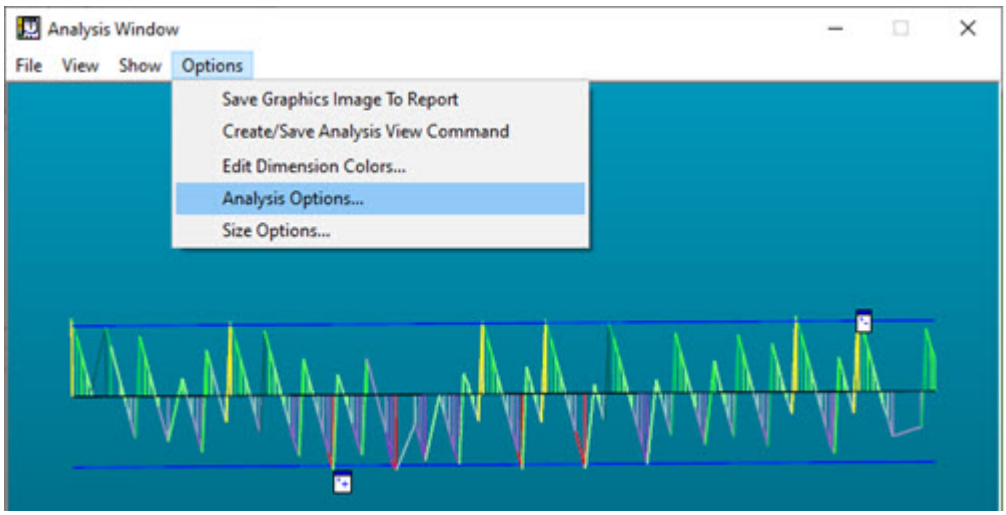
This **Graphical Analysis** dialog box determines the information that gets displayed in the Analysis window.



Graphical Analysis Dialog Box

To access this dialog box:

1. Select **Insert | Report Command | Analysis** from the menu to show the **Analysis** dialog box.
2. From the **Analysis** dialog box, from the **Dimension list** box, select the dimension.
3. Click **View Window** to show the Analysis window.
4. To open the **Graphical Analysis** dialog box, from the Analysis window's menu, select **Options | Analysis Options**.



Analysis Options Menu Option

You can quickly access the analysis settings from the main PC-DMIS menu (**Edit | Graphic Display | Analysis Options**) when:

- You change the dimension option from `GRAPH=OFF` to `GRAPH=ON` (or, for Geometric Tolerances, from `CADGRAPH=OFF` to `CADGRAPH=ON`), and...
- You want to change the analysis settings shown in the Graphic Display window

Prior to this, PC-DMIS required that you create an `ANALYSIS` command to get to these options.

The **Graphical Analysis** dialog box contains two tabs:

- **Analysis Options** - For a description of the items on this tab, see "Analysis Options tab" below.
- **Display Options** - For a description of the items on this tab, see the "AnalysisWindow Object" topic in the "Reporting Measurement Results" chapter where it covers the **Reporting Options** tab.

Note that when you open the **Graphical Analysis** dialog box from an analysis view, it shows the tab names as **Analysis Options** and **Display Options**. When you open the dialog box from within a report, it shows the tab names as **Analysis Options** and **Reporting Options**.

Analysis Options tab

The **Analysis Options** tab of the **Graphical Analysis** dialog box has these areas:

All dimensions area

Show arrow heads - This shows the arrow heads on the deviation lines.

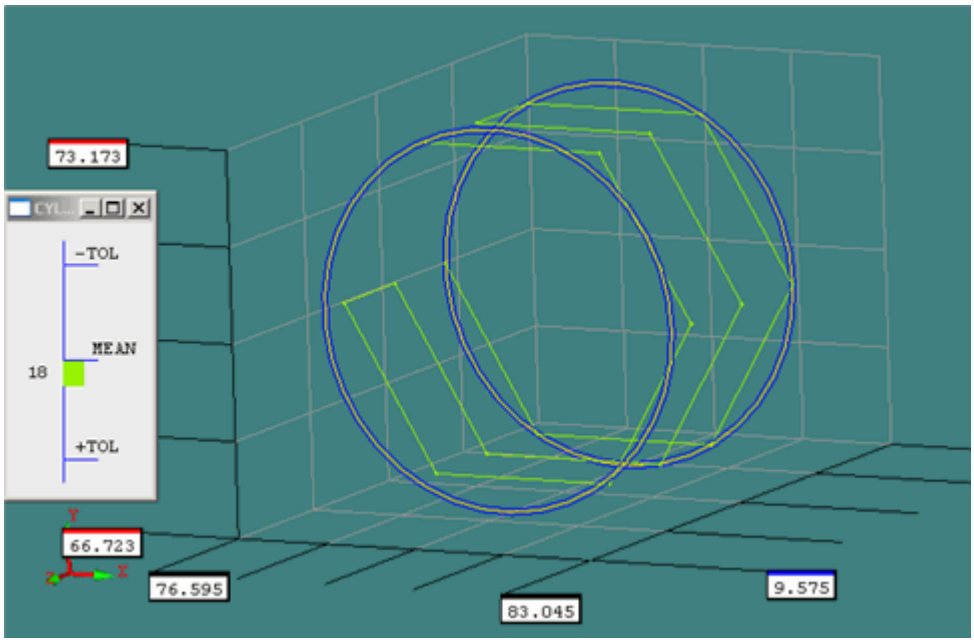
Show lines between measured hits - This shows the lines between the measured hits.

Show lines between nominals - This draws the lines that PC-DMIS draws between the nominal values.

Show tolerance lines - This displays the acceptable tolerances for the dimension.

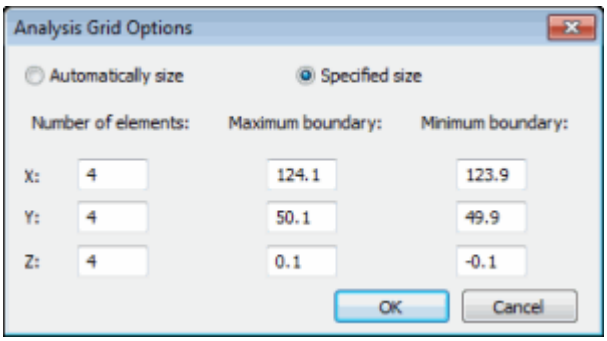
Show max/min deviations - This allows the maximum and minimum deviations to be indicated with a *+ or *- symbol.

Show grid - This displays a 3D grid-like backdrop for the graphical analysis. Enabling the grid may help you better visualize rotated items. It also enables the **Grid Options** button.



A sample Graphical Analysis window with the Grid option enabled.

Grid Options - This displays the **Analysis Grid Options** dialog box. You can use this dialog box to define your grid.



Analysis Grid Options Dialog Box

The dialog box contains these options:

Automatically size - This automatically sizes the grid based on the number of objects specified in the X, Y, and Z fields.

Specified size - This enables the **Maximum Boundary** and **Minimum Boundary** boxes, which let you type specific sizes for the maximum and minimum boundaries.

Number of elements - This defines the number of divisions that PC-DMIS draws on the analysis grid.

Maximum boundary - This defines upper limits of the analysis grid coordinates.

Minimum boundary - This defines the lower limits of the analysis grid coordinates.

Multiplier - This lets you type a scaling factor that magnifies the deviation arrows and the tolerance zone for the graphical analysis mode. If you enter a value of 2.0, PC-DMIS scales the arrows two times the calculated deviation for each feature hit.

This option is used for viewing purposes only and is not reflected in the text-only printout.

Position dimensions area

Show Diameters - This shows the diameters of the available Position dimensions.

Circularity dimension area

Show Best Fit Diameter, **Max Inscribed Diameter**, **Min Circumscribed Diameter** - These check boxes show the diameters for the Circularity dimension. These diameters represent the average, maximum, and minimum deviations for a round feature.

Profile dimensions area

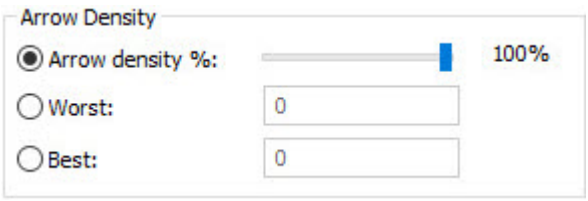
Show Contour Plot - This option only works for Profile dimensions applied to patch scans. The contour plot uses the patch scan's points to create a mesh. It then shades the mesh with the colors related to the profile deviations from each hit.

PC-DMIS can also display the contour plot on the CAD model in the Graphic Display window itself. For information, see "Displaying Surface Profile Contour Plots" in the "Using Legacy Dimensions" chapter.

Show Color Interpolation Between Hits - This inserts colors between hits. This option is available for Profile dimensions.

Show Arrows - This shows arrows for each point's deviation. This option is available for Profile dimensions.

Arrow Density area

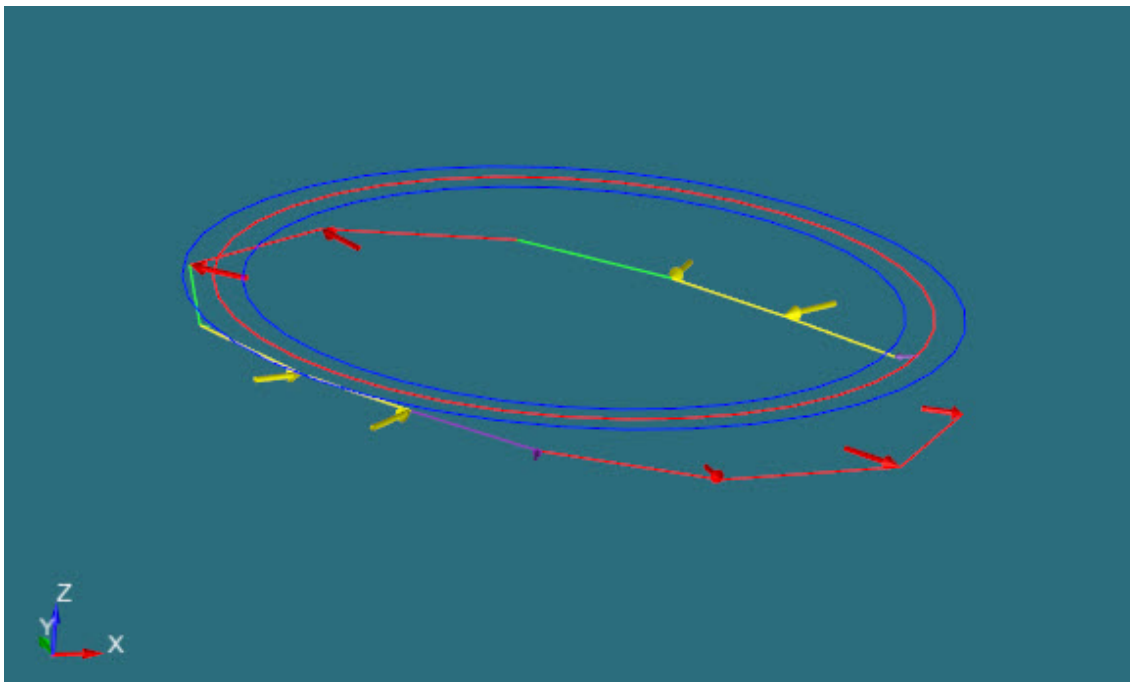


Arrow Density - This slider changes the density of arrows for the points displayed in any dimension in the entire measurement routine that uses Graphical Analysis.

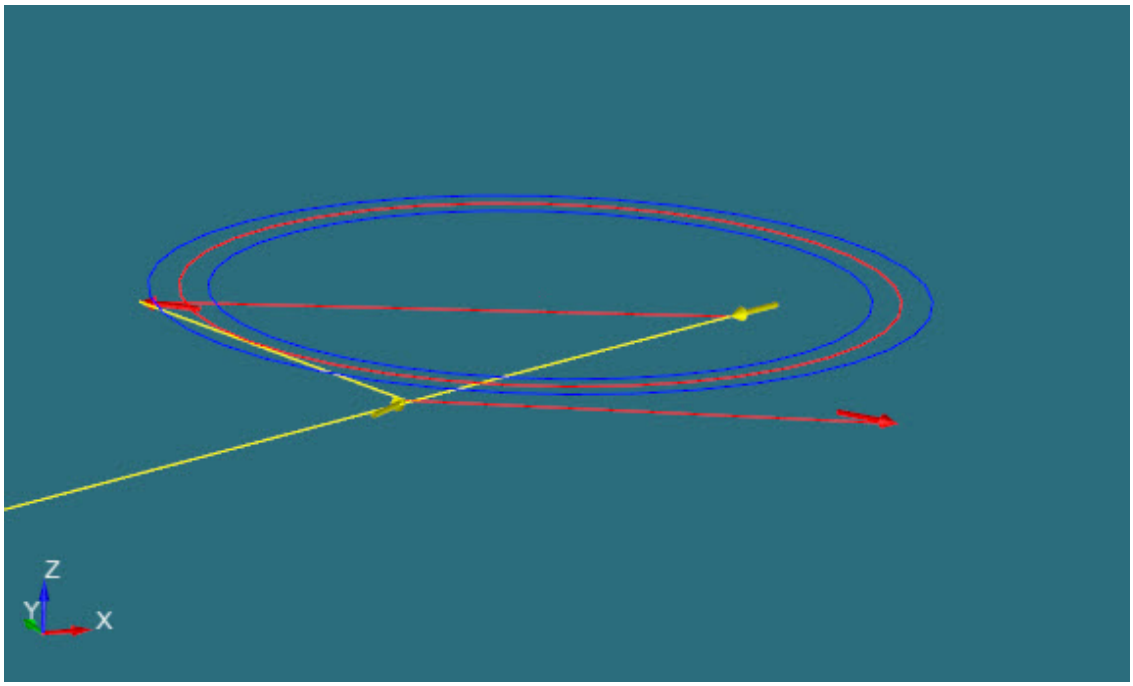
This differs from the **Arrow Density** functionality in the **Reporting** tab of the **Geometric Tolerance** dialog box. The setting in the **Reporting** tab only affects that specific dimension. See "Reporting Tab" in the "Using Geometric Tolerances" chapter.

Worst - This box defines the number of worst deviations for which to display arrows. The examples below show all arrows on the left and the three worst arrows on the right.

Best - This box defines the number of best deviations for which to display arrows.

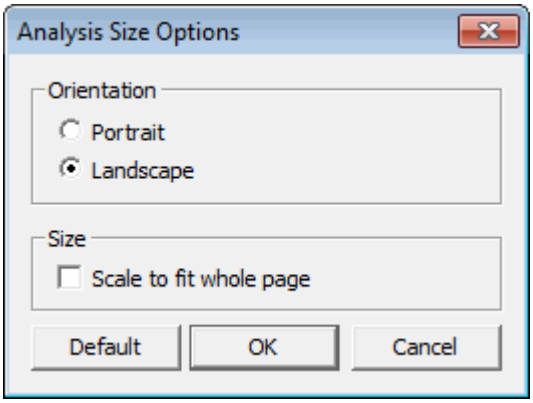


Example showing all the deviations (100% arrow density).



Example showing the three worst deviations.

Size Options



Analysis Size Options Dialog Box

The **Analysis Size Options** dialog box allows you to determine how PC-DMIS displays the Analysis window. These options also allow you to set your `ANALYSISVIEW` command so that PC-DMIS takes the screen capture in one of these selected modes during execution. (See "Create Analysis View Command".)

You can select the following options:

Portrait
This option sizes the Analysis window so that it matches the standard Portrait page size.

Landscape
This option sizes the Analysis window so that it matches the standard Landscape page size.

Scale to fit whole page
This check box scales the window so it would fit an entire printed page.

Inserting a 2D Plot Command

The topics in this section include:

About 2D Plot
Scan Plot

About 2D Plot

The 2D Plot function enables you to creates 2D curve plots. Use this function to generate plots for all types of 2D scans.

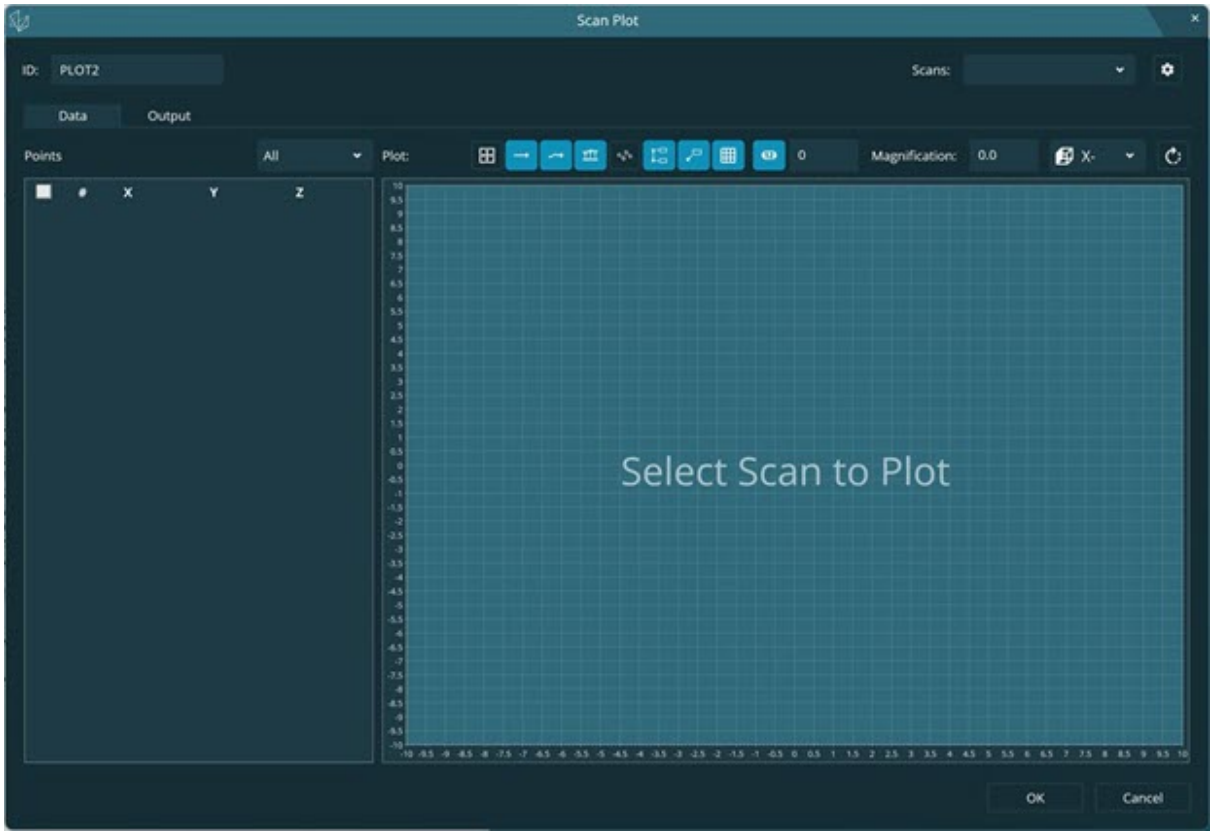
Scan Plot

PC-DMIS allows you to create a 2D plot of your scan.

Creating a Scan Plot

To create a Scan Plot, follow these steps:

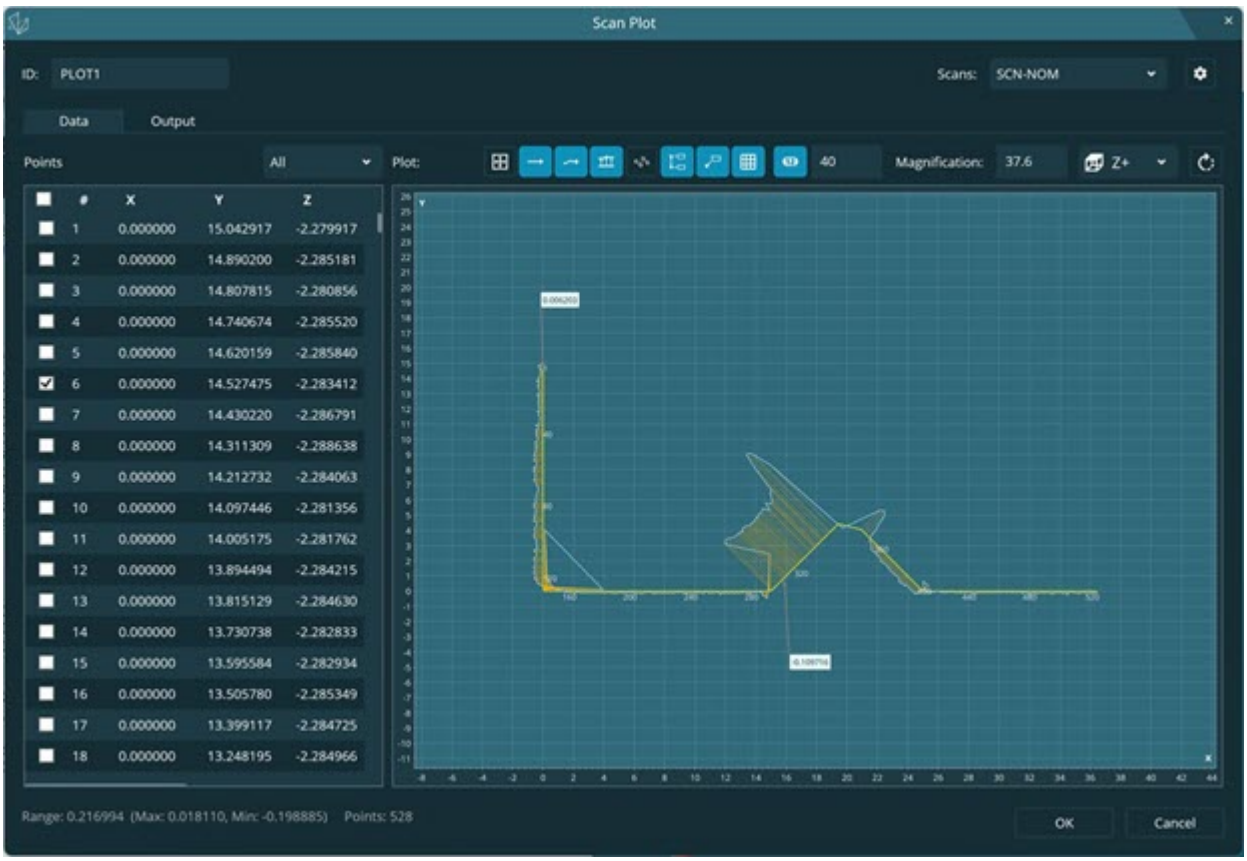
1. Select the **Insert | Report Command | 2D Plot | Scan Plot** menu option to open the **Scan Plot** dialog box.



2. Complete the options:
    - **ID** – Use this box to type in the name of the plot for your scan.
    - **Scans** – From the list, select the scan ID for which you want to create the plot.
3. Click the **OK** button to create the plot.

Understanding the Scan Plot Dialog Box

Select the **Insert | Report Command | 2D Plot | Scan Plot** menu option to open the **Scan Plot** dialog box.

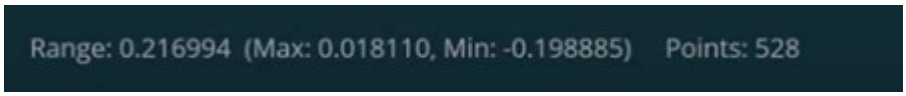


- **ID** - Use this box to enter the name of the plot for your scan.
- **Scans** - From the list, select the scan ID for which you want to generate the plot.
- **Settings** - Select this icon to open the **Settings** dialog box, where you can edit the default settings.

  - **Set transparent background for callout plotting** – Select this check box if you want the callout background to be transparent. This helps save ink when you do not want to print colored callouts.
  - **Hide coordinate axes and grid when plotting** – Select this check box to hide the coordinate axes and grid.

- **Set limit of callouts** – Use this option to define the maximum number of callouts to include in the plot. The default value is **50**. This ensures that the plot area remains uncluttered and provides meaningful information.

- **Status Bar**



The status bar, located at the bottom of the plot area, displays information about the Range, Max and Min deviations, and the total number of points in the scan.

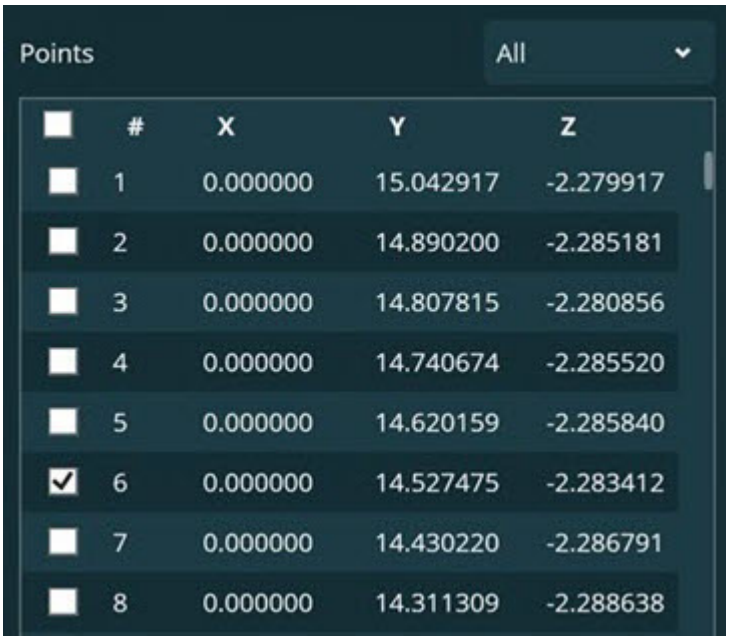
Data Tab

The Data tab has these two areas:

- Points Area
- Plot Area

Points Area

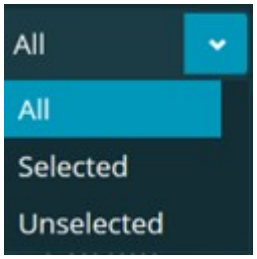
The **Points** area contains a list of all the points in a scan, ordered sequentially. You can drag the parting line to view each point's X, Y, Z, I, J, K, T, and tolerance values. The T value represents the deviation of the scan points from their normal.



- **Check Boxes** – When you select the check box for a point, PC-DMIS shows the deviation for that point in the Plot area. PC-DMIS also adds the point to the textual analysis in the report.

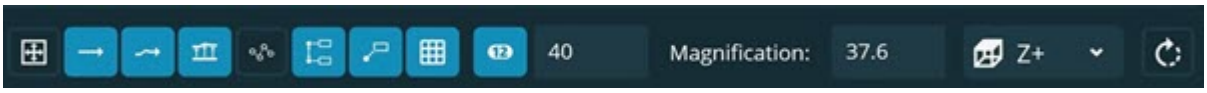
  To select all points, select the check box in the header area. To deselect all points, deselect the check box in the header area.

- **Display** – This list allows you to control which points PC-DMIS displays in the Points area.



  - **All** – Select this option to show all the points in the Points area.
  - **Selected** - Select this option to show only the selected points in the Points area.
  - **Unselected** - Select this option to show only the unselected points in the Points area.

Plot Area



Click the icon to show or hide the plot object.

Scale to Fit – Click this button to adjust the plot to fit entirely within the plot area.

Show Nominal Curve – Click this icon to show or hide the nominal curve.

Show Measured Curve – Click this icon to show or hide the measured curve.

Show Deviations – Click this icon to show or hide the deviation from nominal values.

Show Points on Curve – Click this icon to show or hide the points on the curve.

Show Max/Min Deviations Callouts – Click this icon to show or hide the callouts for the point with the Max deviation and Min deviation. The Max deviation is a positive deviation that indicates more material than normal, while the Min deviation is a negative deviation that indicates less material than normal. Both Max deviation and Min deviation are dynamic and are selected and displayed with each execution.

Show Callouts – Click this icon to show or hide the deviation at nominal points.

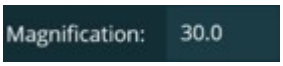
Use Ctrl + left-click on a nominal point in the plot area to display its callouts. PC-DMIS selects this point in the Points area. You can also select points from the point list area to display their callouts.

By default, the number of callouts is limited to 50. You can modify the number of callouts in the **Settings** dialog box. For information on how to modify the number of callouts, see the "Understanding the Scan Plot Dialog Box" section of the PC-DMIS Core documentation.

You can move the callouts to your desired location. To do this, left-click and hold on the callout, then drag it to your desired position. The plot remembers the callout position and prints it with each execution.

Show Grid – Click this icon to show or hide the grid.

Show Point Numbers – When you create a plot, PC-DMIS smartly assigns the point markings. It positions the point number evenly and ensures that the point number markings are spread apart to indicate the scan direction, making it easier to find the zone of interest. Click this icon to enable this feature.



Smart Magnification – When you create a plot, PC-DMIS smartly assigns the appropriate deviation magnification. It calculates the magnification to display the deviation curve clearly, without distortion, providing a clear picture of the deviations. This helps you make decisions quickly. You can also enter a magnification value of your choice.

View Axis list – From the list, select the required view axis. The plot plane can be XY, YZ, or ZX. By default, the corresponding view axes are X+, X-, Y+, Y-, Z+, and Z-, respectively.

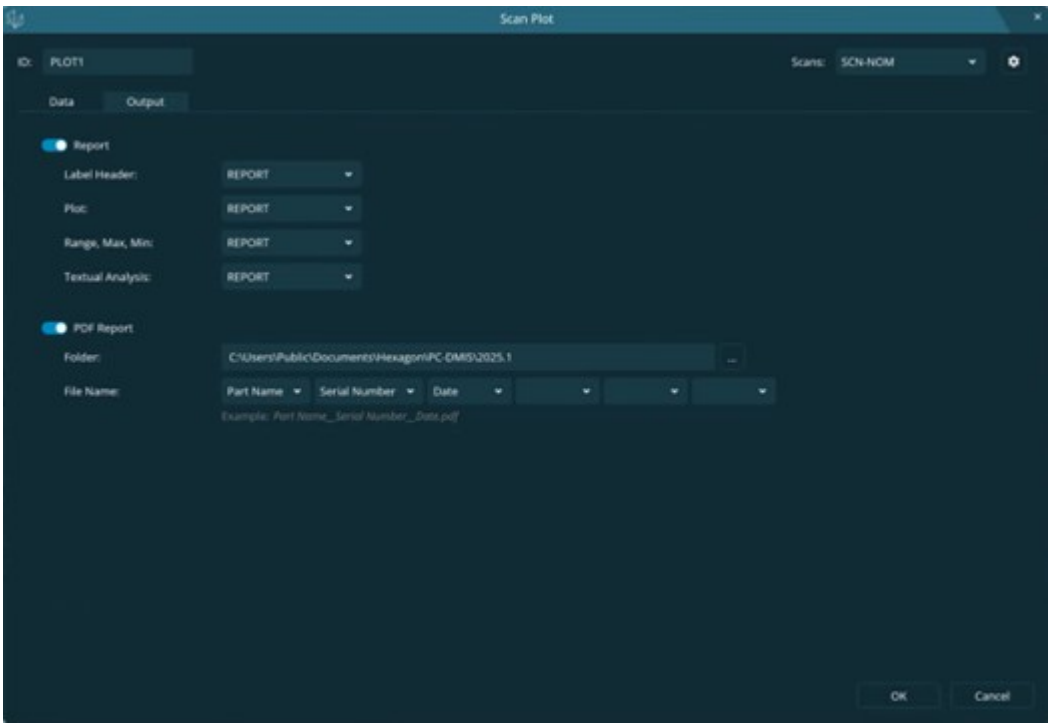
Rotate View – Click this icon to rotate the view by 90 degrees.

Panning and Zooming – Hold the left mouse button and move the plot to the desired position. Use the mouse scroll to zoom in and out.

Copy to Clipboard – You can copy a plot and paste it into any other document for reporting purposes. To copy a plot, right-click in the plot area and select **Copy to Clipboard**.

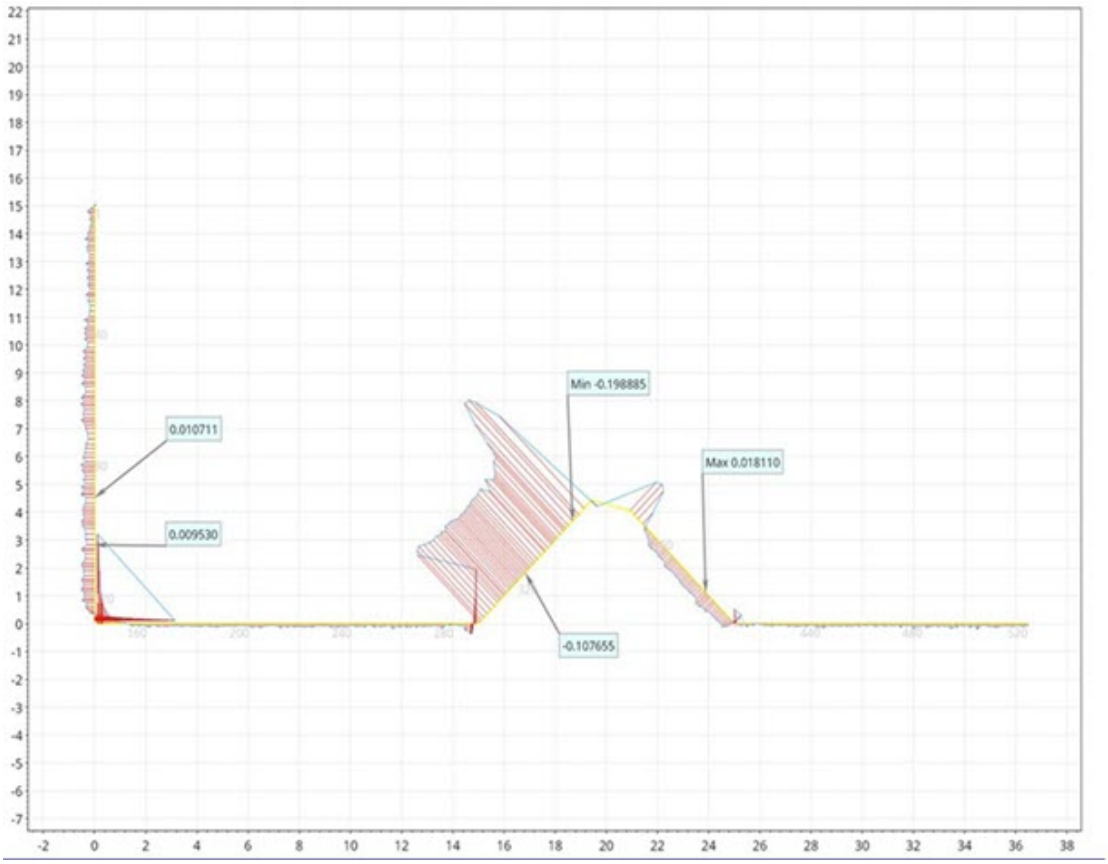
Output Tab

The **Output** tab allows you to set up the report.



- **Report** - Turn on this toggle button to generate a plot report.
  - **Label Header** - This is a **REPORT**/**NONE** option. If this is set to **REPORT**, PC-DMIS inserts the label header in the report. Otherwise, the label header is not inserted into the report.

  - **Plot** - This is a **REPORT**/**NONE** option. If this is set to **REPORT**, PC-DMIS inserts the plot in the report, as displayed in the plot tab. Otherwise, the plot is not inserted into the report.



- **Range**, **Max**, **Min** - This is a **REPORT/NONE** option. If this is set to **REPORT**, PC-DMIS inserts the Range, Max, and Min values in the report. Otherwise, the plot is not inserted into the report.

AX	NOMINAL	MEAS
RANGE	0.000000	0.216994
MAX_T (389)	0.000000	0.018110
MIN_T (345)	0.000000	-0.198885

- **Textual Analysis** - This is a **REPORT/NONE** option. If this is set to **REPORT**, PC-DMIS inserts the Range, Max, and Min values in the report. Otherwise, the plot is not inserted into the report.

HITS	THEO X	Y	Z	VECTOR I	J	K	T Value	+TOL	-TOL	OUTTOL
93	0.000000	4.518857	-2.284768	-1.000000	0.000000	0.000000	0.010711	0.050000	0.050000	0.000000
106	0.000000	2.827435	-2.288707	-1.000000	0.000000	0.000000	0.009530	0.050000	0.050000	0.000000
324	16.830554	1.830554	-2.280705	0.707107	-0.707107	0.000000	-0.107655	0.050000	0.050000	0.057655

- **PDF Report** - Turn on this toggle button to save the report as a PDF file.
  - **Folder** - Use this box to type the path to save the PDF file or click the Browse (…) button and navigate to the location where you want to save it.
  - **File Name** - Use the list box to define a name rule for the PDF file. The options in the list are **Part Name**, **ID**, **Serial Number**, **Revision Number**, **Date**, and **Time**.

Multiple Plots in a Single File

PC-DMIS allows you to save all the plots of a single run or a single part in one PDF file. If the plot file name is found in the selected folder, the Plot command appends the current plot to the same file. If the file is not found, PC-DMIS creates a new PDF file.

Multiple Plots in a Single File

- You can include multiple plots in the report for the same scan. To do this, follow these steps:

    1. Create Plot Command 1:

        a. From the **Label Header** and **Plot** list in the **Output** tab, select **REPORT**.

        b. From the **Range**, **Max**, **Min** and **Textual Analysis** list in the **Output** tab, select **NONE**.

    2. Duplicate the Command:

        a. Copy the first plot command and give it a new name.

        b. Adjust the plot to focus on the area of interest.

        c. From the **Range**, **Max**, **Min** and **Textual Analysis** list in the **Output** tab, select **REPORT**.

    This approach generates a report with three views of the scan.

- You can save multiple plots of the same part or multiple parts of the same type in a single file.

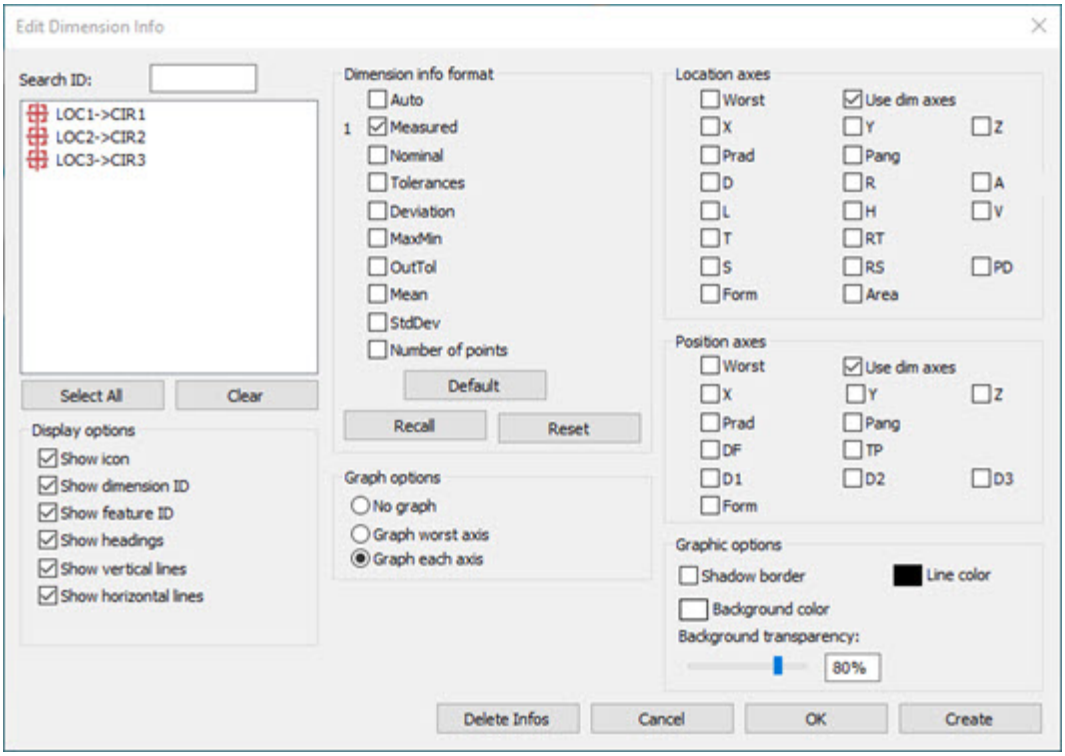
    - **Appending to the same file** - If the file name to save the plot is the same, PC-DMIS appends the new plots to the existing file.

    - **Separate File for Each Plot** - If you include the date and time in the file name, PC-DMIS saves each plot to a separate file.

    - **Single File for a Specific Part** - If you include the part number and serial number in the file name, PC-DMIS saves all the plots for a part with the same serial number to a single file.

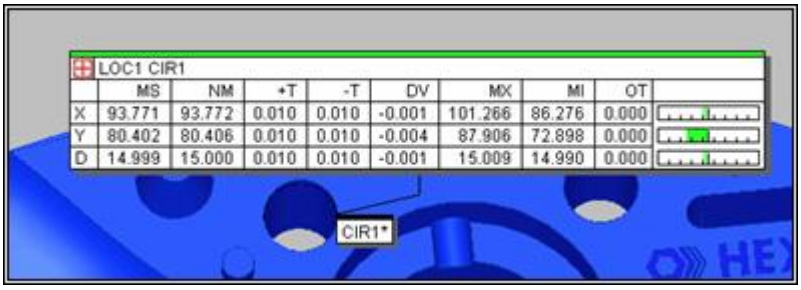
    - **Single File for Multiple Parts** - If you use only the part name in the file name, PC-DMIS saves plots from multiple parts to a single file.

Inserting Dimension Info Boxes



Edit Dimension Info Dialog Box

The **Insert | Report Command | Dimension Info** menu option brings up the **Edit Dimension Info** dialog box. This dialog box allows you to create a `DIMINFO` command that displays selected dimensional information in the Graphic Display window. See "DIMINFO Command" for information on editing a `DIMINFO` command in the Edit Window.



Example of a Dimension Info dialog box.

The dimension list inside the dialog box shows an asterisk (*) if the dimension already contains a DIMINFO text box. The dimension list shows a number sign (#) if the DIMINFO text box is hidden from view in the Graphic Display window.

```
LOC1->CIR1->*
LOC2->CIR2->*#
LOC3->CIR3
```

Example of a Dimension list with existing and hidden DIMINFO boxes.

The **Auto** check box (under the **Dimension Info Format** area of this dialog box) automatically determines the appropriate dimension output format to display, based on the type of dimension that was selected. To override this option, select the desired **Dimension Info format** check box(es). PC-DMIS indicates the order of the output selections by displaying a number to the left of the check box. In this way, you can define the order of the format to meet your needs. You can clear a check box by selecting it a second time.

If this dialog box is open, you can click on a feature in the Graphic Display window to automatically highlight dimensions that use that feature in the **Dimension List** box.

This **Edit Dimension Info** dialog box allows you to edit the dimension information that PC-DMIS displays. The following paragraphs describe the commands and options available to this dialog box.

Dimension Info Creation Rules

When you create a DIMINFO box in the **Edit Dimension Info** dialog box, the **Dimension List** box only displays those dimensions that exist above the cursor's current position.

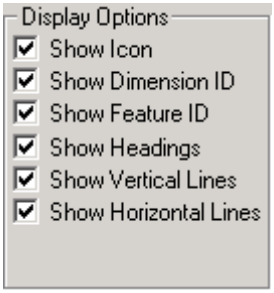
When you create DIMINFO boxes in Text Box Mode and box-select inside the Graphic Display window, PC-DMIS does not create a DIMINFO box for every selected feature. It only creates them for features that exist above the cursor's current position.

If the measurement routine has multiple viewsets, the DIMINFO box only appears in the viewset that contains the corresponding dimension and lower. For example, suppose that your measurement routine has these commands:

```
VIEWSET1
F1 = FEAT
VIEWSET2
VIEWSET3
D1 = DIMENSION
VIEWSET4
```

You could only add a DIMINFO box for feature F1 if the insertion point existed below D1. Once the DIMINFO box is created, PC-DMIS shows it in VIEWSET3 and 4, but not in VIEWSET1 and 2.

Display Options



The **Display Options** area allows different display possibilities for each dimension within the Graphic Display window.

The following options available are:

Show Icon check box
This check box toggles the displays the appropriate dimension icon in the header portion of the **Dimension Info** box.

Show Dimension ID check box
This check box toggles the ability to display the ID for the dimension within the Graphic Display window.

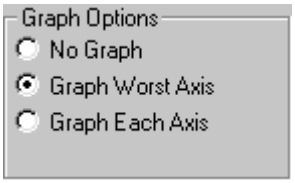
Show Feature ID check box
This check box toggles the ability to display the ID for the feature whose dimensions are being measured.

Show Headings check box
This check box toggles the display of row and column headings in the Dimension Info box.

Show Vertical Lines check box
This check box toggles the display of the vertical lines between columns inside the Dimension Info box.

Show Horizontal Lines check box
This check box toggles the display of the horizontal lines between rows inside the Dimension Info box.

Graph Options

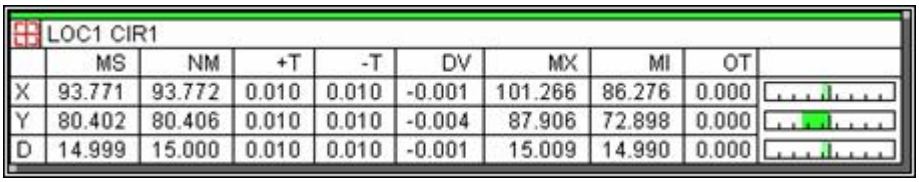


The **Graph Options** area allows for graphical display of the dimension percentage in the **Dimension Info** box.

No Graph - Select this option if you do not want to display a graph in the **Dimension Info** box.

Graph Worst Axis - Select this option to display a graph at the top of the **Dimension Info** box. Only the worst dimension percentage will be used for the graph.

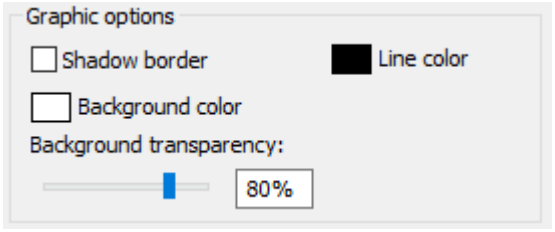
Graph Each Axis - Select this option to display a dimension percentage graph for every axis in the Dimension Info box.



	MS	NM	+T	-T	DV	MX	MI	OT	
X	93.771	93.772	0.010	0.010	-0.001	101.266	86.276	0.000	
Y	80.402	80.406	0.010	0.010	-0.004	87.906	72.898	0.000	
D	14.999	15.000	0.010	0.010	-0.001	15.009	14.990	0.000	

Dimension Info Box

Graphic Options



Graphic Options Area

The **Graphic Options** area allows you to set the graphic options for the current Dimension Info box in the Graphic Display window.

Shadow border
This check box displays a small shadow under the label.

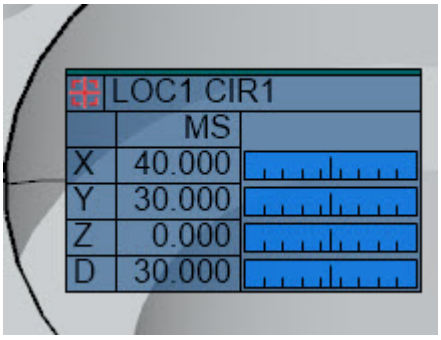
Background Color
This box opens a **Color** dialog box so that you can change the label's background color.

Background Transparency
This slider or the box to its right adjusts the transparency level of the label's

background color. The value can be from 0 (opaque) to 100 (completely transparent).

Line Color
This box opens a **Color** dialog box so that you can change the label's border color.

To change the graphic options for the Dimension Info box, select the desired options from this area and click **Create** or **OK** on the **Edit Dimension Info** dialog box.

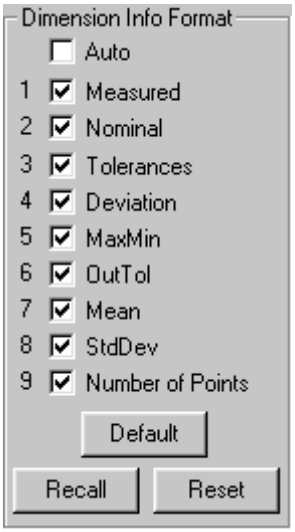


Example of a Dimension Info box set to a blue background color with 50% transparency.

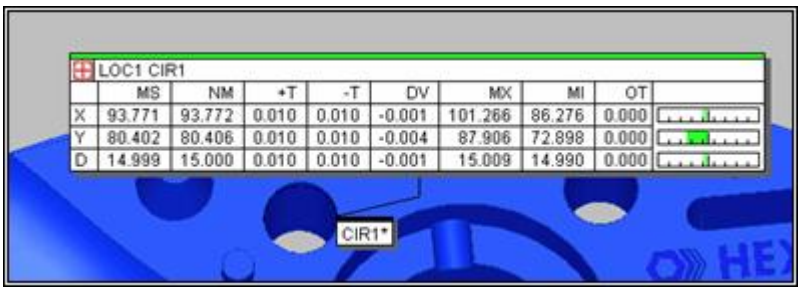
You can also set the default graphic options for future Dimension Info boxes. To do this, select the desired graphic options and then click the **Default** button in the **Dimension Info Format** area.

This topic only applies to legacy feature labels. You can enable legacy feature labels from the **Use legacy graphic labels** check box on the **Feature Appearance** dialog box. If you clear that check box and use the newer feature labels, this area doesn't affect those. For more information, see "Editing Feature Appearance" topic of the "Editing the CAD Display" chapter.

Dimension Info Format



The check boxes in the **Dimension Info Format** area allow you to select the type of information displayed in the Graphic Display window for each dimension. When the **Display** check box is selected from within a dimension's dialog box, PC-DMIS displays the information in the Graphic Display window.



Graphic Display window showing the dimensional data of the CIR1 feature.

The available check boxes in the **Dimension Info** area of the dialog box are:

Auto
The **Auto** check box automatically displays the following information: Measured, Nominal, Tolerances, Deviation, MaxMin, OutTol.

Measured
This check box displays the actual measured dimensions.

Nominal
This check box displays the theoretical values for the dimension.

Tolerances
This check box displays the acceptable tolerance levels either greater than or less than the nominal.

Deviation
This check box displays the deviation of the measured value from the nominal.

MaxMin
This check box displays the maximum and minimum deviation values from the points that make up the dimension. While you can mark this for any dimension, it only makes sense for Line Profile and Surface Profile dimensions. For an example of this, see the "Parameter Settings: Dimension tab" topic of the "Setting Your Preferences" chapter.

OutTol
This check box displays how far out of tolerance the measured value is from the nominal and tolerance values.

Mean
This check box displays the average of all the deviations for the dimension.

StdDev
This check box displays the standard deviation of all the deviations for the dimension.

Number of Points
This check box displays the number of points used to measure the feature of the dimension.

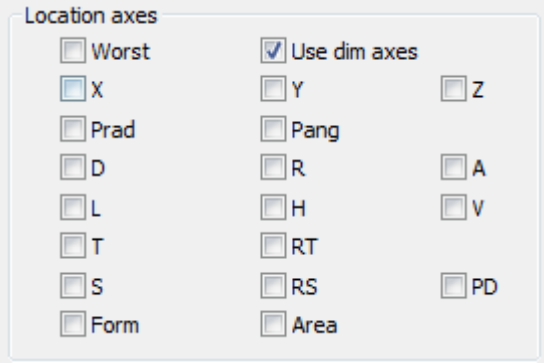
The command buttons are:

Recall - Recalls the set default.

Default - Sets the current selection of check boxes as the default.

Reset - Clears out any check marked boxes in this section and selects the **Auto** check box.

Location Axes



This table describes the available **Location Axes** and their functions that you can choose from when you edit dimension information.

The following options are available:

Worst check box
This option uses the axis that gives the worst out of tolerance condition. PC-DMIS searches through the available axes and selects the one that gives the worst case scenario for the dimension.

Use Dim Axes check box
This option simply uses the axes that were previously defined in the dimension's dialog box.

X check box
This option displays the X axis value.

Y check box
This option displays the Y axis value.

Z check box
This option displays the Z axis value.

Prad check box
Prad stands for Polar Radius. This option selects Polar coordinates.

Pang check box
Pang stands for Polar Angle. This option selects Polar coordinates.

Area check box
This option displays the area of a selected Blob feature. It appears as AR in the DIMINFO box and in the report. (PC-DMIS Vision uses Blob features.) Legacy reports only show this information if you also mark the **Use dim axes** check box from the **Position** area. For information on **Use dim axes**, see "Position Axes".

D check box
This option displays the Diameter value.

R check box
This option displays the Radius (half of Diameter) value.

A check box
This option displays the Angle (for cones) value.

L check box
This option displays the Length (used for cylinders, slots, cones, and ellipses).

H check box
This option displays the Height.

V check box
This option displays the Vector location.

T check box
For points on curved surfaces, this option displays the deviation from the nominal along the feature's vector.

RT check box
This option displays the deviation along the report vector.

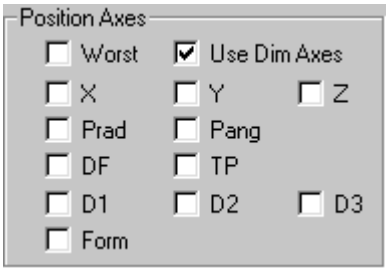
S check box
This option displays the deviation along the surface vector.

RS check box
This option displays the deviation along the surface report vector.

PD check box
This option displays the diameter of a circle (perpendicular to the pin vector).

Form check box
This option displays the integrated form dimension for the feature. See "Default Axes for Location Dimensions" in the "Using Legacy Dimensions" chapter.

Position Axes



This table describes the available **Position Axes** and their functions that you can choose from while editing dimension information.

The following options are available:

Worst check box
This check box uses the axis that gives the worst out of tolerance condition. PC-DMIS searches through the available axes and selects the one that gives the worst case scenario for the dimension.

Use Dim Axes check box
This check box simply uses the axes that were previously defined in the dimension's dialog box.

X check box
This check box displays the X axis value.

Y check box
This check box displays the Y axis value.

Z check box
This check box displays the Z axis value.

Prad check box
Prad stands for Polar Radius. This option selects Polar coordinates.

Pang check box
Pang stands for Polar Angle. This check box selects Polar coordinates.

DF check box
This check box displays diameter of the feature.

TP check box
This check box displays the position tolerance and its associated deviation.

D1 check box
This check box displays the diameter/width value of the first datum.

D2 check box
This check box displays the diameter/width value of the second datum.

D3 check box
This check box displays the diameter/width value of the third datum.

Form check box
This check box displays the integrated form dimension for the feature. See "Default Axes for Position Dimensions" in the "Using Legacy Dimensions" chapter.

Delete Infos

The **Delete Infos** button removes all `DIMINFO` commands from the measurement routine.

DIMINFO Command

You can create the `DIMINFO` command from the **Edit Dimension Info** dialog box (**Insert | Report Command | Dimension Info**).

When you insert a `DIMINFO` command in the measurement routine, PC-DMIS displays the dimension data in the Graphic Display window. You can modify the `DIMINFO` command in the Edit window. The command line for the `DIMINFO` command reads:

```
DIMINFO/Dim ID ;ICON,DIMID,FEATID,VERT,HORIZ; HEADINGS,
"GRAPH OPTION";"DIMENSION OUTPUT", ,$
"AXES OUTPUT"
```

Dim ID - This is the ID of the selected dimension.

ICON - A toggle string that shows a blank when it is OFF, and "ICON" when it is ON. When it is ON, the `DIMINFO` displays the dimension or feature ID with the appropriate icon.

DIMID - A toggle string that shows a blank when it is OFF, and "DIMID" when it is ON. When it is ON, the DIMINFO displays the dimension ID with the dimensional information.

FEATID - A toggle string that shows a blank when it is OFF, and "FEATID" when it is ON. When it is ON, the DIMINFO displays the ID of the dimension's first feature with the dimensional information.

VERT - A toggle string that shows a blank when it is OFF, and "VERT" when it is ON. When it is ON, the `DIMINFO` displays with the vertical lines between the columns.

HORIZ - A toggle string that shows a blank when it is OFF, and "HORIZ" when it is ON. When it is ON, the `DIMINFO` displays with the horizontal lines between the rows.

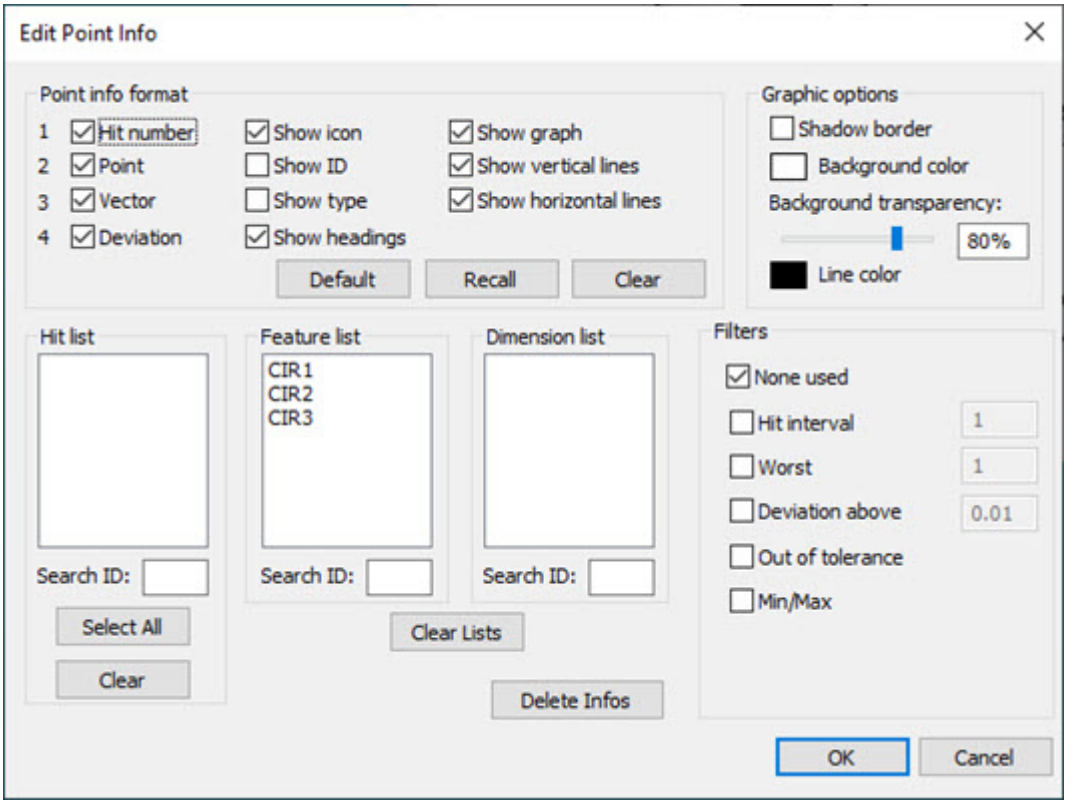
HEADINGS - A toggle string that shows a blank when it is OFF and "HEADINGS" when it is ON. When it is ON, PC-DMIS displays the column and row headers in the **Dimension Info** box.

GRAPH OPTION - The graph option for graphical display of the dimension percentage in the **Dimension Info** box. Possible toggle values are GRAPH WORST and GRAPH AXIS. When the toggle string shows a blank, PC-DMIS does not show the graph in the box.

DIMENSION OUTPUT - This defines the format of the displayed dimension information based on the order of selection. Available output options are MEAS, NOM, TOL, DEV, MAXMIN, OUTTOL, MEAN, STDDEV, and NUMPTS. You cannot duplicate options.

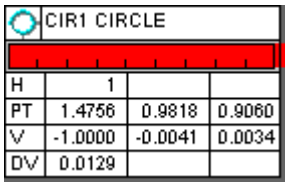
AXES OUTPUT - This defines the format of the displayed dimension axes based on the order of selection. Available axes depend upon the dimension type and include all location and position axes. If you toggle USEDIM in the first axis position, the DIMINFO displays the axes specified in the dimension. If you toggle WORST in any axis position, PC-DMIS displays the axis that produced the largest out-of-tolerance value. This output is only available for Location and Position dimensions . You cannot duplicate axes.

Inserting Point Info Boxes



Edit Point Info Dialog Box

The **Insert | Report Command | Point Info** menu option brings up the **Edit Point Info** dialog box. This dialog box creates a `POINTINFO` command that displays information boxes in the Graphic Display for individual points. Each information box may display the hit number, the XYZ position, the IJK vector, and the deviation related to a particular hit. For information on the `POINTINFO` command in the Edit Window, see the "POINTINFO Command" topic.



Point Info box showing the first hit in a circle feature

To insert point information in the Graphic Display window, follow these steps:

1. Choose **Insert | Report Command | Point Info** to open the **Edit Point Info** dialog box.

2. From the **Feature list** or the **Dimension list**, select at least one feature or dimension. The individual points associated with the item you choose appear in the **Hits List** box. If you choose more than one dimension or feature, the **Hit list** does

not appear, and all the points from the dimensions or features appear in the information box.

3. From the **Hit List** box, select one or more points that you want to show.
4. From the **Point info format** area, mark the items that you want to show. Any numbers that appear next to check boxes indicate the row order.
5. Click the **OK** button to create the information boxes in the Graphic Display window.

You can save, change, and move the formats similar manner to the **Dimension Info** box formats. For information on Dimension Info boxes, see "Inserting Dimension Info Boxes".

Point Info Format

This portion of the dialog box allows you to determine the type of information displayed in the Graphic Display area next to a feature or dimension. The check boxes in this section must be used in conjunction with the **Create** (**Apply** when editing) or the **OK** button to either select or remove information to display. Any numbers that appear next to check boxes indicate the row order.

Hit Number - This check box displays the hit sequence of a point on a feature.

If a circle generally takes four hits, and one of the hits used to measure that circle was selected, this option displays a number (one through four) indicating when that hit was taken in the sequence: 1st, 2nd, 3rd, or 4th.

Point - This check box displays the X, Y, and Z point location of the hit selected.

Vector - This check box displays the IJK vector of the hit selected.

Deviation - This check box displays the deviation of the point from the nominal.

- *For features*, the point deviation is the distance between the measured hits and the actual hits.
- *For dimensions*, the point deviation is related to the dimension type and is the same as the deviation displayed for the individual points in the textual analysis of the dimension.

Show Icon - This check box displays the feature's icon in the header portion of the Point Info box.

Show ID - This check box allows you to display the ID of a particular feature or dimension.

Show Type - This check box allows you to display the type of feature or dimension displayed.

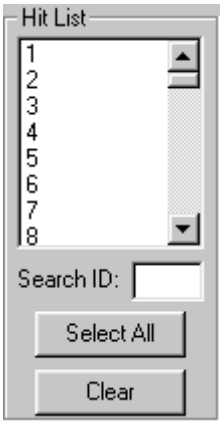
Show Headings - This check box allows you to display the row headers in the Point Info box.

Show Graph - This check box allows you to display the dimension percentage graph in the Point Info box.

Show Vertical Lines - This check box allows you to display or hide the vertical lines inside the Point Info box.

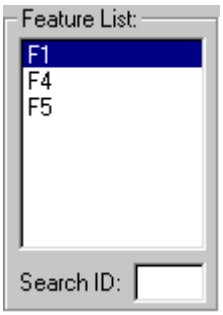
Show Horizontal Lines - This check box allows you to display or hide the horizontal lines inside the Point Info box.

Hit List



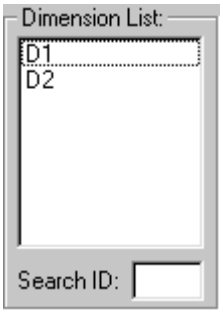
The **Hit List** box contains a list of all the hits associated with a particular feature or dimension. Simply select the hits whose information you would like to view.

Feature List Area



The **Feature List** box contains a list of all the features associated with a particular measurement routine. Select the feature whose hit information you would like to view. The hits associated with that feature will be displayed in the **Hit List** box.

Dimension List Area



The **Dimension List** box contains a list of all the dimensions that are associated with a particular measurement routine. Select the dimension whose hit information you would like to view.

PC-DMIS shows the hits that are associated with that dimension in the **Hit List** box.

Delete Infos

The **Delete Infos** button removes all `POINTINFO` commands from the measurement routine.

POINTINFO Command

The `POINTINFO` command is created by the **Edit Point Info** dialog box (**Insert | Report Command | Edit Point info**).

When a `POINTINFO` command has been inserted in the measurement routine, a **Point Info** box is displayed in the Graphic Display window. The `POINTINFO` command can be modified in the Edit window. The command line for the `POINTINFO` command reads:

```
POINTINFO/Dim ID or Feat ID; FILTER FILTER_TYPE
FILTER_NUM;ICON,ID,TYPE,VERT,HORIZ; HEADINGS,
GRAPH; ”OUTPUT FORMAT” ,$
"HIT NUMBERS"
```

Dim ID or Feat ID - The ID of the dimension or feature to be displayed.

FILTER_TYPE - A toggle string that shows the filter option of NONE, INTERVAL, WORST, DEVIATION, or OUTTOL.

FILTER_NUM - A number field available when the filter option is INTERVAL, WORST, or DEVIATION.

ICON - A toggle string that shows a blank when it is OFF, and "ICON" when it is ON. When it is ON, the `POINTINFO` will display the dimension or feature ID with the appropriate icon.

ID - A toggle string that shows a blank when it is OFF, and "ID" when it is ON. When it is ON, the `POINTINFO` will display the dimension or feature ID with the point information.

TYPE - A toggle string that shows a blank when it is OFF, and "TYPE" when it is ON. When it is ON, the `POINTINFO` will display feature or dimension type (i.e. CIRCLE, POINT, ROUNDNESS, LOCATION) with the point information.

VERT - A toggle string that shows a blank when it is OFF, and "VERT" when it is ON. When it is ON, the `POINTINFO` will display with the vertical lines between the columns.

HORIZ - A toggle string that shows a blank when it is OFF, and "HORIZ" when it is ON. When it is ON, the `POINTINFO` will display with the horizontal lines between the rows.

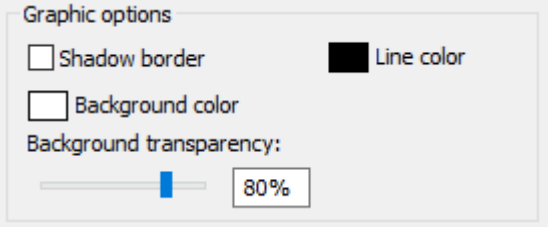
HEADINGS - A toggle string that shows a blank when it is OFF and "HEADINGS" when it is ON. When it is ON, the row headers will be displayed in the **Point Info** box.

GRAPH - A toggle string that shows a blank when it is OFF and "GRAPH" when it is ON. When it is ON, the dimension percentage will be graphically displayed in the **Point Info** box.

OUTPUT FORMAT - The format of the point information that will be displayed based on the order of selection. Available output options are HIT, PT, V, and DEV. Options cannot be duplicated.

HIT NUMBERS - The hit numbers are the specific numbered hits that are controlled by this particular `POINTINFO` command. Multiple hits can be displayed using the same `POINTINFO` command. Each field in the hit numbers is a toggle string. When a hit is ON, the string in the number position is the hit number (1,2,3, etc.). When a hit is OFF, the toggle string displays a blank.

Graphic Options



Graphic Options Area

The **Graphic Options** area allows you to set the graphic options for the current Point Info box in the Graphic Display window.

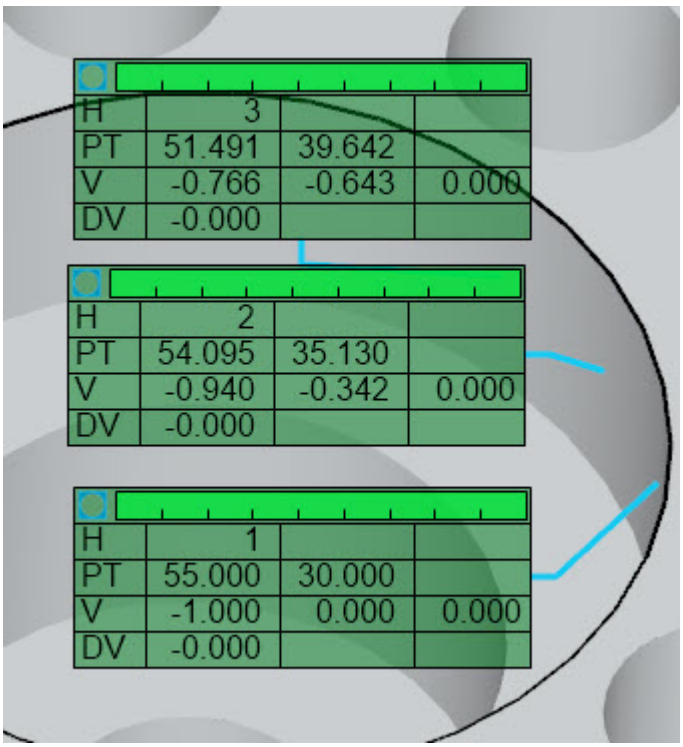
Shadow border
This check box displays a small shadow under the label.

Background Color
This box opens a **Color** dialog box so that you can change the label's background color.

Background Transparency
This slider or the box to its right adjusts the transparency level of the label's background color. The value can be from 0 (opaque) to 100 (completely transparent).

Line Color
This box opens a **Color** dialog box so that you can change the label's border color.

To change the graphic options for the Point Info box, select the desired options from this area and click **Create** or **OK** on the **Edit Point Info** dialog box.

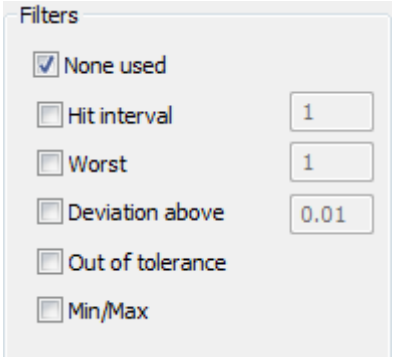


Example of Point Info boxes set to a green background color with 50% transparency.

You can set the default graphic options for future Point Info boxes. To do this, select the desired graphic options and then click the **Default** button in the **Point Info Format** area.

> This topic only applies to legacy feature labels. You can enable legacy feature labels from the **Use legacy graphic labels** check box on the **Feature Appearance** dialog box. If you clear that check box and use the newer feature labels, this area doesn't affect those. For more information, see "Editing Feature Appearance" topic of the "Editing the CAD Display" chapter.

Filters



Filters Area

The **Filters** area allows you to set filter options that PC-DMIS uses when it displays individual point information.

None Used - This filter option tells PC-DMIS not to use any filtering for the `POINTINFO` command. All hits selected will be displayed with the `POINTINFO` command.

Hit Interval - This filter option checks the integer value in the box to its right, and only displays the particular hit interval chosen with the `POINTINFO` command. For example, if 2 were chosen, then every 2nd hit would be displayed, if 3, then every 3rd hit, if 4 then every 4th hit, etc.

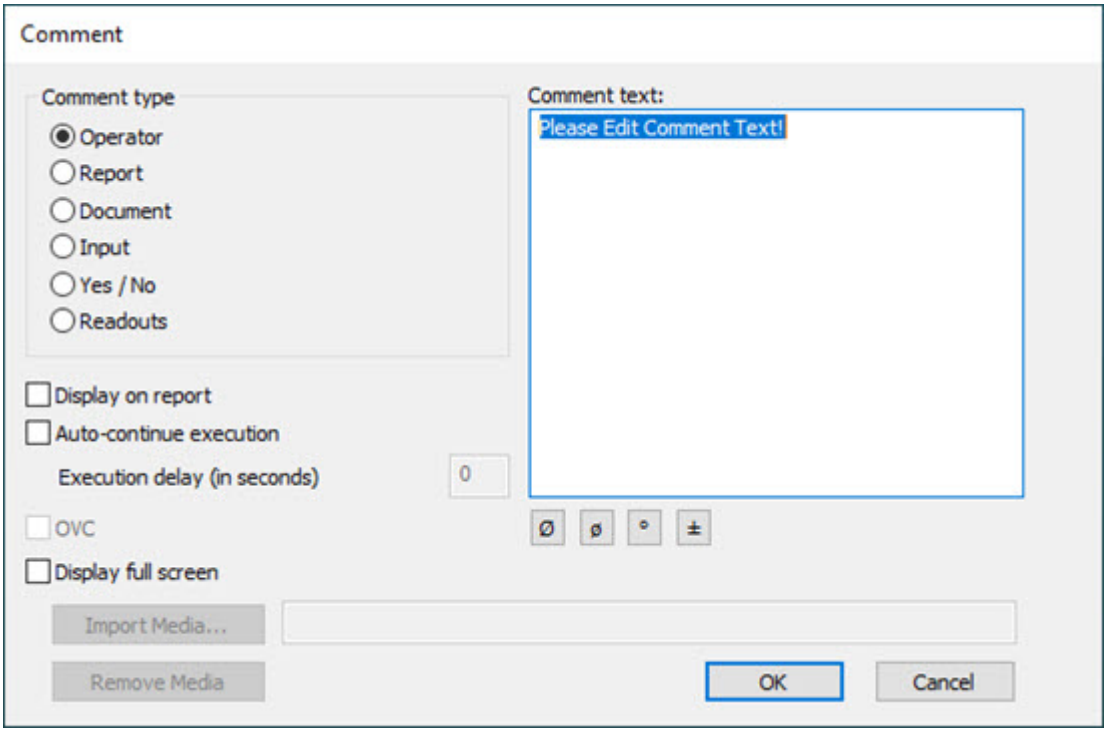
Worst - This filter option checks the integer value in the box to its right, and only displays the hits that give the worst deviations for the dimension. For example, if you type 3 in the box, the `POINTINFO` command only displays the three worst deviations.

Deviation Above - This filter option checks the value in the box to its right, and only displays the hits that have deviations above the value in the box for the dimension. For example, if you type 0.1 in the box, the `POINTINFO` command only displays hits that have deviations above the value in the box.

Out of Tolerance - This filter option only displays the hits that have deviations that are out of tolerance. For hits related to features (not related to dimensions), only those hits with deviations larger than the **Show Deviations Tolerance** on the **Setup Options** dialog box's **General** tab are displayed. (See "Show Deviations Tolerance" in the "Setting Your Preferences" chapter.)

Min/Max - This filter option only shows those hits that are the maximum and minimum deviation values for the dimension. For an example of MaxMin values, see the "Parameter Settings: Dimension tab" topic of the "Setting Your Preferences" chapter.

Inserting Programmer Comments



Comment Dialog Box

The **Insert | Report Command | Comment** menu option allows you to add operator notes, instructions, or supported media files into the Edit window that will be displayed in a message box when the measurement routine is executed or when the inspection report is printed. There is no limit to the length of the comment text; however, in Command Mode only 255 characters will fit per line. When the text reaches the right side of the Edit window, press the Enter key. (This allows the entire text to be viewed in the **Comment Text** box.) To create a new line, place the cursor in the desired location of the **Comment Text** box and press the Enter key.

Typing Comments

You can also type a comment into the Edit window in Command Mode. To do this, type `COMMENT` , press the Tab key on your keyboard, and then type the comment type you want: such as `OPERATOR` or `REPORT` and so forth. Press Tab to accept the command or to move to the field containing the comment text.

After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

Using Variables in Comment Strings

Suppose you want to concatenate, or add, a variable to an existing comment string. You can do this in two ways. First, you can press Enter, and type the variable on a new comment line, like this:

```
C1          =COMMENT/INPUT,NO,Full Auto-Continue=NO,
             Type your variable
             ASSIGN/V1=C1.INPUT
             COMMENT/OPER,NO,Full Auto-Continue=NO,
             Your V1 variable is
             V1
```

Second, you can place variables and your comment string on the same line by placing quotation marks around the non-variable string and using the plus sign operator to add the variable to the string, like this:

```
COMMENT/OPER,NO,Full Screen=YES,Auto-Continue=NO,

"Your V1 variable is "+V1
```

Changing Comment Color

You may want your comments to stand out more. You can do this by changing your comment color.

- To change them in the Edit window, see "Defining Edit Window Colors".
- To change comment colors for comments that appear in the Report window, change the **Colors** property of the TextReportObject in the in the Report Template Editor.

Another way to change a comment's color in the report is to insert this special prefix directly in front of the comment text:

```
~~#
```

The # represents 01, 02, 03, or 04. These stand for the Marked, Unmarked, Step Mode, or Error colors respectively. For information on these colors, see the **Edit Window Colors** dialog box.

So, to display some comment text in the text color assigned to Error, your command needs to look like this:

```
COMMENT/rept, ~~04 Some Comment Text
```

Operator

The **Operator** option displays the comment text to the operator when the software executes the comment command.

To use **Operator** option:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **Operator** option.
4. In the **Comment Text** box, type the comment text.
5. Click the **OK** button to close the dialog box and insert the comment.

   The Edit window command line for this option reads:
   ```
   COMMENT/OPER,TOG1,FULL SCREEN=TOG2,AUTO-CONTINUE=TOG3,
   comment text
   ```

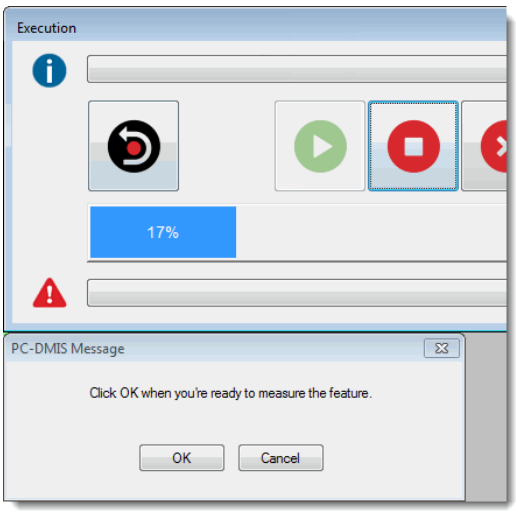
   TOG1 - This YES/NO field allows you to specify whether or not the comment shows up in the inspection report.

   TOG2 - This YES/NO field determines whether or not the comment will be displayed to fill the entire screen.

   TOG3 - This YES/NO field determines whether or not the comment will use a countdown timer and automatically continue execution once the timer reaches zero. When the countdown timer reaches zero, PC-DMIS will continue as if you had clicked the **OK** button.

How it Works

When PC-DMIS executes the measurement routine, a message box displays the appropriate comments just under the **Execution** dialog box.



PC-DMIS Message Dialog Box

PC-DMIS doesn't allow you to edit this message. Click the **OK** button to continue execution. Click **Cancel** to cancel measurement routine execution.

Report

Report

The **Report** option sends text to the inspection report.

To use the **Report** option:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **Report** option.
4. In the **Comment Text** box, type the comment text.
5. Click the **OK** button to close the dialog box and insert the comment.

When PC-DMIS executes the measurement routine, these messages are not viewed. PC-DMIS does, however, send these comments to the inspection report when it is printed.

The Edit window command line for this option reads:

```
COMMENT/REPT,

comment text
```

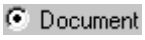
To visually separate content, you can include dashes (or other characters) as a second comment line, like this:

```
COMMENT/REPT,

Inspection Report Comment

COMMENT/REPT,

-----------------------------------
```

The above would look like this in the report:

```
Inspection Report Comment

-----------------------------------
```

Document



The **Document** option adds text to the internal measurement routine. Its only use is to store programmer notes. It does not appear in the inspection report (unless you mark **Display On Report**), nor does it display anything when executed.

To use the **Document** option:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment type** area of the **Comment** dialog box, select the **Document** option.
4. In the **Comment Text** box, type the comment text.
5. Click the **OK** button to close the dialog box and insert the comment.

The Edit window command line for this option reads:

```
$$ TOG1,
    Please Edit Comment Text!
```

TOG1 = This YES/NO field allows you to specify whether the comment shows up in the inspection report or not.

If your measurement routine contains unsupported commands, (perhaps from a newer version of PC-DMIS) when you open the measurement routine, PC-DMIS converts them to this comment type. For more information, see "Save As" in the "Using Basic File Options" chapter.

Input

The Input option is similar to the **Operator** option in that it allows you to display text when executing a measurement routine. In addition to displaying a message box with the previously entered text, a comment box will appear. This allows you to enter *numerical* information that will be written to the inspection report.

This option is extremely helpful to users wishing to enter a part number or serial number while executing the measurement routine.

To use the **Input** option, follow these steps:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **Input** option.
4. In the **Comment Text** box, type the comment text.
5. Click the **OK** button to close the dialog box and insert the comment.

   The Edit window command line for this option reads:

   ```
   comment ID=COMMENT/INPUT,TOG1,Full Screen=TOG2,
   comment text
   ```

   TOG1 = This YES/NO field allows you to specify whether or the comment shows up in the inspection report.

   TOG2 = This YES/NO field allows you to specify whether or not the comment fills the entire screen when displayed.

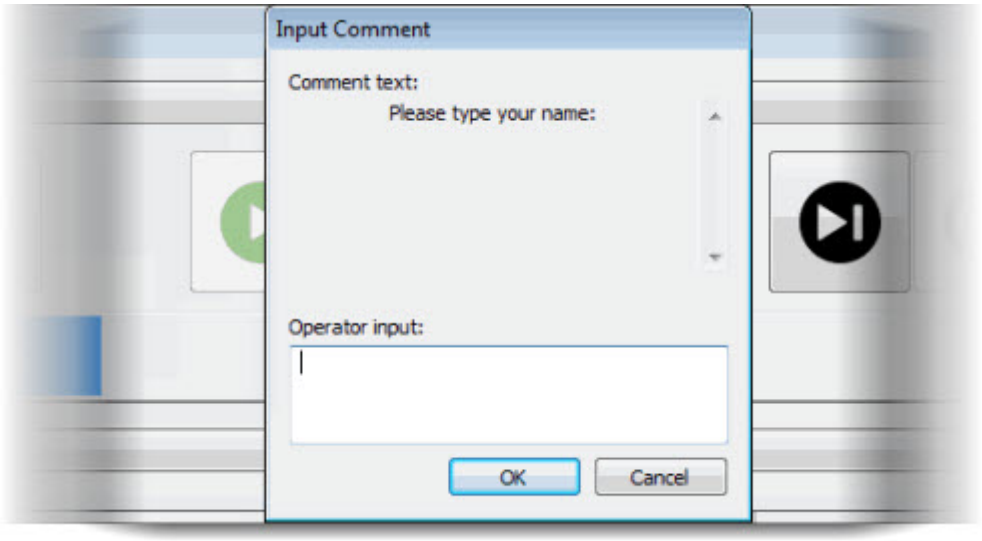
The input is assigned to the variable specified on the left hand side of the command. The variable is coerced to be of type string. This variable can then be used anywhere an expression is allowed by using this syntax <COMMENT ID>.INPUT. For example if your comment ID is C1, you can pass that variable into another variable like this:

```
C1 = COMMENT/INPUT,NO,Full Screen=NO,
```

```
"Please type your name:"
ASSIGN/V1=C1.INPUT
```

How it Works

During execution, the message box displays the prompt to enter the necessary number (such as a serial number) and then it displays this value in the inspection report. This prompt appears on top of the **Execution** dialog box:



Input Comment Dialog Box

Once the operator types his text and clicks **OK**, PC-DMIS stores it in the comment's variable. The variable is ID.INPUT (by default the ID for the variable is C1 for the first Input comment, C2 for the second, and so on). If the operator clicks **Cancel**, it stops execution.

Yes / No

The **Yes / No** option functions similarly to the Operator option in that it allows you to display text when you execute a measurement routine. In addition, **Yes**, **No** and **Cancel** buttons appear at the bottom of the dialog box of the **PC-DMIS Message** dialog box; this enables the operator to answer yes or no to simple questions.

To use the **Yes / No** option, follow these steps:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **Yes / No** option.

4. In the **Comment Text** box, enter the comment text.

5. If you do not click Yes or No on the **PC-DMIS Message** dialog box and you want the measurement routine to auto-execute after a set time, select the **Auto-continue execution** check box. You can then enter the number of seconds for the delay in the **Execution delay (in seconds)** box. For details, see "Auto-Continue Execution".

6. Click the **OK** button to close the dialog box and insert the comment.

   The Edit window command line for this option reads:

   ```
   comment ID = COMMENT/YESNO,TOG1,FULL SCREEN=TOG2,AUTO-CONTINUE=TOG3,

   comment text
   ```

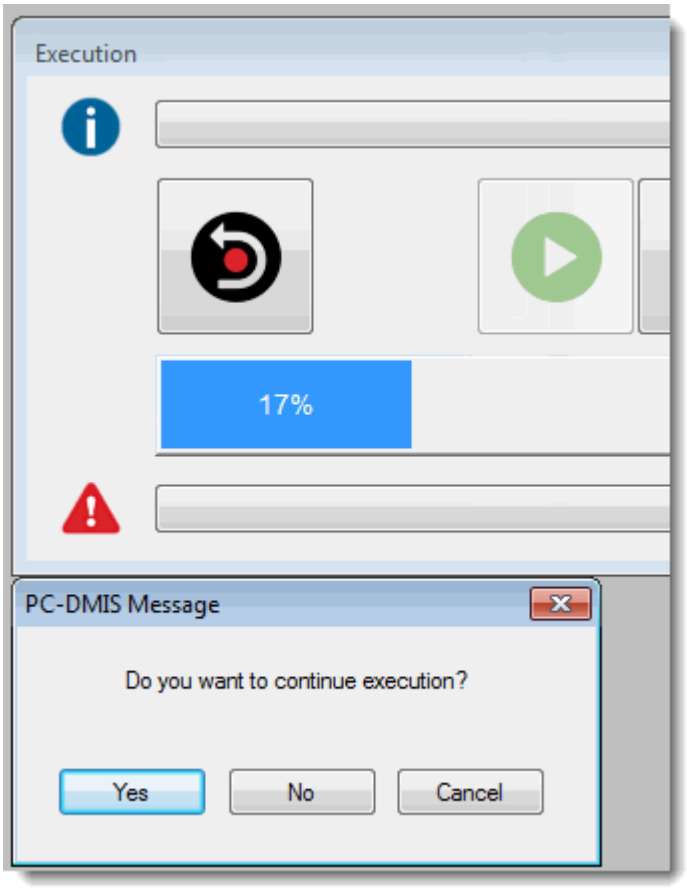
   TOG1 - This YES/NO field allows you to specify whether or not the comment shows up in the inspection report.

   TOG2 - This YES/NO field allows you to specify whether or not the comment fills the entire screen when displayed.

   TOG3 - This YES/NO field determines whether or not the comment will use a count-down timer and automatically continue execution once the timer reaches zero. If the operator does not click a button by the time 0 is reached, the measurement routine execution continues with the assumption that the operator clicked the **Yes** button.

How it Works

During execution, the **PC-DMIS Message** dialog box appears below the **Execution** dialog box. This enables the operator to click **Yes**, **No**, or **Cancel**:



PC-DMIS Message dialog box with buttons.

Execution pauses until the operator clicks a button. A variable stores the answer. The variable is C1.INPUT.

The answer the operator selects is accessible using expressions with the ID of the comment.

Example 1: If the comment has an ID of C1 and the operator clicks the **Yes** button, the expression "C1.INPUT" contains the value "YES". If the operator clicks the **No** button, the expression "C1.INPUT" contains the value " NO ". This option is useful to users who want to branch or loop on a "YES" or "NO" string response.

Example 2: If the comment has an ID of C1 and the operator clicks the **Yes** button, the expression "C1.INPUTVALUE" contains the value 1. If the operator clicks the **No** button, the expression "C1.INPUTVALUE" contains the value 2. This option is useful to users who want to branch or loop on a 1 or 2 integer response.

In either case, if the operator clicks **Cancel**, the measurement routine execution is canceled.

When you use a conditional branching statement to test for the value of a YES / NO comment using the .INPUT expression, be aware that your test must look for an uppercase "YES" or "NO" value. A lowercase "Yes" or "No" does not work. For information on measurement routine branching, see the "Branching by Using Flow Control" chapter.

Readouts

The **Readouts** option displays the comment text at the bottom of the Probe Readouts window. You must select the **Display prompt history** check box on the **Probe Readouts Setup** dialog box for this to work.

To use the **Readouts** option:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **Readouts** option.
4. In the **Comment Text** box, type the comment text.
5. Click the **OK** button to close the dialog box and insert the comment.

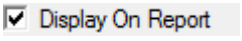
   The Edit window command line for this option reads:
   ```
   COMMENT/READOUTS,TOG1,
   comment text
   ```

   TOG1 = This YES / NO field allows you to specify whether or not the comment shows up in the inspection report.

For more information on setting up Probe Readouts, see "Setting Up the Probe Readouts Window" in the "Setting Your Preferences" chapter.

For more information on the Probe Readouts window, see "Using the Probe Readouts Window" in the "Using Other Windows, Editors, and Tools" chapter.

Display On Report



This check box allows you to determine whether or not your comment and any submitted text (if using an Operator comment, for example) appear in the final report generated in the

Report window. If selected, the comment's YES/NO toggle field, which specifies whether or not a comment shows up in the report, is set to YES.

This check box is hidden if you select the **Report** comment type.

The Edit window command line for this option reads:

```
COMMENT/TOG1,TOG2,
comment text
```

TOG1 = Any comment type except Report.

TOG2 = This YES/NO field allows you to specify whether or the comment shows up in the inspection report.

Auto-Continue Execution

Auto-continue execution: This check box determines whether a comment that PC-DMIS displays during execution closes itself after a specified number of seconds. By default, this check box is not selected.

Operators may find this useful when they use a measuring device that is not near the computer, and they want the execution to continue after a certain amount of time. If you select this check box, the **Execution delay (in seconds)** box becomes available.

Execution delay (in seconds): This box defines the number of seconds to delay execution while PC-DMIS displays the comment. You can specify any number of seconds from 1 to 600, inclusive.

- If you use an execution delay with a movie file and the movie's play time exceeds the delay value, PC-DMIS waits until the movie finishes playing before continuing.
- If your movie value is shorter than the specified execution delay, the comment displays until the execution delay counts down to zero.

During execution, the software displays a count-down timer on the Operator comment type's **OK** button inside parentheses:

For the Yes / No comment type, it appears on the **Yes** button.

Once the timer reaches zero:

- For an Operator comment, execution continues as if the operator clicked the **OK** button.

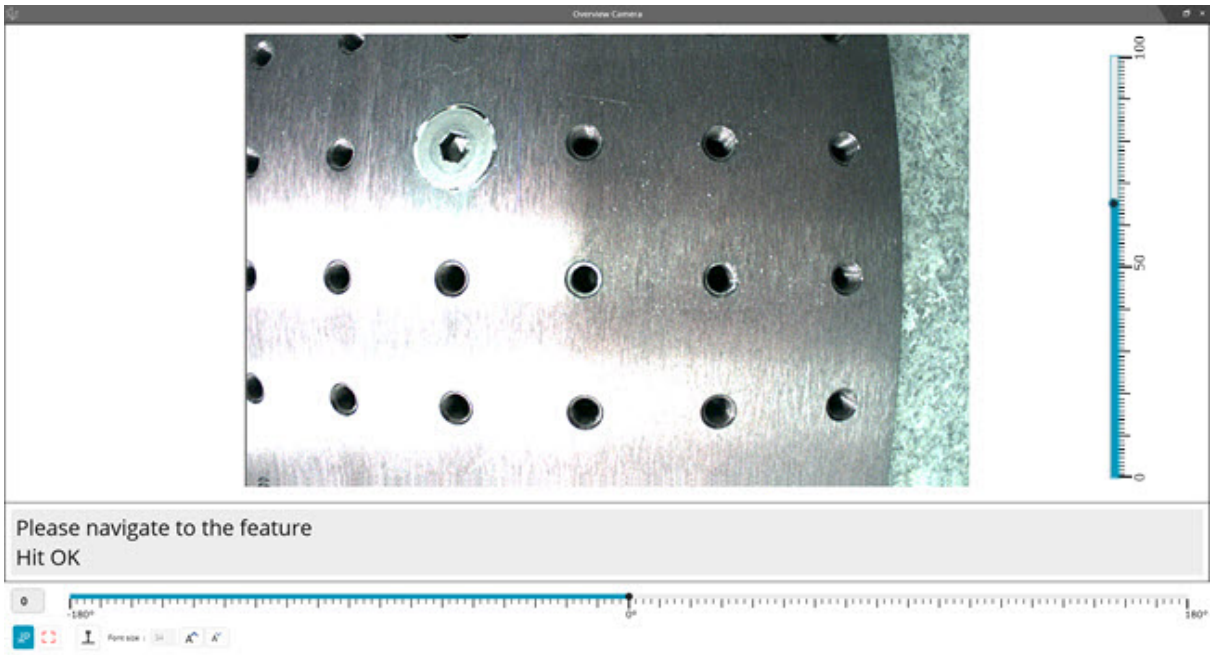
- For a Yes / No comment, execution continues as if the operator clicked the **Yes** button.

OVC

The **OVC** option displays the **Overview Camera** dialog box, which you can use to enter comments.

To use the **OVC** option, follow these steps:

1. Place the cursor in the Edit window where you want to insert the comment command.
2. Select the **Insert | Report Command | Comment** menu option to open the **Comment** dialog box.
3. From the **Comment** dialog box, select the **OVC** check box to open the **Overview Camera** dialog box.



Overview Camera Dialog Box

4. In the box, type the comment for the operator. You can use the and icons to increase and decrease the font size respectively.
5. Click "X" to close the **Overview Camera** dialog box.
6. From the **Comment** dialog box, click the **OK** button to close the dialog box and insert the comment.

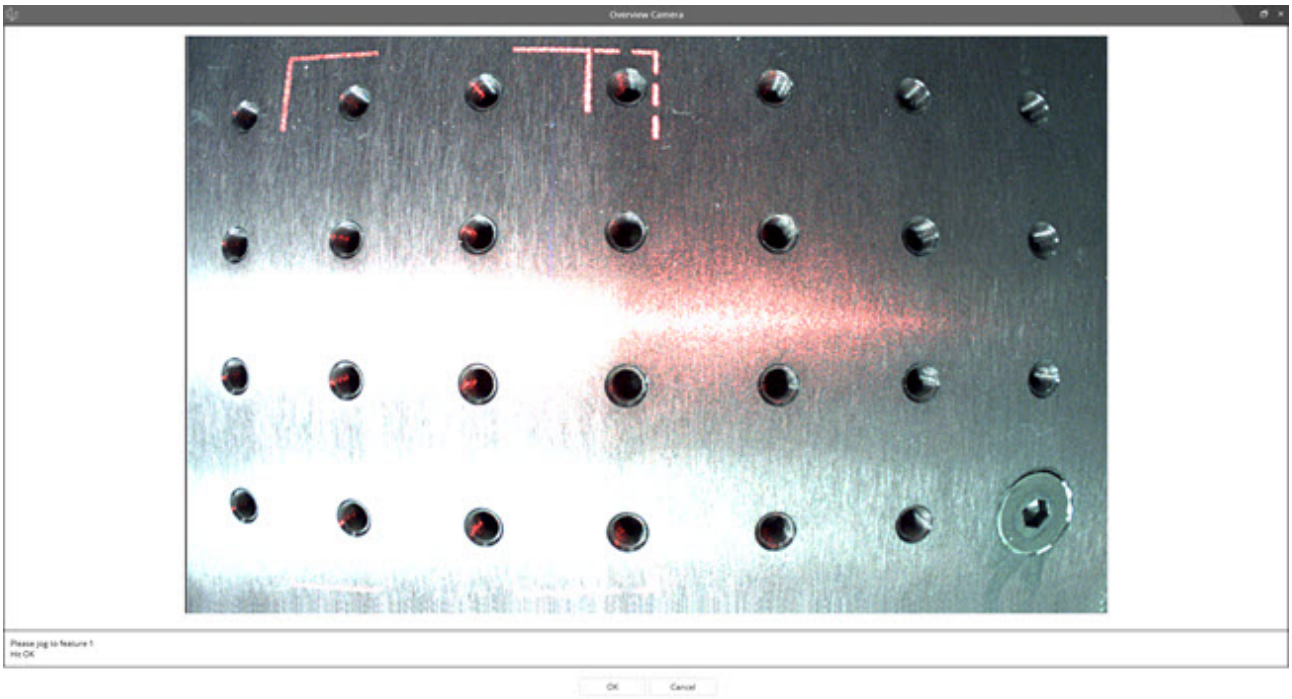
   The Edit window command line for this option reads:

```
 COMMENT/OPER,TOG1,FULL SCREEN=TOG2,AUTO-
CONTINUE=TOG3,OVC=TOG4,
comment text
```

- **TOG1** - This is a YES/NO field. If this is set to YES, PC-DMIS inserts the comment into the inspection report. Otherwise, the comment is not inserted into the report.
- **TOG2** - This is a YES/NO field. If this is set to YES, PC-DMIS displays the comment in a full screen. Otherwise, the comment is not displayed in a full screen.
- **TOG3** - This is a YES/NO field. If this is set to YES, the comment uses a countdown timer, and PC-DMIS automatically continues execution when the timer reaches zero. At zero, PC-DMIS behaves as if you clicked the **OK** button and continues the execution of the measurement routine. Otherwise, the countdown timer is not used.
- **TOG4** - This is a YES/NO field. If this is set to YES, PC-DMIS displays the OVC comment during the execution of the measurement routine. Otherwise, the OVC comment is not displayed during the execution of the measurement routine.

How it Works

When PC-DMIS executes the measurement routine, a message box displays the relevant comments just below the **Execution** dialog box.



PC-DMIS Message Dialog Box

PC-DMIS doesn't allow you to edit this message. Click **OK** to continue execution of the measurement routine. Click **Cancel** to cancel the execution of the measurement routine.

Display Full Screen

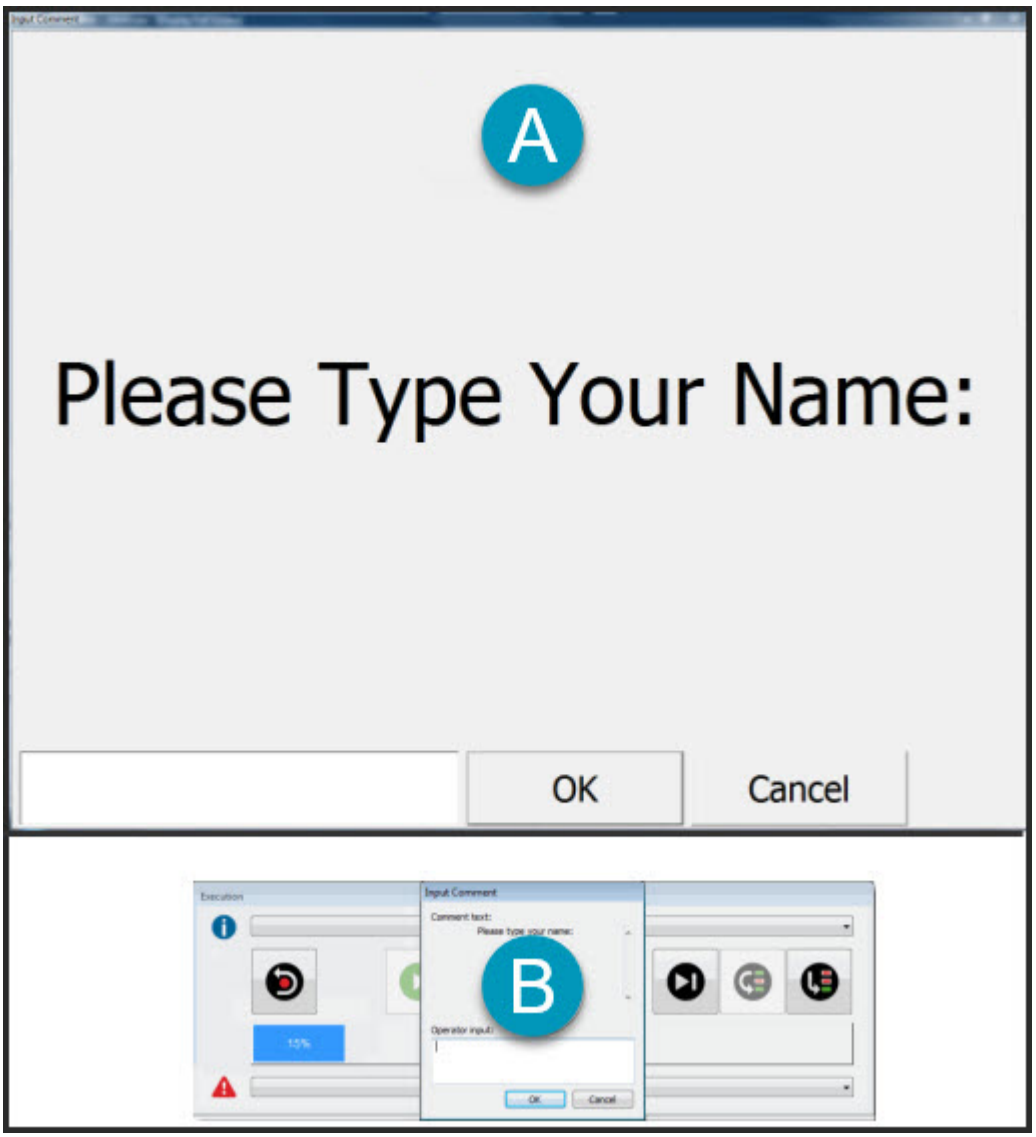
This check box determines whether or not your comment's text is displayed in an enlarged size that fills the entire screen once the command is executed. This functions with **Input**, **Yes / No**, and **Operator** comments. If you select other comment types, PC-DMIS disables this check box. Also, with this check box marked, you have the option to display certain media (images or movies) within the displayed comment. In that case, the **Import Media** and **Remove Media** buttons become enabled. For information on those buttons, see "Import Media" or "Remove Media".

> The Edit window command line for this option reads:
> `COMMENT/TOG1,TOG2,Full Screen=TOG3,`
> `comment text`

TOG1 = An INPUT, YESNO or OPERATOR comment.

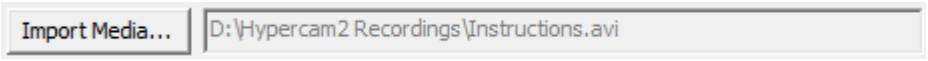
TOG2 = This YES/NO field allows you to specify whether or not the comment shows up in the inspection report.

TOG3 = This YES/NO field allows you to specify whether or not the comment fills the entire screen when displayed.



Example of a full screen input comment (A) and a normal sized comment (B).

Import Media



To import this button, you must first mark the **Display Full Screen** check box.

The **Import Media** button lets you browse to a valid media file and *import* that file into your measurement routine. PC-DMIS then displays it within a full-screen comment when the measurement routine executes that comment. When you click **OK**, the file is *imported into* the measurement routine and *becomes a component* of the measurement routine. This makes it so you don't need to remember to move image and video files when you move the measurement routine to another computer system or directory. However, it can also make your measurement routine size quite large, depending on the type and number of media files you choose to include.

The chosen file appears in the box to the right of the **Import Media** button.

Valid image file types include the following:

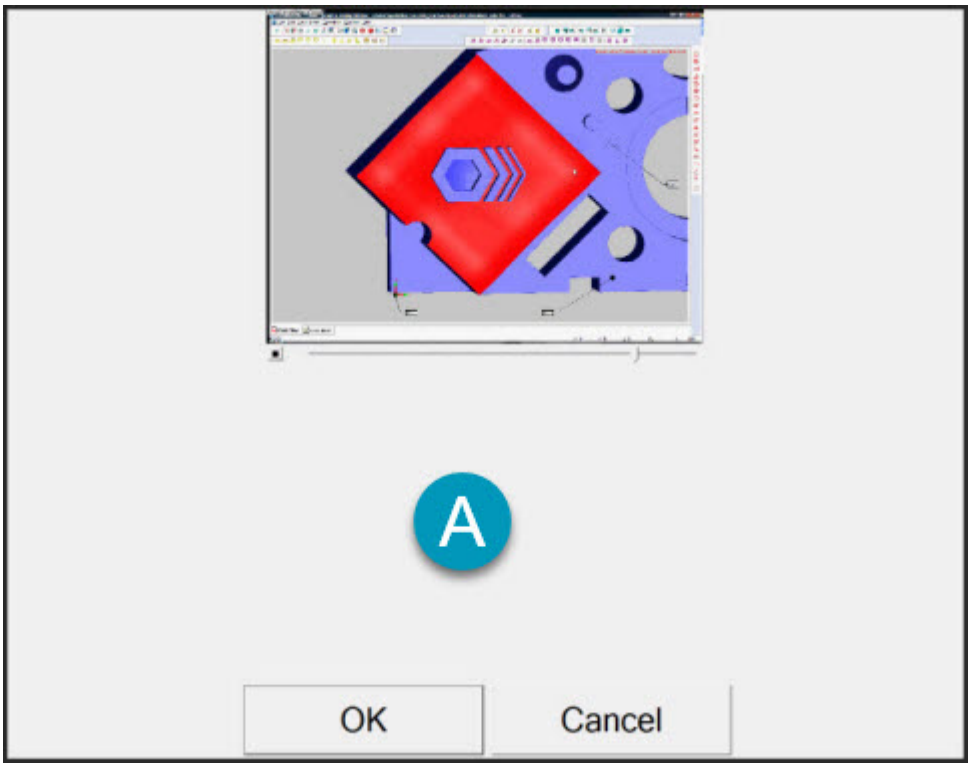
- .bmp
- .png
- .jpg

Valid movie file types include the following:

- .avi

About the Media Container

When PC-DMIS executes a comment containing a media file, it sizes the displayed media to fit within a portion of the full-screen comment called a "container". This means PC-DMIS sizes the image or movie to fit within the container.



This sized image shows a full-screen Operator comment. Notice that the media container in the top portion of the comment contains a movie sized to fit within that container.
(A) - Take four hits on the top of the part, as shown in the video above.

The horizontal slider below the container displays the current progression of a loaded movie file. You can drag the slider to jump to different spots along the movie's timeline.

This icon stops a movie file from playing.

This icon resumes playing a stopped movie file.

Supported AVI Codecs

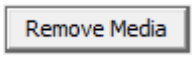
A video codec is the algorithm that compresses and decompresses a video file. If you try to use an .avi video file with an unsupported codec in an Operator comment, during execution, you may receive an "MCI Error" that says, "No Driver installed" or "The specified file cannot be played on the specified MCI device."

If you get any of these errors, check your video format and your codec.

You must use an .avi file that was created or compressed with the Microsoft Video 1 (MS-CRAM) codec.

- To check your video's codec, you can use the free and open source VLC media player from https://www.videolan.org/vlc/.
- To create a video with the Microsoft Video 1 codec, here are some possible options to consider:
  - Use the older free and unregistered HyperCam 2 software and record with the Microsoft Video 1 codec.
  - Use a newer recording application that supports recording or exporting in the Microsoft Video 1 codec. For example, with TechSmith Camtasia 2021, you can import a modern video format, such as .mp4 file, and then export that file as an .avi with the Microsoft Video 1 codec.

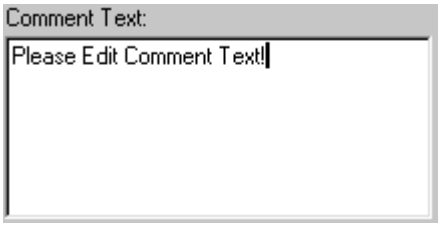
Remove Media



This becomes enabled if you first mark the **Display Full Screen** check box.

The **Remove Media** button removes the media from the comment. While this removes the media from the measurement routine, be aware that due to limitations with the Windows file system, you may still see a file size higher than it was originally before you imported the media. In these cases, you can always do a **File | Save As** and save the measurement routine to a different file name to get a measurement routine with its original file size.

Comment Text



The **Comment Text** box contains the text you wish to display in conjunction with the Comment Type options as shown here:

- Operator
- Report
- Input
- Document
- Yes / No
- Readouts

Inserting ASCII Characters

ASCII Character Buttons

These buttons on the **Comment** dialog box insert some commonly used ASCII characters into the **Comment Text** box.

Embedding Reports or Templates into a Measurement Routine

You can embed these reports into your measurement routine:

- Legacy report (HyperView report)
- Report template
- Custom report
- Label template

When PC-DMIS executes the embedded code, it launches the report or template. You can then print it, view it, and pass values to and from the embedded item.

Legacy reports are older reports that use the HyperView report editor. PC-DMIS versions 3.0, 3.2, 3.25, 3.5, or 3.7 used this editor. For information on these reports, see "Working with Legacy (HyperView) Reports".

To embed a report or template:

1. Make sure the Edit window is in Command Mode.
2. Select **Insert | Report Command**, and then choose one of these menu items.
   - Custom Report
   - Template Report
   - Legacy Report
   - Report Label

   A dialog box appears allowing you to select your report or template.
3. Select the report or template you want to embed in the Measurement Routine.
4. Click the **Open** button to insert the command.

PC-DMIS inserts one of the following commands into the Edit window at the cursor position:

- `REPORT/CUSTOM`
- `REPORT/TEMPLATE`
- `REPORT/LEGACY`
- `REPORT/LABEL`

The REPORT Command

The Report command object is identified with the `REPORT` command in the Edit window.

The REPORT Command Syntax

A `REPORT` command has the following syntax in the Edit window in Command Mode:

```
<ID>        =REPORT/<TOG1>, FILENAME= <PATHWAY>,
AUTOPRINT=<TOG2>, INLINE REPORT=<TOG3>,
Section=<NUM1>,REPORTMODE=<MODE>
            PARAM/=
            ENDREPORT/
```

<ID>

This is the ID for the REPORT command.

REPORT/<TOG1>

This changes the type of `REPORT` command. It can be CUSTOM, LEGACY, TEMPLATE, or LABEL.

FILNAME=<PATHWAY>

This determines the report or template to load. <PATHWAY> is the full file name path that points to the desired file. If you only use the name of the report or template, PC-DMIS looks in the default Reporting directory.

AUTOPRINT =<TOG2>

<TOG2> can switch between PRINT and PDF and NO. It only applies to `REPORT/LEGACY` commands.

Setting AUTOPRINT=PRINT prints the HyperView report to the default printer after it loads and executes. The report closes immediately after the print job has spooled and execution of the measurement routine resumes.

Setting AUTOPRINT=PDF prints the HyperView Report to a Portable Document Format (PDF) file. The following rules apply:

- The name of the generated PDF file has the same base name as the report or template file name appended by a numerical index and extension.
- The generated file is located in the same directory as the Report file.
- If a file exists with the same name as the generated file name, the index automatically increments itself until a unique file name is found.

Setting AUTOPRINT=NO does not print anything during execution of this command.

INLINE REPORT=<TOG3>

This is only used with Legacy (HyperView) Reports that have AUTOPRINT set to PDF. It determines if HyperView report information appears in line with the other generated data from the Report window in a PDF output.

<TOG3> switches between ON and OFF.

ON - When this command executes, and the **Output Configuration** dialog box generates a PDF automatically by the **Overwrite**, **Auto**, or **Append** options, the data from the specified legacy (HyperView) report appears in line with the other normal report output in the PDF file.

If the **Prompt** option is used, and this is set to ON, then multiple PDFs are generated, one for the HyperView report and additional PDFs as needed to cover the rest of the Report window's content.

OFF - Data from the HyperView report appears in its own PDF file based off the name of the HyperView report. It is not included in the PDF generated from the **Output Configuration** dialog box.

Be aware that this option overrides the AUTOPRINT option for HyperView commands.

For more information on legacy HyperView reports, see "Working with Legacy (HyperView) Reports".

Section=<NUM>
<NUM> determines the section into which the report or template is inserted. This item does NOT apply to label templates.

If you type -1 or a value greater than the number of sections in the default template, the report gets inserted at the end.

REPORTMODE=<MODE>
When a measurement routine executes and a default report template is specified, it starts to generate the report using the default report template. Once it reaches the REPORT command, the generation changes based on the specified report mode:

<MODE> - This can either be SWITCH or INSERT. This mode only applies to the REPORT/TEMPLATE commands.

SWITCH - This switches the report template to use the one in this command until another REPORT command is encountered or until the end of the measurement routine.

INSERT - This generates report data for previously-executed commands. It uses the template in this command. When this REPORT command finishes, PC-DMIS returns to using the default report template to generate the rest of the report data.

PARAM/
The PARAM/ option allows you to map PC-DMIS expressions to a report or a report template's variables or properties. This option does not apply to REPORT/LABEL commands.

The syntax of the PARAM/ option is:

```
PARAM/{object or variable name}={PC-DMIS expression}
```

If you have a Global Variable in a report named NumBoltHoles, the following PARAM/ option would map the PC-DMIS expression to NumBoltHoles:

```
PARAM/NumBoltHoles=360.0/angle
```

Changes to the variable in the editor can also change the PC-DMIS variable.

Using the following PARAM/ option, the variable NumBoltHoles is set equal to the variable numBolts. Any changes within the HyperView report to the HyperView variable NumBoltHoles are reflected in the PC-DMIS variable numBolts:

```
PARAM/NumBoltHoles=numBolts
```

After execution, any objects you changed by passing parameters into a Report template or a HyperView report revert to what they had been before execution if you redraw or modify the report in any way. This means if you want to retain the results of passing a parameter to your template or report, you need to first print the report before you make modifications. You can do this using the AUTOPRINT parameter or by printing directly from the Report window.

See "Using the PARAM command to Map PC-DMIS Expressions to Object Properties" for additional information on using parameters.

You can press F9 on a `REPORT/CUSTOM` command to open the custom report in the Custom Report Editor.

REPORT/TEMPLATE Report Generation

PC-DMIS generates the default report as usual until it reaches this command. When it executes this command, PC-DMIS utilizes the specified template and generates a separate report for all the measurement routine commands in a new section within the current report. After PC-DMIS generates this new report section, it switches back to the previous report and continues to generate commands in the original report from that point forward.

Using the PARAM command to Map PC-DMIS Expressions to Object Properties

PC-DMIS expressions can be mapped to property objects using the `PARAM/` command in the Edit window. The syntax for mapping a PC-DMIS expression to an object property is:

PARAM/{object code}.{property name}={PC-DMIS expression}

The following PARAM/ option changes the BorderStyle property of the object, Text1: `PARAM/Text1. BorderStyle =1`

Inserting External Objects

To add external objects to your report, select the **Insert | Report Command | External Object** menu item. For more information, see the "Adding External Elements" chapter.

Inserting a Print Command

When PC-DMIS executes a `PRINT/REPORT` command, it sends the measurement results up to that point in the measurement routine to the defined output destination (printer or file). After PC-DMIS executes the command, it clears the contents of the Report window, and only the remaining measurement results appear in the Report window.

However, you can always click the **View Report Mode** icon on the **Reporting** toolbar to view the full report.

To insert a `PRINT/REPORT` command into the Edit window, do the following:

1. If the Edit window isn't already visible, choose **View | Edit Window** to open the Edit window.
2. Choose **View | Command Mode** to place the Edit window into Command Mode.
3. Select the **Insert | Report Command | Print Command** menu option (or type **PRINT** at the desired location and press Tab). PC-DMIS inserts a `PRINT/REPORT` command with various options you can configure.
4. To configure the command, press F9 on the command and use the **Output Configuration** dialog box. For more information, see "Editing the PRINT/REPORT Command".

The `PRINT/REPORT` command controls the printing of reports from within the measurement routine. When PC-DMIS executes this command, a report is generated and sent to the specified output destination.

To control how PC-DMIS handles a report header *after* a `PRINT/REPORT` command, see "Header after Print Command" area in the "Edit Text Reporting" topic of the "Reporting Measurement Results" chapter.

Related Topics:

Using the Print Command in a Loop

Understanding the Syntax for the PRINT/REPORT Command

The syntax of the command is as follows:

```
PRINT/REPORT,EXEC MODE=END,$
   TO_FILE=OFF,TOG1,AUTO,OUTPUT_FORMAT/TOG2,$
   RESET_REPORT=YES,AUTO OPEN=OFF,$
   TO_PRINTER=OFF,COPIES=,$
   TO_DMIS_REPORT=OFF,FILE_OPTION=TOG3,FILENAME=,$
REPORT_THEORETICALS=ALL,REPORT_FEATURE_WITH_DIMENSIONS=YES,$
   TO_EXCEL=OFF,TOG4, AUTO OPEN=OFF,$
   PREVIOUS_RUNS=DELETE_INSTANCES
```

EXEC MODE = Because DMIS and PC-DMIS manage report data differently, this option controls how and when the software sends report data to DMIS and PC-DMIS output files. DMIS requires that the output filenames and other printing parameters be defined first *before* the routine is executed. PC-DMIS, on the other hand, waits until *after* the measurement routine is executed before deciding where to dump the report data. With this option, you can support both formats. It takes one of two values, either **START** or **END**.

START

This tells PC-DMIS to begin printing when it either encounters the end of the measurement routine or another `PRINT/REPORT` command. If you import a DMIS file, and it has a print report command, it uses **START** as its initial value.

END

This tells PC-DMIS to print everything already executed up to this `PRINT/REPORT` command. If you don't import a DMIS file with a print report command, but instead insert your own `PRINT/REPORT` command from within PC-DMIS, it has **END** as its initial value.

If your measurement routine contains more than one `PRINT/REPORT` command, be aware that PC-DMIS may not print some commands. For example, if your first `PRINT/REPORT` command uses **END** and your second print report command uses **START**, then PC-DMIS does not print everything in between.

Also if a `PRINT/REPORT` command uses **START** and then your very next print report command uses **END**, the second printing is empty, because the first print report command has cleared the report buffer.

TO_FILE= Use this option to indicate whether the printed report should be sent to a file (**ON**) or not (**OFF**).

TOG1 This value controls the action taken when sending the print output to an .rtf, .pdf, or .txt file. The options available are **APPEND / OVERWRITE / AUTO / PROMPT**. Depending on the option selected, different information may appear.

APPEND

Use this option to append report information to the specified filename. When you select the **APPEND** option, this keyword is followed by a filename field, for example, `APPEND=D:\REPORTS\FILE001.RTF` . Note that you must specify the complete path; otherwise, PC-DMIS assumes the same directory as the measurement routine. Also, if the file does not exist, PC-DMIS creates it when the `PRINT/REPORT` command is executed.

OVERWRITE

Use this option to overwrite report information to the specified filename. When you select the **OVERWRITE** option, this keyword is followed by a filename field, for example, `OVERWRITE=D:\REPORTS\FILE001.RTF` . If the file does not exist, PC-DMIS creates it when the `PRINT/REPORT` command is executed. If you don't provide a pathway, it defaults to the measurement routine's folder. If there is no path in this field, you can set this to a variable name (see the example below).

AUTO

Use this option to have PC-DMIS generate the report filename automatically. When you select the **AUTO** option, this keyword is followed by a numeric field, for example, `AUTO=10` . PC-DMIS uses the name of the measurement routine to name the generated file, and appends the numerical index value to the end of it with the .rtf extension. Also, PC-DMIS sends the generated file to the same directory as the measurement routine. If a file exists with the same name as the

generated filename, **AUTO** increments the index until a unique filename is found.

PROMPT

Use this option to cause a **Save As** dialog box to appear that you can use to type a filename and select a location to save the report information.

OUTPUT_FORMAT - Use this option to determine whether PC-DMIS sends output to a text file (.txt), Rich Text Format (.rtf) or Portable Document Format (.pdf) file.

TOG2= You can set TOG2 to **RTF**, **PDF**, **TXT**, or **PDF3D**. For information on the toggle options, refer to the Output Configuration topic.

RESET_REPORT - Use this option to determine if PC-DMIS resets the report after each `PRINT/REPORT` command.

YES

This option clears the report buffer and prints only the items executed after the current `PRINT/REPORT` command until it encounters the end of the measurement routine or another `PRINT/REPORT` command.

NO

If the measurement routine hits a `PRINT/REPORT` command, PC-DMIS prints everything in the buffer from:

- The beginning of the measurement routine up to the current `PRINT/REPORT` command if there were no other `PRINT/REPORT` commands before the current `PRINT/REPORT` command, or
- From the last `PRINT/REPORT` command that was set to **YES** up to the current `PRINT/REPORT` command, or
- The beginning of the measurement routine through all `PRINT/REPORT` commands as long as they were all set to **NO** up to the current `PRINT/REPORT` command.

AUTO OPEN= Use this option to determine whether PC-DMIS opens the generated .rtf, .pdf, or .txt file after the `PRINT/REPORT` command executes. This can be **ON** or **OFF**.

TO_PRINTER= Use this option to indicate whether to send the report to the printer (**ON**) or not (**OFF**).

COPIES= This value determines the number of report copies to send to the printer.

TO_DMIS_REPORT= Use this option to determine whether or not PC-DMIS sends the report to a DMIS Output file (.dmo) specified in **FILENAME**. This can be **ON** or **OFF**.

FILENAME= Use this option to specify a DMIS output directory location and filename. If **TO_DMIS_REPORT** is set to **ON** then PC-DMIS stores the report data in the file specified. For example, if `FILENAME=D:\PCDMISREPORTS\MYDMIS.DMO`, PC-DMIS causes the report data to be saved in the MYDMIS.dmo file in a PCDMISREPORTS folder on drive D. If you don't provide a pathway, it defaults to the measurement routine's folder. If there is no path, you can set this to a variable name.

FILE_OPTION=TOG3 You can set TOG3 to the following file options for your DMIS file output:

APPEND

This appends the report data to the end of the DMIS file specified in **FILENAME**.

OVERWRITE

This overwrites the DMIS file specified in **FILENAME** with the latest report data.

INDEX

This appends to the DMIS filename specified in **FILENAME** a numerical value that increments on subsequent measurement routine executions. For example, if `FILENAME=MYDMIS.DMO`, on the next execution the filename increments by one, becoming mydmis001.dmo, mydmis002.dmo, mydmis003.dmo, and so on, similar to the **AUTO=** option.

REPORT_THEORETICALS= This tells how PC-DMIS reports theoretical values in the DMIS output file.

ALL

PC-DMIS outputs all theoretical values along with the measured values to the output DMIS file.

NONE

No theoretical values are included in the report.

IMPORT_SETTINGS

Only those theoretical values explicitly output by the original DMIS file are output in the report.

REPORT_FEATURE_WITH_DIMENSIONS= This determines whether or not PC-DMIS keeps the measured features and associated tolerances together in the output file. It takes either **YES** or **NO**.

YES

In the DMIS report file, PC-DMIS writes the measurement results immediately before their associated tolerance results for each dimension associated to the feature itself. If a feature is not associated with any tolerance, PC-DMIS does not generate any output.

NO

In the DMIS report file, PC-DMIS writes the measurement results exactly when the feature is measured, but not later when PC-DMIS executes the associated dimensions.

TO_EXCEL= This determines whether or not PC-DMIS sends the output to a Microsoft Excel file. It takes either **ON** or **OFF**.

ON

An Excel file is generated for all the items in the measurement routine to this point.

OFF

No Excel file is generated by this print command.

To modify Excel output options, press F9 on this command and modify the options in the **Excel** tab.

Additional Notes:

- If this is set to **ON**, but the measurement routine's **Output Configuration** dialog box has the **Excel output** check box cleared, PC-DMIS generates the Excel for this specific command only.
- If this is set to **OFF**, but the measurement routine's **Output Configuration** dialog box has the **Excel output** check box marked, an Excel file is generated at the very end of the execution.
- If this is set to **ON**, and the measurement routine's **Output Configuration** dialog box has the **Excel output** check box also marked, you may end up with two generated output files, one for each. In this case, dimensions that follow a `PRINT/REPORT` command are generated in a different file.
- Looping is supported with the Excel output. If you run the `PRINT/REPORT` command through a loop and the Excel output is set to append, it generates the content as additional worksheets inside the Excel file.

- To change the number of decimal places for the Excel output file output, you can insert a `DISPLAYPRECISION` command just before the `PRINT/REPORT` command.

TOG4= This value controls the action taken when sending the print output to an Excel file (.xls, .xlsx, csv). For information on the toggle options, refer to the similar TOG1 item above.

FILENAME= Use this option to specify the output directory location and filename for the Excel file. If **TO_EXCEL_OUTPUT** is set to **ON**, then PC-DMIS stores the report data in the file specified. For example, if `FILENAME=D:\PCDMISREPORTS\MYEXCEL.XLS` , PC-DMIS causes the report data to be saved in the MYEXCEL.xls file in a PCDMISREPORTS folder on drive D. If you don't provide a pathway, it defaults to the measurement routine's folder. If there is no path in this field, you can set this to a variable name (see the example below).

AUTO OPEN= Use this option to determine whether PC-DMIS opens the generated Excel file (.xls, .xlsx, .csv) after the `PRINT/REPORT` command executes. This can be **ON** or **OFF**.

PREVIOUS_RUNS= Use this with a `PRINT/REPORT` command from within a loop. When a feature is measured more than once in a single execution, previous instances of that feature's measured data are saved. You can choose to delete (**DELETE_INSTANCES**) or keep (**KEEP_INSTANCES**) these instances.

Variable for FILENAME

In this example, the variable V1 takes the value of the Input comment stored in C1.INPUT. Then the FILENAME statement for the Excel output uses the value of variable V1 for the Excel file's folder and filename:

```
C1=COMMENT/INPUT,YES,FULL SCREEN=NO,

Type your Excel report pathway and file name:

ASSIGN/V1=C1.INPUT

PRINT/REPORT,EXEC MODE=END,$

TO_FILE=OFF,AUTO=1,AUTO OPEN=OFF,$

TO_PRINTER=OFF,COPIES=1,$

TO_DMIS_REPORT=OFF,FILE_OPTION=INDEX,FILENAME=,$

REPORT_THEORETICALS=NONE,REPORT_FEATURE_WITH_DIMENSIONS=NO,
$

TO_EXCEL=ON,OVERWRITE,FILENAME=V1,AUTO OPEN=ON,$

PREVIOUS_RUNS=DELETE_INSTANCES
```

For example, if the operator typed D:\Temp\MyReport.xls in the Input comment, PC-DMIS would save the Excel data to that directory and report name.

Editing the PRINT/REPORT Command

You can edit the `PRINT/REPORT` command's options in its **Output Configuration** dialog box.

1. Place the cursor over the `PRINT/REPORT` command line.
2. Press F9.
3. Make any changes.
4. Click **OK**.

PC-DMIS updates the command to match your change. You may not always see a visual indication in the command text itself of the change.

Understanding the Different Output Configuration Dialog Boxes

Note that PC-DMIS uses the **Output Configuration** dialog box in these locations:

From File | Printing | Report Window Print Setup

The main **Output Configuration** dialog box from that menu does *not* insert or modify the `PRINT/REPORT` command. Instead, it controls the overall printing defaults for the report. Unlike the `PRINT/REPORT` command, the main dialog box only affects printing when the measurement routine has completed execution. For information on the options in this dialog box, see "Printing the Report Window" in the "Using Basic File Options" chapter.

For Each `PRINT/REPORT` Command

When you press F9 on one of these commands, its specific **Output Configuration** dialog box opens.

- You can use the `PRINT/REPORT` command more than once in the measurement routine.
- Each `PRINT/REPORT` commands can have different outputs.
- Each `PRINT/REPORT` command is a unique instance of the information contained in the main **Output Configuration** dialog box. Therefore, options that are selected from the main dialog box are independent of any `PRINT/REPORT` command.

Inserting a Form Feed Command

PC-DMIS lets you insert a `FORMFEED` command into the Edit window. This command causes the printed page of a report to be ejected from the printer when the `FORMFEED` command has been marked and executed. The `FORMFEED` command has no effect when printing to a file.

To insert a `FORMFEED` command in to the Edit window, follow these steps:

1. Open the Edit window (**View | Edit Window**).
2. Click the **Command Mode** icon from the **Edit window** toolbar, placing PC-DMIS into Command Mode.
3. Select the **Insert | Report Command | Form Feed** menu option (or type `FORMFEED` at the desired location and press Tab). PC-DMIS displays the `FORMFEED` command.

Working with Viewsets

You can use `VIEWSET` commands to create and save different Graphic Display window views (termed "viewsets"). You can later recall and display saved viewsets in your Graphic Display window and CAD reports with `RECALL/VIEWSET` commands.

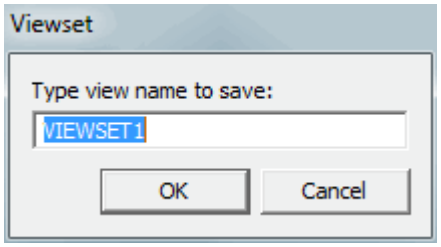
Viewsets store the following:

- The CAD model's orientation
- The CAD model's shading
- ID label visibility
- ID label position

The **Create Viewset** menu option allows you to create an unlimited number of viewsets in your measurement routine. You can recall multiple viewsets as often as desired.

To Create a Viewset:

1. Set up the view as desired. To do this, select **Edit | Graphic Display Window | View Setup** and use the **View Setup** dialog box to modify the zoom and rotation of the part in the Graphic Display window. For information on how to do this, see "Setting Up the Screen View" in "Editing the CAD Display".
2. Select **Insert | Report Command | Create Viewset** or from the **Graphic Modes** toolbar, choose **Create Views** ( ). A small **Viewset** dialog box opens with the prompt "Type view name to save." This dialog box contains a default name starting with VIEWSET1, and then it increments the number for subsequent viewsets (VIEWSET2, VIEWSET3, and so on):



Viewset dialog box with a default name.

3. In the box, type the desired name (19-character limit) for the view.
4. Click the **OK** button or press Enter. PC-DMIS sets the current viewset equal to the name you chose by inserting a `<NAME>=VIEWSET` command, where <NAME> is the defined viewset name.

To Recall a Viewset:

To recall the created viewset, do one of the following:

- From the **Settings** toolbar, select the **Views** list.
- From the PC-DMIS portable **QuickMeasure** toolbar, select **Recall Views** ( ).
- From the menu bar, choose **Insert | Report Command | Recall Viewset**.

Select the viewset that you want to recall. PC-DMIS inserts a `RECALL/VIEWSET,<NAME>` command into the Edit window.

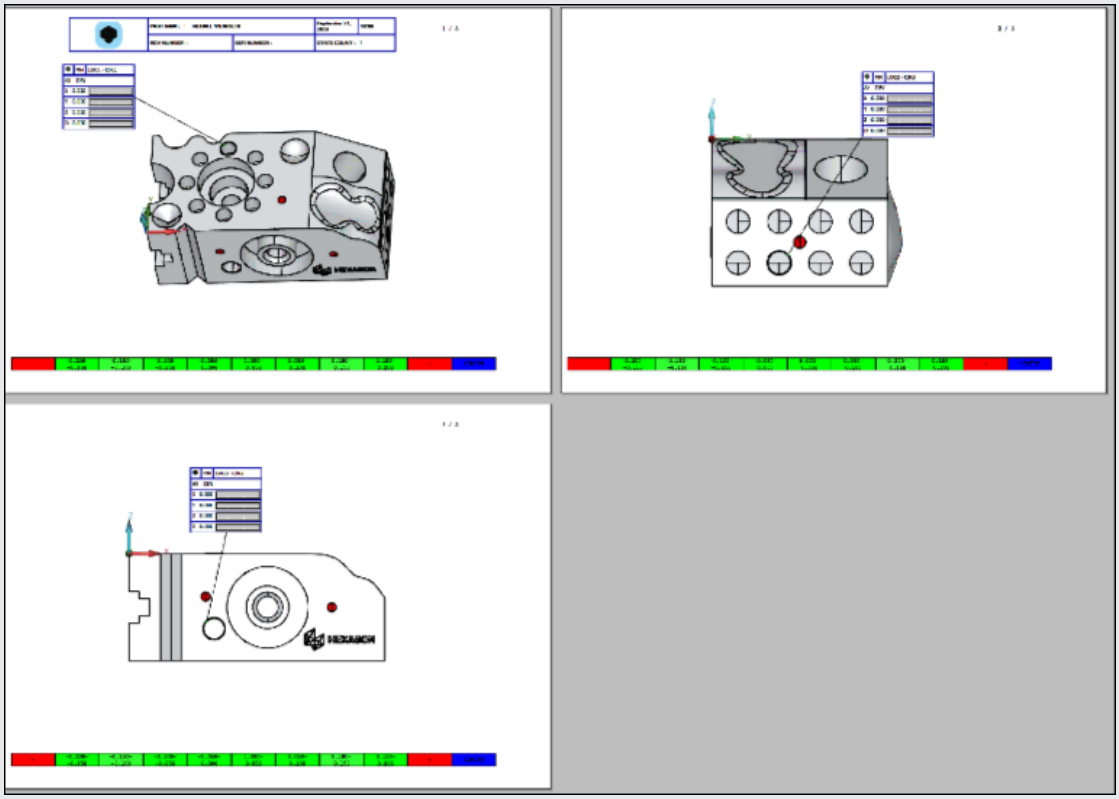
When your cursor is on or under the `RECALL/VIEWSET,<NAME>` command in the Edit window, PC-DMIS displays the created viewset in the Graphic Display window, where <NAME> is the defined viewset name. If you mark and execute this command, PC-DMIS displays the saved view in the Graphic Display window during measurement routine execution as well.

Additionally, recalled viewsets appear in your final report if the Report Window uses a template or custom report that displays a CADReportObject.

You can make viewsets in reports appear on a new page for each `RECALL/VIEWSET` command if your measurement routine has a dimension or colormap command after the `RECALL/VIEWSET` command. This approach orients the view in the report and forces a new page for the dimensions to show.

This example in the Edit window shows three `RECALL/VIEWSET` commands, each with a dimension command after it. These dimension commands force each viewset to appear on its own page inside the report:

```
             RECALL/VIEWSET,VIEWSET1
DIM LOC1= LOCATION OF CIRCLE CIR1  UNITS=MM ,$
GRAPH=OFF  TEXT=OFF  MULT=10.00  OUTPUT=BOTH  HALF ANGLE=NO
AX     NOMINAL       +TOL       -TOL       MEAS        DEV      OUTTOL
X       40.000       0.050      0.050     40.000      0.000      0.000 ---#----
Y       50.500       0.050      0.050     50.500      0.000      0.000 ---#----
Z        0.000       0.050      0.050      0.000      0.000      0.000 ---#----
D        8.200       0.050      0.050      8.200      0.000      0.000 ---#----
END OF DIMENSION LOC1
             RECALL/VIEWSET,VIEWSET2
DIM LOC2= LOCATION OF CIRCLE CIR3  UNITS=MM ,$
GRAPH=OFF  TEXT=OFF  MULT=10.00  OUTPUT=BOTH  HALF ANGLE=NO
AX     NOMINAL       +TOL       -TOL       MEAS        DEV      OUTTOL
X      120.000       0.050      0.050    120.000      0.000      0.000 ---#----
Y       23.000       0.050      0.050     23.000      0.000      0.000 ---#----
Z      -42.000       0.050      0.050    -42.000      0.000      0.000 ---#----
D        8.000       0.050      0.050      8.000      0.000      0.000 ---#----
END OF DIMENSION LOC2
             RECALL/VIEWSET,VIEWSET3
DIM LOC3= LOCATION OF CIRCLE CIR2  UNITS=MM ,$
GRAPH=OFF  TEXT=OFF  MULT=10.00  OUTPUT=BOTH  HALF ANGLE=NO
AX     NOMINAL       +TOL       -TOL       MEAS        DEV      OUTTOL
X       40.000       0.050      0.050     40.000      0.000      0.000 ---#----
Y        0.000       0.050      0.050      0.000      0.000      0.000 ---#----
Z      -35.000       0.050      0.050    -35.000      0.000      0.000 ---#----
D       10.000       0.050      0.050     10.000      0.000      0.000 ---#----
END OF DIMENSION LOC3
```



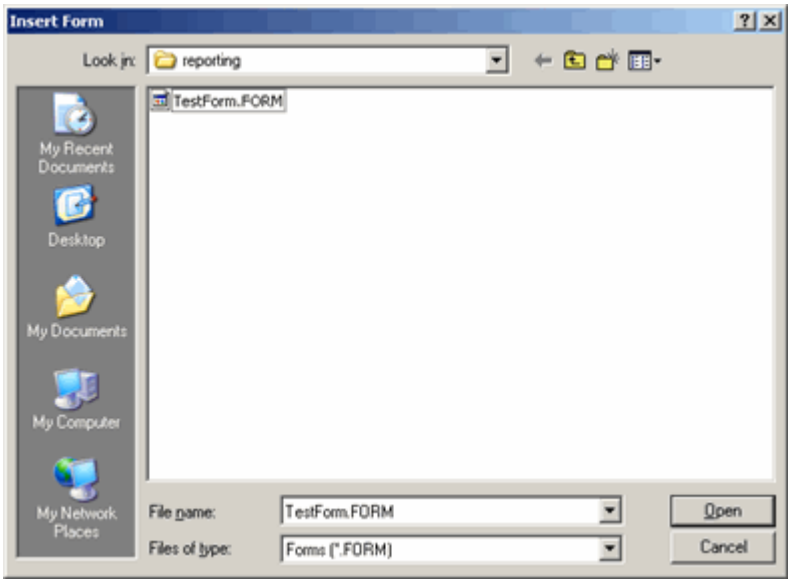
To Update a Viewset:

You can also modify an existing viewset. From the Edit window, select the `RECALL/VIEWSET,<NAME>` , modify the Graphic Display window as desired for that view, and then select the **Insert | Report Command | Save Viewset** menu option. Or from the **Graphic Modes** toolbar, select the **Save Viewset** icon ( ).

If you just want to change and then save the current viewset without creating a new viewset, select the **Save Viewset** menu option instead.

Inserting a FORM Command

The **Insert | Report Command | Form** menu item lets you insert a predefined .FORM file into your measurement routine. When you select this menu item, an **Insert Form** dialog box appears.



Insert Form Dialog Box

Navigate to the .FORM file, select it, and click **Open**. PC-DMIS inserts a FORM/FILENAME command similar to the following into the Edit window:

```
CS1=FORM/FILENAME=C:\PCDMIS40RELEASE\REPORTING\TESTFORM.FORM
    PARAM/=
    ENDFORM/
```

FILENAME = This defines the location of the .FORM file to load.

PARAM/ - This lets you send information to the form. For example, using a PARAM statement of `PARAM/Text1. Text =C1. INPUT` sends the value of the comment `C1. INPUT` to the control.

Alternately, you can type FORM and press TAB to insert the FORM/FILENAME command into the measurement routine, and then manually type the path to load the desired form.

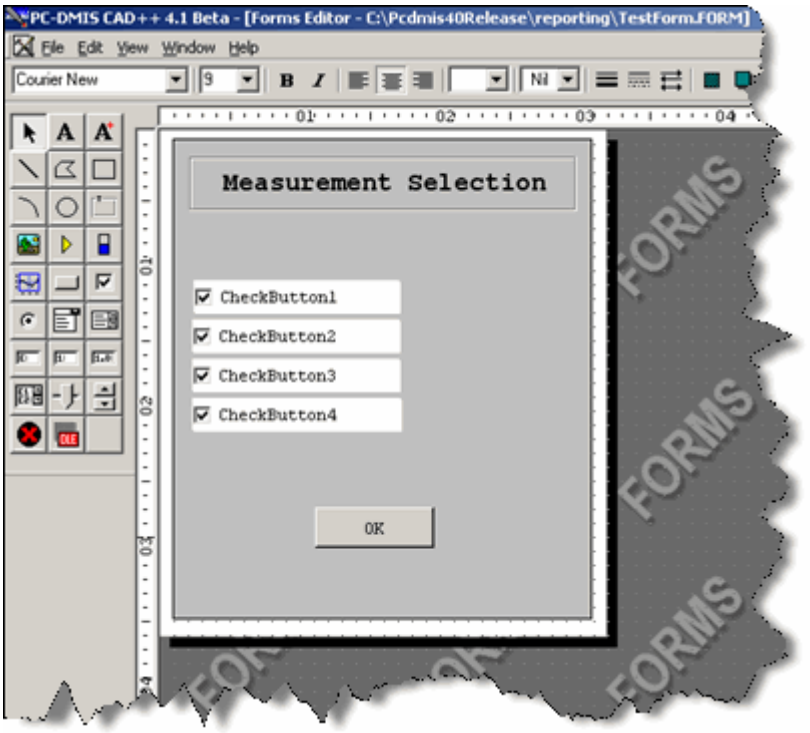
When PC-DMIS executes this command, it launches the defined form, and then passes any defined parameters to the controls in the form.

Communicating between the FORM and the Measurement Routine

You can pass values to and from a form by using a combination of ASSIGN and PARAM statements. This lets you establish useful two-way communications between the form and the measurement routine.

ASSIGN Statements - You can initialize values for a form's controls through ASSIGN statements. These statements also create variables that can later receive updated values from the form when the form closes.

For example, suppose you have a form that has four unnamed check boxes, like this:



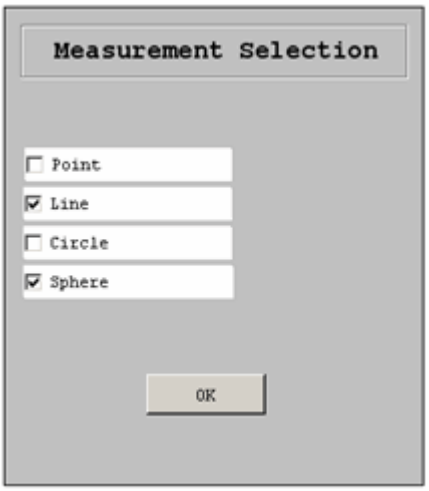
However, you want the measurement routine to give them names and values dynamically. In the Edit window, you can use the ASSIGN statements to name and initialize the check boxes as follows:

```
ASSIGN/CHECK1VALUE=0
ASSIGN/CHECK1TEXT="Point"
ASSIGN/CHECK2VALUE=1
ASSIGN/CHECK2TEXT="Line"
ASSIGN/CHECK3VALUE=0
ASSIGN/CHECK3TEXT="Circle"
ASSIGN/CHECK4VALUE=1
ASSIGN/CHECK4TEXT="Sphere"
```

PARAM Statements - Now that you have several variables, you can use the PARAM statements to establish the connection between the variables and the form itself, like this:

```
CS1=FORM/FILENAME= C:
\PCDMIS40RELEASE\REPORTING\TESTFORM.FORM
   PARAM/CHECKBUTTON1.VALUE=CHECK1VALUE
   PARAM/CHECKBUTTON1.TEXT=CHECK1TEXT
   PARAM/CHECKBUTTON2.VALUE=CHECK2VALUE
   PARAM/CHECKBUTTON2.TEXT=CHECK2TEXT
   PARAM/CHECKBUTTON3.VALUE=CHECK3VALUE
   PARAM/CHECKBUTTON3.TEXT=CHECK3TEXT
   PARAM/CHECKBUTTON4.VALUE=CHECK4VALUE
   PARAM/CHECKBUTTON4.TEXT=CHECK4TEXT
   PARAM/=
   ENDFORM/
```

When the form loads, CHECKBUTTON1 is unchecked, and its text label displays "Point"; CHECKBUTTON2 is checked, and its text label is "Line"; CHECKBUTTON3 is unchecked, and its text label is "Circle"; and CHECKBUTTON4 is checked, and its text label is "Sphere", like this:



With the form loaded, you can select or clear the check boxes to something different. When you close the form, the variables that initially set the state of the checked state (CHECK1VALUE, CHECK2VALUE, CHECK3VALUE, and CHECK4VALUE) now hold the current values of the check boxes.

Inserting Screen Captures

This topic details the old way to add screen captures. It remains for backwards compatibility. Try out the new **Snapshot** menu option instead; it has more functionality. For information, see "Inserting Snapshots" in this chapter.

You can use the **Insert | Report Command | Screen Capture** menu item to display screen captures of the part in your report. This menu option places a `DISPLAY/METAFILE` command in the Edit window. When this command is marked and executed, PC-DMIS automatically refreshes the screen capture and displays it in the report.

Command line in the Edit window:

```
DISPLAY/METAFILE,"___",TOG1,TOG2
```

___ = This field will allow you to enter a description of the captured screen. The maximum text is 255 characters.

TOG1 = This field determines the size of the screen capture. You can toggle between the following: 25%, 50%, 75%, 100%, TO FIT.

TOG2 = This field determines the quality of the screen capture. You can toggle between the following: **HIGH**, **GOOD**, **LOW**

For this to appear in the Report window, right-click on some text within the Report window, select **Edit**, and when the **Report** dialog box appears, select the **Show Screen Capture** check box.

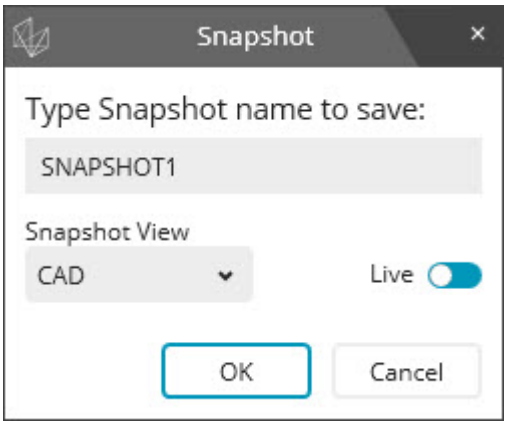
This command is similar to the `ANALYSISVIEW` command and the `DISPLAY/METAFILE` command that are created in the Analysis Window. For more information on the Analysis Window, see "Analysis area" in this chapter.

A **True Type Font** for text in the Graphic Display window gives the best results when capturing the image.

Inserting Snapshots

The `SNAPSHOT` command displays screen captures of the part in your report.

To access the **Snapshot** dialog box, select **Insert | Report Command | Snapshot** or from the **Graphic Modes** toolbar, click the **Insert Snapshot** button ( ).



Snapshot Dialog Box

Type Snapshot name to save - This box lets you type the snapshot name.

Snapshot View - This list lets you select **CAD** or **OVC** for the snapshot view.

Live - This button toggles Live Mode On and Off.

- When Live Mode is On, the software captures snapshots of the measurements from a continuous data stream. This means that PC-DMIS displays the probe data “live” (real-time) in the Graphic Display and Probe Readouts window.
- When Live Mode is Off, PC-DMIS takes a measurement as a single snapshot. PC-DMIS does not update the Graphic Display and the Probe Readout windows.

OK - This button inserts a `SNAPSHOT` command with that name in the Edit window. When this command is marked and executed, PC-DMIS automatically refreshes the screen capture and displays it in the report.

In the Edit window in Command Mode, you can also manually type `SNAPSHOT` and press Tab. When you press Tab, PC-DMIS stores the properties for the state of the Graphic Display window in the command. During execution, PC-DMIS inserts the capture into the report the moment the software executes the command.

Command line in the Edit window:
```
TOG1 =SNAPSHOT/
```

TOG1 = This field holds the name of the snapshot.

Differences from the DISPLAY/METAFILE Command

The `SNAPSHOT` command provides better screen captures than the `DISPLAY/METAFILE` command. While the snapshot functionality is similar to `DISPLAY/METAFILE`, it has these differences:

- A `SNAPSHOT` command updates the various analysis-related items you see on the Graphic Display window in the screen capture. These analysis items include graphical analysis, pointcloud color maps, dimension info labels, scan arrows with deviation colors, and so on. `DISPLAY/METAFILE` results in a much more static image.
- A `SNAPSHOT` command saves your screen orientation and zoom level.
- A `SNAPSHOT` command works with multiple views.

Default Snapshot Names

If you don't type a snapshot name, the default names follow this pattern SNAPSHOT1, SNAPSHOT2, SNAPSHOT3... and so on.

If you first select a CAD model view from the list on the **CAD Setup** toolbar and then access the **Snapshot** dialog box, the default pattern follows the name of the current model view.

Suppose you have a CAD model view with the name of **Right** to show the right side of your part. If you select **Right** from the list and then access the **Snapshot** dialog box, PC-DMIS shows the name of the snapshot as "Right1". If you do it again, PC-DMIS names the next snapshot as "Right2", and so on.

Inserting a Snapshot Command

1. In the Graphic Display window, pan, zoom, and rotate your part to the way you want it.

2. From the **Graphic Modes** toolbar, click **Insert Snapshot** ( ) to show a **Snapshot** dialog box.

3. In the **Snapshot** dialog box, type a name for the screen capture, select a view, and set the Live Mode.

4. Click **OK** to insert the `SNAPSHOT` command at the selected spot in the Edit window.

5. If PC-DMIS cannot insert at the current position, it shows a message box that asks how you want to handle the insertion. You can choose one of these three items on the message box:

    - **Next Position** - This inserts the command at the next valid position.
    - **End of Program** - This inserts the command at the end of the measurement routine.
    - **Don't Insert** - This does not insert the command.

Viewing the Snapshot in the Graphic Display Window

1. Select the `SNAPSHOT` command in the Edit window.

2. Press Ctrl + E to execute only that command. This shows the snapshot in the Graphic Display window.

Updating a Snapshot Command from the Graphic Display Window

1. From the Edit window, select the `SNAPSHOT` command you want to update.

2. Press F9 to open its **Snapshot** dialog box. The Graphic Display window loads the view stored in the snapshot.

3. In the Graphic Display window, pan, zoom, and rotate your part to the way you want it for the updated snapshot.

4. If you like, modify the name.

5. Click **OK** to update that snapshot with the new Graphic Display window properties.

Updating a Snapshot Command from Your Report

1. Execute the routine and then select **View | Report Window** to access the report in the Report window.

2. From the Report window, double-click on the snapshot image to convert it into a CADReportObject (CRO).

3. Pan, zoom, and rotate on the part model in the CRO to modify it. For information on working with the CRO, see "CadReportObject" in the "Reporting Measurement Results" chapter.

4. Double-click outside of the CRO to accept your CRO changes to the report. PC-DMIS applies these same CRO changes to your `SNAPSHOT` command in the Edit window. Subsequent executions of that command use the updated `SNAPSHOT` command.

Adding Snapshots into Custom Reports

1. In the Report window, edit a custom report, or create a new custom Report. For information on custom reports, see "Creating Custom Reports" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

2. From the Edit window in Summary Mode, select the snapshot command you want in your report.

3. Drag and drop it into the custom report to see the screen capture.

Deleting Referenced Commands

Any command in the Edit window that creates a visible element in the Graphic Display window (such as a Dimension Info box) is considered a referenced command by the snapshot command. The snapshot command requires the referenced command to create its screen capture. Snapshot commands cannot reference commands that don't exist. Therefore, if you delete any of these referenced commands, PC-DMIS asks if you want to also delete the associated snapshot commands.

Using File Input / Output

Using File Input / Output: Introduction

This chapter explains how to input and output information to and from your measurement routines. The available menu options allow for opening files in read or write mode. Data can then be read from or written to these files. The file I/O commands allow data to be read in from external files that are to be used in a measurement routine . Also, measurement and tolerancing information can be written back out to files using these commands. You can also perform other file operations by using these commands.

This chapter details these file I/O operations and includes functional examples for each of the various operations. These examples use items discussed in the "Branching by Using Flow Control" chapter and the "Using Expressions and Variables" chapter.

The main topics described in this topic include:

- Understanding Basic File I/O Concepts
- Using the File I/O Dialog Box
- Opening a File for Reading or Writing
- Closing an Opened File after Reading or Writing
- Reading a Character from a File
- Reading a Line from a File
- Reading a Block of Text from a File
- Reading Text up to a Delimiter
- Writing a Character to a File
- Writing a Line to a File
- Writing a Block of Text to a File
- Positioning a File Pointer at the Beginning of a File
- Saving a File Pointer's Current Position
- Recalling a Saved File Pointer's Position
- Copying a File

- Moving a File
- Deleting a File
- Checking for a File's Existence
- Displaying a File Dialog Box
- Checking for the End of a File or the End of a Line

Command Mode Commands after Comments

Because many of the code samples in this chapter use typed `COMMENT` commands, consider the following:

After you insert a PC-DMIS comment, to type additional PC-DMIS commands in Command Mode, you must first press Enter *twice* after the `COMMENT` command. This tells PC-DMIS that you no longer want to add text to the comment but are ready to add a new command.

Understanding Basic File I/O Concepts

Checking for File Existence:
For all the file I/O operations, you probably want to first check for the file's existence. This should probably be put in an IF / THEN loop so that if the check fails, you can notify the user. When you write to a file, you must first create the file inside the windows environment.

See "Checking for a File's Existence".

Opening and Closing Files:
For operations that read from or write to files, you need to first open them to your system's processes. You do this by assigning the file to a variable called a file pointer. When you open a file, you can specify whether the file is opened for reading, writing (overwriting), or appending. Once opened, you can then read from or write to the file. When you're finished working with a file, you should close the file pointer; this closes the file and allows it to be accessed by other system processes. You cannot open files that are already opened by another process.

See "Opening a File for Reading or Writing" and "Closing an Opened File after Reading or Writing".

File Pointers and Positions:
File pointers are variables that point to a file. They store an opened file's name and location and are then used to read from or write to that file. Once a file is opened and set

to a file pointer, the pointer behaves like a cursor acts in a word processor. They indicate where you are currently reading from or writing to within the file.

- If you're appending to a file, the file pointer is usually at the end of the file.
- If you're reading a file or overwriting a file, the file pointer should usually start at the beginning of a file.

Using Delimiters When Writing or Reading

When writing data, consider using delimiters to separate pieces of data. This makes it easier to read the data back into a measurement routine. A delimiter can be any character or string of characters. For example, suppose you have a point, named PNT1 with the X,Y, and Z measured values of 2.5,4.3,6.1. You can easily write these values separated by a comma delimiter into a data file with code similar to the following:

```
FILE/WRITELINE,FPTR,PNT1.X + "," + PNT1.Y + "," + PNT1.Z
```

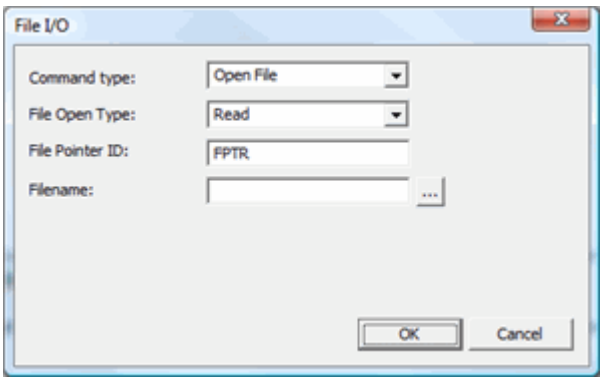
When reading data, you can separate the incoming data based on a specified delimiter and place the data into variables for later manipulation. For example, suppose you want to read in the same X, Y, and Z values listed above. The values should be in a single line of text like this: 2.5,4.3,6.1. You can separate the text at the comma and place those values into corresponding variables using a line of code similar to the following:

```
V1=FILE/READLINE,FPTR,{ValX}+","+{ValY}+","+{ValZ}
```

You can then use `ValX` , `ValY` , and `ValZ` as normal variables in your measurement routine. Resulting in: ValX = 2.5, ValY = 4.3, and ValZ = 6.1.

Using the File I/O Dialog Box

All file I/O commands are initially inserted into the measurement routine by selecting the appropriate file I/O menu option (select **Insert | File I/O Command** from the menu). Once a command exists in the Edit window, you can then press F9 on the command to access its associated **File I/O** dialog box.



File I/O Dialog Box

This dialog box provides a visual way to edit the *current* file I/O command. Alternately, you can modify a command inside the Edit window by using the techniques discussed in the "Using the Edit Window" chapter.

You should not use this dialog box to insert *new* file I/O commands. That must be done by selecting the appropriate menu option or by typing the commands directly into the Edit window.

Opening a File for Reading or Writing

The **Insert | File I/O Command | File Open** menu option allows you to put a command in the Edit window that opens a file from your computer during execution of the measurement routine.

Files can be opened simply to view information or to add and save information.

The syntax of this command in the Edit window is:

```
<filepointername> =File/Open,<filename>,<openmode>
```

Descriptions for some of the components of this command are:

<filepointername> - This is the user chosen ID of the filepointer that is used to access the opened file. This ID is used to refer to the open file in other file I / O commands.

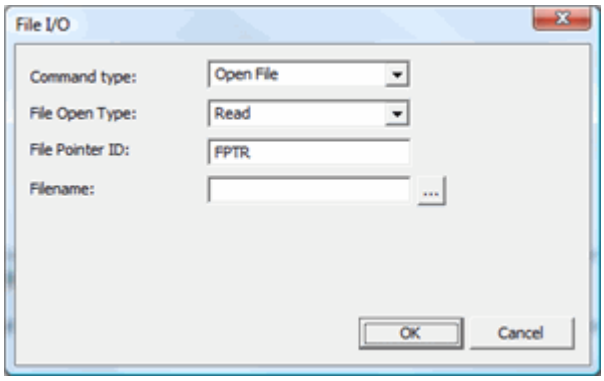
<filename> - This is the filename of the file to open.

<openmode> - This is the mode the file should be opened in. Files can be opened in the following modes: Read, Write, or Append.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.

2. Place your cursor on the File Open command.
3. Press F9.



Sample Code for File Open

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

This code opens a file named TEST.TXT for reading, writing, and appending. It stores the file name to a file pointer named `FPTR` .

```
FPTR=FILE/OPEN,C:\PCDMISW\TEST.TXT,READ
FPTR=FILE/OPEN,C:\PCDMISW\TEST.TXT,WRITE
FPTR=FILE/OPEN,C:\PCDMISW\TEST.TXT,APPEND
```

Be aware that you can use an input comment to take the full pathway as an input and use it in a `FILE/OPEN` command. You can also do the same thing by using the `FILE/DIALOG` command. Consider these examples:

```
C1=COMMENT/INPUT,Type the full pathway and name of the
file.
V1=FILE/DIALOG, CHOOSE A FILE TO OPEN
FPTR=FILE/OPEN,C1.INPUT,READ
FPTR=FILE/OPEN,V1,READ
```

See "Displaying a File Dialog Box".

Closing an Opened File after Reading or Writing

The **Insert | File I/O Command | File Close** menu option allows you to insert a command in the Edit window that closes an opened file when the measurement routine is executed. Closing files frees up resources used when files are open and commits any changes that have been made to the file to disk.

The syntax of this command in the Edit window is:

```
File/Close, <filepointername>,<closemode>
```

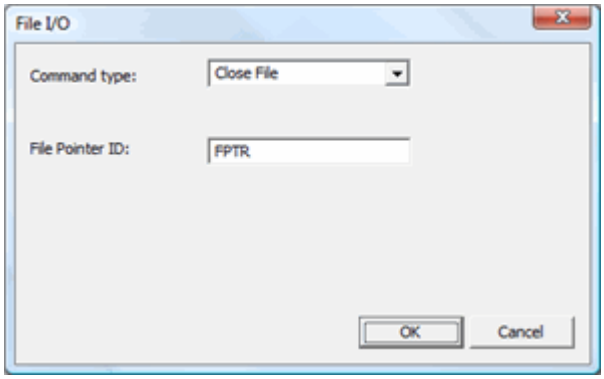
Descriptions for some of the components of this command are:

<filepointername>
This is the ID used to identify the file and is created when the file is opened.

<closemode>
This parameter has two options, KEEP or DELETE. Using KEEP, PC-DMIS simply closes the file defined in the file pointer. Using DELETE, PC-DMIS closes the file and then deletes it.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Close command.
3. Press F9.



Sample Code for File Close

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

This code simply closes the file assigned to the file pointer, `FPTR` :

```
FILE/CLOSE,FPTR,KEEP
```

This code, which uses the `DELETE` parameter, closes and deletes the file assigned to `FPTR` :

```
FILE/CLOSE,FPTR,DELETE
```

Reading a Character from a File

The **Insert | File I/O Command | Reading Commands | Read Character** menu option places a command in the Edit window that reads a single character from the file specified by the filepointername field (see syntax below). It assigns that character to the variable specified in the variable name field.

The syntax of this command in the Edit window is:

```
<varname> = File/ReadCharacter,<filepointername>
```

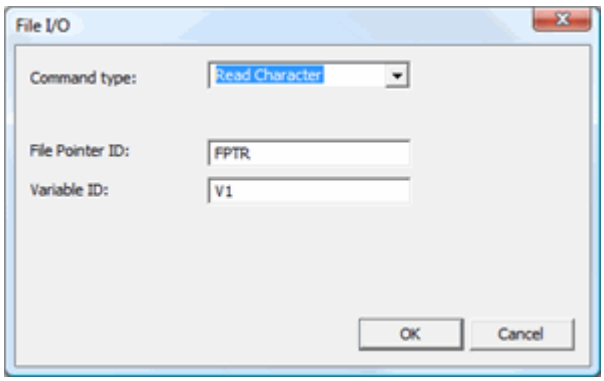
Descriptions for some of the components of this command are:

<filepointername> - This is the ID used to open the file.

<varname> - This is the name of the variable that holds that character.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Read Character command.
3. Press F9.



Sample Code for Read Character

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this example that reads in a line from a data file one character at a time until it encounters a space.

```
   V1=FILE/EXISTS,test.txt
   IF/V1<>0
      COMMENT/OPER,Able to read from data file. Click OK to
continue.
      ASSIGN/V3=""
      FPTR=FILE/OPEN,D:\Program Files\pcdmis35\test.txt,READ
      DO/
         V2=FILE/READ_CHARACTER,FPTR
         ASSIGN/V3=V3+V2
      UNTIL/V2==" "
      FILE/CLOSE,FPTR
      COMMENT/OPER,"The first word from a line of text from
the file is: "+V3
   END_IF/
   ELSE/
      COMMENT/OPER,Wasn't able to read from data file.
Routine will now quit.
      GOTO/END
   END_ELSE/
   END=LABEL/
   ROUTINE/END
```

Code Explanation

V1=FILE/EXISTS

This line checks to see if the specified file exists. The file must be placed in the directory where PC-DMIS resides for this code to work, otherwise the line containing the file must

also contain the full pathway for the file. `V1` receives the result of the file check. It's a non-zero value if it exists; 0 otherwise.

```
IF/V1<>0
```

This line takes the value of `V1` and checks to see if it evaluates to a non-zero value. If so, then a comment appears signifying that it's ready to begin the read process. If equal to zero then the measurement routine ends.

```
ASSIGN/V3=""
```

This line creates an empty string and assigns it to `V3` . The code uses this variable to build a string from the individual read in characters. If you don't create the empty string, then `V3` has its default value of 0.

```
FPTR=FILE/OPEN
```

This line opens the specified file for reading and assigns it to the default file pointer `FPTR` .

```
DO
```

This line begins a `DO / UNTIL` loop. It bounds the `FILE/READ_CHARACTER` code so that characters are continually read in one at a time. The loop exits whenever it reads in a character space.

```
V2=FILE/READ_CHARACTER,FPTR
```

This line reads in a character from the open file tied to the file pointer, `FPTR` . The character is stored in the variable, `V2` .

```
ASSIGN/V3=V3+V2
```

This line uses the empty `V3` variable, concatenates the string `V3` with `V2` , and then reassigns the value to `V3` . So, with subsequent runs of the DO/UNTIL loop, `V3` will get one more character added to it.

```
UNTIL/V2==" "
```

This line ends the DO / UNTIL loop once the `FILE/READ_CHARACTER` code encounters a space character from the opened file.

```
FILE/CLOSE,FPTR
```

This line closes the opened data file, thereby allowing it to be accessed by other system processes. The rest of the code finishes running and displays the first word from the data file in an operator comment.

Reading a Line from a File

The **Insert | File I/O Command | Reading Commands | Read Line** menu option places a command in the Edit window that reads a line from the specified file during execution. This command sets the variable specified by the variable ID to 1 (true) or 0 (false) to indicate success (true) or failure (false) of the call. The expression required by this command can be used to delimit the line read in and to automatically fill up variables and references with data read in from the file. Information is read in from the input file up to the next carriage return character.

The syntax of this command in the Edit window is:

```
<varname> = File/ReadLine,<filepointername>,<expr>
```

Descriptions for some of the components of this command are:

<varname> - This is the name of the variable that holds the result indicating success or failure of the ReadLine command. It returns "OK" or "EOF".

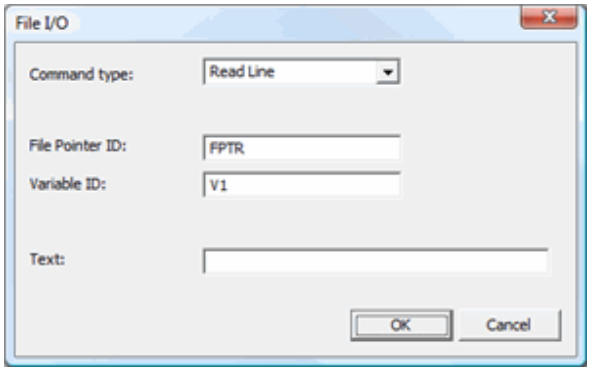
<filepointername> - This is the name specified for the file pointer when the file was opened.

<expr> - This is the destination variable(s) for the input data. Input data can be delimited by text to allow for ease in parsing incoming lines of data. Variables and feature references should be surrounded by curly brackets.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Read Line command.

3. Press F9.



Sample Code for Read Line

> The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this example that reads in a line from a data file one line at a time until the `FILE/READ_LINE` command encounters an empty line. The measurement routine then displays the resulting block of text and quits.

```
.
.
.
V1    =FILE/EXISTS,D:\HEXAGON\PCDMIS
FILES\BASIC_SCRIPTS\TEST.TXT
      IF/V1<>0
         COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
         Able to read from data file. Click OK to continue.
         ASSIGN/V3=""
FPTR  =FILE/OPEN,D:\HEXAGON\PCDMIS
FILES\BASIC_SCRIPTS\TEST.TXT,READ
         DO/
V2          =FILE/READLINE,FPTR,{LINE}
            ASSIGN/V3=V3+LINE
            COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
            "The current value of variable V3 is: "+V3
         UNTIL/V2=="EOF"
         FILE/CLOSE,FPTR,DELETE
         COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
         "The block of text reads as follows: "+V3
      END_IF/
      ELSE/
         COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-
CONTINUE=NO,OVC=NO,
         Wasn't able to read from data file. Routine will
now quit.
         GOTO/END
      END_ELSE/
END   =LABEL/
.
.
.
```

Code Explanation

Much of this code is similar to that explained in "Sample Code for Read Character". Only code unique to this example is listed here.

DO

This line begins a DO / UNTIL loop. It bounds the `FILE/READ_LINE` code so that the lines are continually read in one at a time. The loop exits when it reaches the end of the file.

V2=FILE/READLINE,FPTR,{LINE}

This line reads in all the text until it encounters a carriage return. Instead of storing the text in `V2`, like FILE/READ_CHARACTER would, however, this code acts differently.

- `V2` in this case returns two values: either "OK" or "EOF". "OK" if there's still a line to read in. "EOF" if the end of the file is reached.
- The `{LINE}` code is a user-entered variable that stores the actual text. It is enclosed in curly brackets to tell PC-DMIS it's a variable and not a part of any delimiting text. Without the curly brackets, PC-DMIS would look for a string of characters in the file called "LINE" and would return only the text after "LINE" and before the carriage return.

`ASSIGN/V3=V3+LINE`

This line uses the empty `V3` variable and concatenates the string `V3` with `LINE`, and then reassigns the concatenated value to `V3`. So, with subsequent runs of the DO/UNTIL loop, `V3` will get one more line added to it.

`UNTIL/V2=="EOF"`

This line tests for the condition of the DO / UNTIL loop. Once the `FILE/READLINE` code encounters the end of file, the loop exits. Once the routine flow exits the loop, the rest of the code finishes running and displays the entire block of code inside an operator comment.

`Result=File/ReadLine,F1,"Part ID :"+{V1}` - This causes any text appearing in the read in line after the text "Part ID:" to be assigned to V1. The line is read in from the file opened using F1 as the file pointer name. The result of the read (success or failure) would be stored in the variable Result.

```
File/ReadLine,F1,"Location:"+{VARX}+","+{VARY}+","+
{VARZ}+","+{VARI}+","+{VARJ}+","+{VARK}

ASSIGN/CIR1.XYZ=MPOINT(VARX,VARY,VARZ)

ASSIGN/CIR1.IJK=MPOINT(VARI,VARJ,VARK)
```

The above three command lines read in comma delimited text after the string "Location:" and store the values in the X, Y, Z, and I, J, K values of CIR1.

`File/ReadLine,F1,"Value # "+loopvar+" : "+{var2}` - This causes var2 to be filled with the text appearing after the colon. The loopvar variable in this example is not surrounded by curly brackets and as a result contributes to the delimiting text.

Sample Code Dealing with Numbers Containing Preceding Zeros

If the file you are reading contains lines of numbers, you will notice that PC-DMIS ignores preceding zero characters. For example, if your line contained a value of 005450, it would

read this value strictly as a number and return the value of 5450, ignoring the two preceding zeros. You may or may not want this.

Suppose you have a text file created by an external barcode reader software and it contains these two lines of data:

290291143;582750;0010

291143;5827;0010

You could use some simple code like this to get the number values in-between the semi-colons:

```
   ASSIGN/FIRST_VALUE=0
   ASSIGN/SECOND_VALUE=0
   ASSIGN/THIRD_VALUE=0
   ASSIGN/LINENUM=1
FPTR=FILE/OPEN,D:\TEMP\CODES.TXT,READ
   DO/
   INLINE=FILE/READLINE,FPTR,{FIRST_VALUE}+";"+
{SECOND_VALUE}+";"+{THIRD_VALUE}
   COMMENT/OPER,NO,"LINE NUMBER: "+LINENUM
      ,"First Value: "+FIRST_VALUE
      ,"Second Value: "+SECOND_VALUE
      ,"Third Value: "+THIRD_VALUE
   UNTIL/INLINE=="EOF"
   FILE/CLOSE,FPTR,KEEP
```

While this will successfully parse the lines of text and return the number values, it will also remove any preceding zeros for any values it returns. So, the THIRD_VALUE variable would contain a value of 10, instead of 0010.

To keep the preceding zero values, you would need to treat the whole line as a string and instead use the INDEX, LEFT, and MID string functions to locate the positions of semi-colons in a line of text and obtain the number values:

```
        FPTR=FILE/OPEN,D:\TEMP\CODES.TXT,READ
           ASSIGN/LINENUM=1
           DO/
        LINESTATUS=FILE/READLINE,FPTR,{LINESTR}
           ASSIGN/LINESTR=STR(LINESTR)
           ASSIGN/FIRST_INDEX=INDEX(LINESTR,";")
           ASSIGN/FIRST_VALUE=STR(LEFT(LINESTR,FIRST_INDEX-1))
           ASSIGN/REMAINSTR=STR(MID(LINESTR,(FIRST_INDEX)))
           ASSIGN/SECOND_INDEX=INDEX(REMAINSTR,";")
           ASSIGN/SECOND_VALUE=STR(LEFT(REMAINSTR,SECOND_INDEX-1)
    )
           ASSIGN/THIRD_VALUE=STR(MID(REMAINSTR,SECOND_INDEX))
           COMMENT/OPER,NO,"LINE NUMBER: "+LINENUM
              ,"First Value: "+FIRST_VALUE
              ,"Second Value: "+SECOND_VALUE
              ,"Third Value: "+THIRD_VALUE
           ASSIGN/LINENUM=LINENUM+1
           UNTIL/LINESTATUS=="EOF"
           FILE/CLOSE,FPTR,KEEP
```

Explanation of Code

Much of this code is similar to what is explained above. Only code explanations unique to the mentioned string functions are listed here.

`ASSIGN/FIRST_INDEX=INDEX(LINESTR,";")`

This line locates the position of the first semi-colon in the line and assigns that to the `FIRST_INDEX` variable.

`ASSIGN/FIRST_VALUE=STR(LEFT(LINESTR,FIRST_INDEX-1))`

This line assigns the `FIRST_VALUE` variable the string of characters up to, but not including, the first semi-colon in the `LINESTR` variable. `LINESTR` contains the entire line of text.

`ASSIGN/REMAINSTR=STR(MID(LINESTR,(FIRST_INDEX)))`

This line assigns the `REMAINSTR` variable (standing for "remaining string") the string of left over characters starting from the `FIRST_INDEX` position (the position of the first semi-colon) until the end of the line.

`ASSIGN/SECOND_INDEX=INDEX(REMAINSTR,";")`

This searches inside the REMAINSTR variable for another semi-colon (the second semi colon in the line) and assigns the position to the `SECOND_INDEX` variable.

`ASSIGN/SECOND_VALUE=STR(LEFT(REMAINSTR,SECOND_INDEX-1))`

This line assigns the `SECOND_VALUE variable` the string of characters up to, but not including, the first semi colon in the `REMAINSTR` variable (the second semi-colon in the entire line).

```
ASSIGN/THIRD_VALUE=STR(MID(REMAINSTR,SECOND_INDEX))
```

This line assigns the `THIRD_VALUE` variable the string of characters starting from the `SECOND_INDEX` position, until the end of the line.

Reading a Block of Text from a File

The **Insert | File I/O Command | Reading Commands | Read Block** menu option places a command in the Edit window that reads in a block of characters from an open file at execution time. The amount of characters read in is indicated by the size parameter.

The syntax of this command in the Edit window is:

```
<varname>=File/Read_Block,<fptrname>,<size>
```

Descriptions for some of the components of this command are:

<varname> - This is a variable id for the variable that receives the value indicating success or failure of the read block operation.

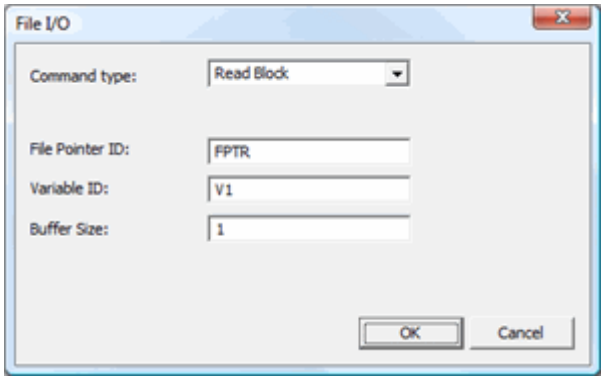
<fptrname> - This is the name specified for the file pointer when the file was opened.

<size> - This is the number of characters to read.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Read Block command.

3. Press F9.



Sample Code for Read Block

> The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Suppose you have various external data files that contain miscellaneous part data and the first few characters of each file designate what the file is for. You can use the `File/Read_Block` command to read only those first few characters before deciding to read in and process every line. Consider this code:

```
   C3=COMMENT/INPUT,Please type the name of the
   ,file code to search for.
   ASSIGN/BLOCKSIZE=LEN(C3.INPUT)
   ASSIGN/FILECODE=C3.INPUT
   DO/
      C1=COMMENT/INPUT,Please type the full pathway,
      ,filename, and extension to the
      ,file you want to process.
      ,Type [Q] to quit.
      IF/C1.INPUT=="Q" OR C1.INPUT=="q"
         COMMENT/OPER,You've chosen to quit. Routine now
ending.
         GOTO/END
      END_IF/
      V1=FILE/EXISTS,C1.INPUT
      IF/V1<>0
         COMMENT/OPER,"Data file ["+C1.INPUT+"] exists.
Click OK to continue."
         FPTR=FILE/OPEN,C1.INPUT,READ
         V2=FILE/READ_BLOCK,FPTR,BLOCKSIZE
         FILE/CLOSE,FPTR
         IF/V2<>FILECODE
            COMMENT/OPER,"The file's code of ["+V2+"]
doesn't match"
            ,"the FILECODE of ["+FILECODE+"]."
         END_IF/
         COMMENT/OPER,"File ["+C1.INPUT+"] is a match."
         ,"The file's code of ["+V2+"] matches"
         ,"the FILECODE of ["+FILECODE+"]."
         COMMENT/OPER,Routine then processes the file.
      END_IF/
      ELSE/
         COMMENT/OPER,"Data file ["+C1.INPUT+"] doesn't
exist. Please retry using an existing data file."
         GOTO/END
      END_ELSE/
   UNTIL/V2==FILECODE
   END=LABEL/
   ROUTINE/END
```

Code Explanation

Some of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`ASSIGN/BLOCKSIZE=LEN(C3.INPUT)`

This line creates a user-defined variable named `BLOCKSIZE` that contains an integer equal to the number of characters found in `C3.INPUT` . This will be used as the size of the block of characters to read in.

ASSIGN/FILECODE=C3.INPUT

This line creates the FILECODE variable and gives it the value of C3.INPUT.

C1=COMMENT/INPUT

This comment stores the full pathway entered by the user into the C1.INPUT variable.

V1=FILE/EXISTS,C1.INPUT

This line checks for the existence of the file name defined in the C1 comment.

DO/

This line begins a DO / UNTIL loop. It bounds the block of code that allows the user to specify a file to read from. It will continue looping until the text assigned to FILECODE variable matches the text read from the file.

V2=FILE/READ_BLOCK,FPTR,BLOCKSIZE

This line reads the amount of characters equal to the integer contained in the BLOCKSIZE variable. The text is then stored in V2 variable.

IF/V2FILECODE

This line begins an IF / END IF code block that tests to see if the text in the V2 variable matches the text stored in the FILECODE variable. If it does match, then the routine continues running. Otherwise it displays a message saying the two codes don't match.

UNTIL/V2==FILECODE

This line checks the condition of the DO / UNTIL loop to see if the text in the V2 variable matches the text in the FILECODE variable. If the statement evaluates to false, the DO loop runs again, allowing the user to choose a different file name. If the statement evaluates to true, then the loop exits and the routine displays a message saying it matches. PC-DMIS could then continue to read each line of data from the specified data file.

Reading Text up to a Delimiter

The **Insert | File I/O Command | Reading Commands | Read Up To** menu option places a command in the Edit window that reads all text "up to" one of the given delimiters from the specified file during execution. Any text read by this command is placed in the specified destination variable. The command stops reading text when PC-DMIS encounters the following:

- Defined delimiters

- Carriage returns
- Line feed characters

If the end of the file is reached, the destination variable is set to "EOF" (End of File).

The syntax of this command in the Edit window is:

```
<varname> = FILE/READ_UPTO,<fptrname>,<delimiters>
```

Descriptions for some of the components of this command are:

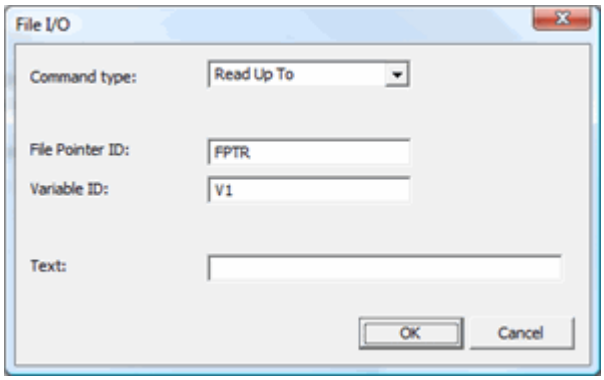
<varname> - This is the name of the destination variable.

<fptrname> - This is the name specified for the file pointer when the file was opened.

<delimiters> - This is a string which contains zero or more delimiter characters.

To access the dialog box associated with this File I / O command:

1. Select the **Edit Window** to open the Edit window.
2. Place your cursor on the `FILE/READ` command.
3. Press F9. The **File I/O** dialog box opens.



Once the dialog box appears:

1. Type the variable name that receives the read in information into the **Variable ID** box.
2. Type the file pointer name into the **File Pointer ID** box.
3. Type the delimiter into the **Text** box (be sure and use quotation marks around your chosen delimiter).
4. Click **OK**.

Sample Code for Read Up To

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this example where you have a text file named "sample.txt" in your D:\Temp directory which contains this information on the first line.

```
CIR1:2.54:CIRCLE
```

To use the Read Up To command on this file:

1. Insert a `FILE/OPEN` command in the Edit window.
2. Use a File Pointer Name of your choice to name your File Open command. This example uses "sample" as the file pointer name.

The File Open command should look something like this:

```
SAMPLE      =FILE/OPEN,D:\TEMP\SAMPLE.TXT,READ
```

Now, use PC-DMIS Read Up To commands to define some variables that call different segments of data. This example uses the following variables looking for a colon character ":" (without the quotation marks) as the delimiter.

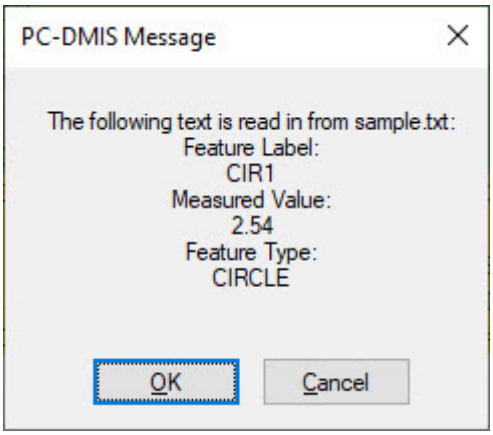
```
V_LABEL    =FILE/READ_UPTO,SAMPLE,:
V_VALUE    =FILE/READ_UPTO,SAMPLE,:
V_TYPE     =FILE/READ_UPTO,SAMPLE,:
```

Thus, when PC-DMIS executes these lines, it sets these variables to hold these values:

- V_LABEL = CIR1
- V_VALUE = 2.54
- V_TYPE = CIRCLE

To display this on screen during execution, you can use an operator comment such as the one shown here:

```
COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,OVC=NO,
The following text is read in from sample.txt:
Feature Label:
V_LABEL
Measured Value:
V_VALUE
Feature Type:
V_TYPE
```



Writing a Character to a File

The **Insert | File I/O Command | Writing Commands | Write Character** menu option inserts a command into the Edit window that causes a single character to be output to a file on your computer upon execution.

The syntax of this command in the Edit window is:

```
File/Write_Character,<fptrname>,<expr>
```

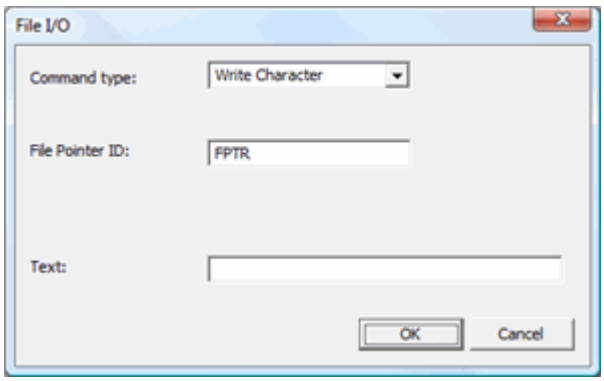
Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file pointer specified when the file was opened.

<expr> - This is the character to be written to file. If the expression evaluates to more than one character, only the first character is written.

To access the dialog box associated with this File I / O command, follow these steps:

1. Open the Edit window.
2. Place your cursor on the Write Character command.
3. Press F9.



Sample Code for Write Character

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this code that writes a string provided by the user to a data file one character at a time.

```
C1=COMMENT/INPUT,Type the name of the file to write
,to (include the complete path).
FPTR=FILE/OPEN,C1.INPUT,WRITE
C2=COMMENT/INPUT,Type something to send to the file.
,This will send the string one character
,at a time.
ASSIGN/COUNT=0
ASSIGN/LENGTH=LEN(C2.INPUT)
DO/
   ASSIGN/WRITETHIS=MID(C2.INPUT,COUNT,1)
   FILE/WRITE_CHARACTER,FPTR,WRITETHIS
   ASSIGN/COUNT=COUNT + 1
UNTIL/COUNT==LENGTH
```

Code Explanation

Some of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`FPTR=FILE/OPEN,C1.INPUT,WRITE`

This line opens the file specified in the C1 comment for writing, and assigns it to the file pointer, `FPTR` . All data in this file will get overwritten as long as the file pointer begins at the start of the data file.

`ASSIGN/COUNT=0`

This line assigns a user defined variable COUNT a value of zero. This is used for looping purposes to print the string one character at a time.

ASSIGN/LENGTH=LEN(C2.INPUT)

This line uses the LEN( ) function to return the length of a string. This function takes one parameter, the string. It counts the number of characters in the string (including spaces) and returns an integer value of that amount. In this case, the user defined variable, LENGTH holds this value.

DO/

This line begins a DO / UNTIL loop. Code between the DO and the UNTIL statements will be executed until the loop's condition evaluates to true.

ASSIGN/WRITETHIS=MID(C2.INPUT,COUNT,1)

This line creates a user defined variable called WRITETHIS and uses the MID( ) function to return a substring character from the C2.INPUT string and give it to WRITETHIS .

MID( ) takes three parameters.

- Parameter 1: is the string from which to get values. In this case C2.INPUT is used.
- Parameter 2: is the position in the string to take the character from. The first character in a string would be position 0, the second position 1, the third position 2 and so forth. In this case, the variable COUNT is used.
- Parameter 3: is how many characters starting from the position of the second parameter to grab. In this case, the value of 1 is used (the sample only writes one character at a time, so there's no reason to get more).

FILE/WRITE_CHARACTER,FPTR,WRITETHIS

This line writes the character stored in the WRITETHIS variable to the file specified by the file pointer, FPTR .

ASSIGN/COUNT=COUNT+1

This line takes the current **COUNT** value, increments it by one, and then places the new value back into COUNT .

UNTIL/COUNT==LENGTH

This line tests the condition of the DO / UNTIL loop. In this case, the loop will keep incrementing the COUNT variable until it has the same value as the LENGTH variable. Then the loop will exit, ending the routine.

Writing a Line to a File

The **Insert | File I/O Command | Writing Commands | Write Line** menu option inserts a command into the Edit window that causes a line of text to be output to a file on your computer upon execution. Use expression syntax to output variables and measurement routine information to file. A carriage return is automatically appended to the text written out.

The syntax of this command in the Edit window is:

```
File/WriteLine,<fptrname>,<expr>
```

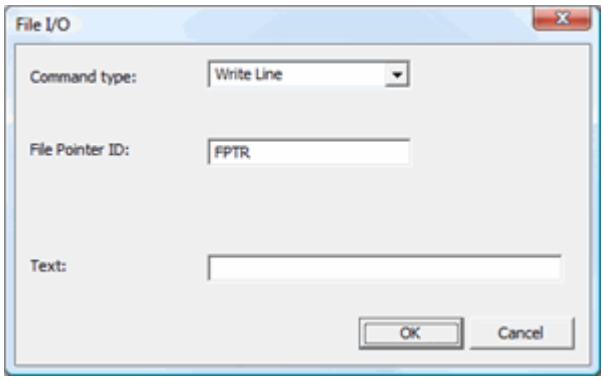
Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file reference specified when the file was opened.

<expr> - This is the text to be written to file. Expressions can be used in this field.

To access the dialog box associated with this File I / O command, follow these steps:

1. Open the Edit window.
2. Place your cursor on the Write Line command.
3. Press F9.



Sample Code for Write Line

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Suppose you want to export some measured XYZ values to a data file. The following code allows you to input a feature label and a data file and send the X,Y, and Z data for that feature to a data file.

```
C1=COMMENT/INPUT,Type the label of the feature.
,to use.
C2=COMMENT/INPUT,Type the name of the file to write
,to (include the complete path).
FPTR=FILE/OPEN,C2.INPUT,APPEND
ASSIGN/FEATNAME=C1.INPUT
ASSIGN/ALLVALS=FEATNAME.X+","+FEATNAME.Y+","+FEATNAME.Z
COMMENT/OPER,"Text to write is: "+ALLVALS
FILE/WRITELINE,FPTR,ALLVALS
FILE/CLOSE,FPTR
```

Code Explanation

Some of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`FPTR=FILE/OPEN,C2.INPUT,APPEND`

This line opens the file specified in the C2 comment for appending, and assigns it to the file pointer, `FPTR`. If instead, you change `APPEND` to `WRITE`, then existing content in the data file will get overwritten.

`ASSIGN/FEATNAME=C1.INPUT`

This line assigns the string of the feature label from `C1.INPUT` to the user defined variable, `FEATNAME`.

`ASSIGN/ALLVALS=FEATNAME.X+","+FEATNAME.Y+","+ FEATNAME.Z`

This line gives the user defined variable `ALLVALS` the value of `FEATNAME.X,FEATNAME.Y,FEATNAME.Z`, in other words it now holds the X, Y, and Z values of the feature label typed into the C1 input comment.

`FILE/WRITELINE,FPTR,ALLVALS`

This line writes the values contained in `ALLVALS` to the file specified by the file pointer, `FPTR`.

Writing a Block of Text to a File

The **Insert | File I/O Command | Writing Commands | Write Block** menu option inserts a command into the Edit window that causes a block of text to be output to a file on your computer upon execution. Use expression syntax to output the variables and measurement routine information to file. Unlike the write line command, write block does *not* append a carriage return at the end.

The syntax of this command in the Edit window is:

```
File/WriteBlock,<fptrname>,<expr>
```

Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file reference specified when the file was opened.

<expr> - This is the text to be written to file. Expressions can be used in this field.

Unlike the write line command, write block does *not* append a carriage return at the end. However, if you need to place text on a new line inside your text block, you can insert a carriage return and line feed manually by using the CHR(10) code outside of your quoted string, as shown here in this example:

```
FILE/WRITEBLOCK,FPTR, "CHR (10) inserts text... " + CHR(10)
+ " ...on a new line."
```

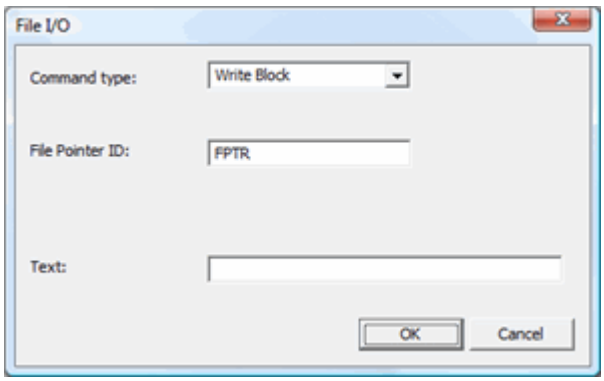
This would yield this result inside your output file:

```
CHR(10) inserts text...

 ...on a new line.
```

Notice that if CHR(10) is inside the quotation marks the actual text of CHR(10) gets sent to the file.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Write Block command.
3. Press F9.



Sample Code for Write Block

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code writes whatever the user inputs into an input comment, and appends a colon to be used as a delimiter.

```
   C1=COMMENT/INPUT,Type any string. PC-DMIS will append a
colon (for delimiter purposes) and write the string to a
file of your choice
   C2=COMMENT/INPUT,Type the name of the file to write
   ,to (include the complete path).
   FPTR=FILE/OPEN,C2.INPUT,APPEND
   ASSIGN/WRITETHIS=C1.INPUT+":"
   COMMENT/OPER,"Text to write is: "+WRITETHIS
   FILE/WRITELINE,FPTR,WRITETHIS
   FILE/CLOSE,FPTR
```

Code Explanation

Some of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`FPTR=FILE/OPEN,C2.INPUT,APPEND`

This line opens the file specified in the C2 comment for appending, and assigns it to the file pointer, `FPTR`.

`ASSIGN/WRITETHIS=C1.INPUT+":"`

This line appends a colon to the text contained in `C1.INPUT` and assigns the new string to the user defined variable, `WRITETHIS`.

`FILE/WRITELINE,FPTR,WRITETHIS`

This line writes the values contained in `WRITETHIS` to the file specified by the file pointer, `FPTR` . You can later read-in text from the file by using the colon as a delimiter.

Positioning a File Pointer at the Beginning of a File

The **Insert | File I/O Command | Position Commands | Rewind To Start** menu option inserts a command into the Edit window that will position the file pointer to the beginning of the file stream.

The syntax of this command in the Edit window is:

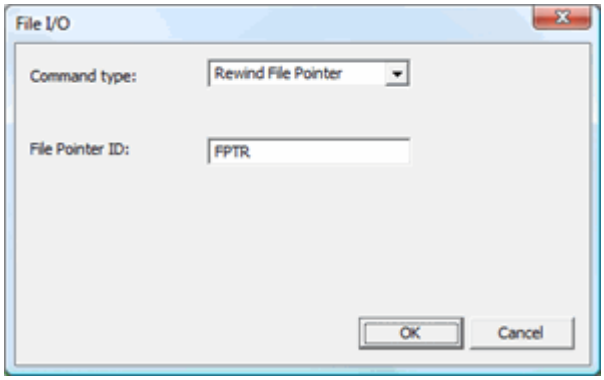
```
File/Rewind,<fptrname>
```

Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file pointer to reposition at the beginning of the file.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Rewind to Start command.
3. Press F9.



Sample Code for Rewind to Start

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this example that reads in data from an external file one line at a time. After each line, you have the option of starting over and reading from the beginning of the file. This illustrates the use of the `FILE/REWIND` command.

```
C1=COMMENT/INPUT,Please type a file to read from.
,(include the full path)
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   DO/
      FPTR=FILE/OPEN,C1.INPUT,READ
      C2=COMMENT/YESNO,Do you want to read from the
beginning?
      IF/C2.INPUT == "YES"
         FILE/REWIND,FPTR
      END_IF/
      V2=FILE/READLINE,FPTR,{LINE}
      COMMENT/OPER,"The current line is: " + LINE
   UNTIL/V2=="EOF"
END_IF/
FILE/CLOSE,FPTR
COMMENT/OPER,Routine quitting.
```

Code Explanation

Some of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`C2=COMMENT/YESNO`

This line asks if you want to start reading the file from the beginning. It stores the YES/NO response into the variable, `C2.INPUT` .

`IF/C2.INPUT == "YES"`

This line begins an IF / END IF block. It tests the condition of `C2.INPUT` having the value of YES. If the condition is true, then PC-DMIS executes the lines following the `IF` statement. If the condition is false, then PC-DMIS executes the code following the `END_IF` statement.

`FILE/REWIND,FPTR`

This line rewinds the file pointer to the beginning of the data file.

`END_IF/`

This line quits the IF / END IF code block.

Saving a File Pointer's Current Position

The **Insert | File I/O Command | Position Commands | Save File Position** menu option inserts a command into the Edit window that saves the current position of a file pointer within the file stream. The saved position can later be recalled using the recall file position command.

The syntax of this command in the Edit window is:

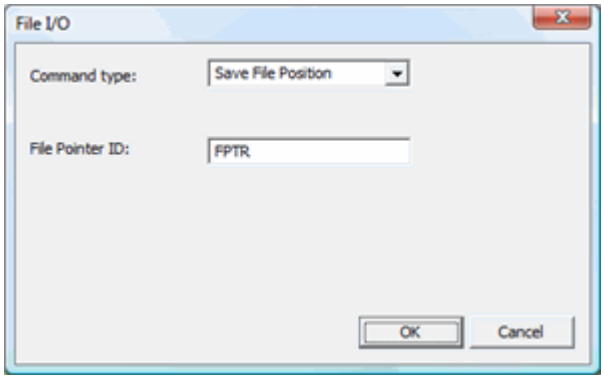
```
File/SavePosition,<fptrname>
```

Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file pointer whose file position is saved.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Save File Position command.
3. Press F9.



Sample Code for Save File Position

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

Consider this example that reads in data from an external file one line at a time. After each line, you have the option of saving the file position for later recall. This illustrates the use of the `FILE/SAVE_POSITION` command.

```
C1=COMMENT/INPUT,Please type a file to read from.
,(include the full path)
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   DO/
      FPTR=FILE/OPEN,C1.INPUT,READ
      C2=COMMENT/YESNO,Do you want to save the file
position and recall it later? The loop will quit.
      IF/C2.INPUT == "YES"
         FILE/SAVE_POSITION,FPTR
         GOTO/QUITLOOP
      END_IF/
      V2=FILE/READLINE,FPTR,{LINE}
      COMMENT/OPER,"The current line is: " + LINE
   UNTIL/V2=="EOF"
END_IF/
FILE/CLOSE,FPTR
QUITLOOP=LABEL/
COMMENT/OPER,You've stopped reading.
ROUTINE/END
```

Code Explanation

This code is similar to that explained in "Sample Code for Rewind to Start".

Only explanations unique to this example are given here.

`C2=COMMENT/YESNO`

This line asks if you want to store the current file position and exit the loop. It stores the YES/NO response into the variable, `C2.INPUT` .

`FILE/SAVE_POSITION,FPTR`

This line stores the file pointer's position in the file stream.

As long as you open the same file with same file pointer name in the same measurement routine, you can recall a stored file position and continue reading where you left off. To continue this example, see the "Sample Code for Recall File Position" topic.

Recalling a Saved File Pointer's Position

The **Insert | File I/O Command | Position Commands | Recall File Position** inserts a command into the Edit window that recalls a previously-saved file position. Use the save file position command to save a position within an open file.

The syntax of this command in the Edit window is:

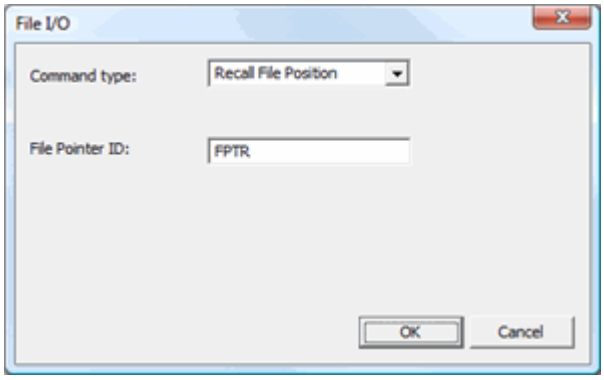
```
File/RecallPosition, <fptrname>
```

Descriptions for some of the components of this command are:

<fptrname> - This is the name of the file pointer whose position is being recalled.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the Recall File Position command.
3. Press F9.



Sample Code for Recall File Position

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

This example opens a previously-closed file, uses a previous file pointer, and recalls the stored file pointer's saved position. It then reads in data from that position. This code illustrates the use of the `FILE/RECALL_POSITION` command. It continues the code sample given in the "Sample Code for Save File Position" topic.

```
   COMMENT/OPER,The routine will now recall the stored file
position.
   FPTR=FILE/OPEN,C1.INPUT,READ
   FILE/REWIND,FPTR
   COMMENT/OPER,To test, file has been rewound.
   ,The first line will be read in to test the rewind.
   V3=FILE/READLINE,FPTR,{LINE}
   COMMENT/OPER,The first line is:
   ,LINE
   FILE/REWIND,FPTR
   FILE/RECALL_POSITION,FPTR
   COMMENT/OPER,Previously stored file position has been
recalled.
   ,Data on the line at the stored position will now print.
   V4=FILE/READLINE,FPTR,{STORED}
   COMMENT/OPER,The text at the stored position is:
   ,STORED
```

Code Explanation

This code is similar to that explained in "Sample Code for Rewind to Start".

Only explanations unique to this example are given here.

`FILE/RECALL_POSITION,FPTR`p

This line recalls the stored file pointer position in the file stream for the file pointer designated as `FPTR` .

`V4=FILE/READLINE,FPTR,{STORED}`

This line reads in the next line after the stored file pointer position and assigns it to the user-defined variable of `STORED` . This variable is then printed out in the next operator comment.

Copying a File

The **Insert | File I/O Command | File Copy** menu option inserts a command into the Edit window that causes a file copy operation to occur upon execution.

The syntax of this command in the Edit window is:

```
File/Copy,<srcfilename>,<destfilename>,<replacemode>
```

Descriptions for some of the components of this command are:

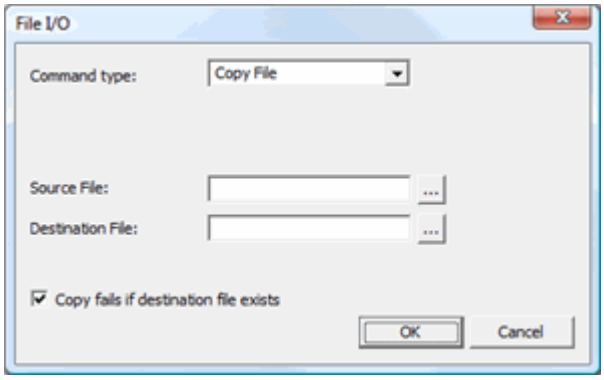
<srcfilename> - This is the name of the source file (the file copied from).

<destfilename> - This is the name of the destination file (the file copied to).

<replacemode> - This is the action to take if the destination file already exists. The two modes are overwritten and fail if the destination exists.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Copy command.
3. Press F9.



Sample Code for File Copy

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code asks for a file name to copy and for a destination directory and file to copy to.

```
C1=COMMENT/INPUT,Please type a file you want to copy.
,(Include full file path)
C2=COMMENT/INPUT,Please type a destination file name.
,(Include full file path)
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   COMMENT/OPER,File exists to copy. Copying commencing.
   FILE/COPY,C1.INPUT,C2.INPUT,FAIL_IF_DEST_EXISTS
   V2=FILE/EXISTS,C2.INPUT
   IF/V2==0
      COMMENT/OPER,"File doesn't exist at: " + C2.INPUT
      ,Copy ending.
      ROUTINE/END
   END_IF/
ELSE/
   COMMENT/OPER,File copy successful.
   ROUTINE/END
END_ELSE/
END_IF/
COMMENT/OPER,File to copy doesn't exist.
```

Code Explanation

Much of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`C1=COMMENT/INPUT`

This line takes the full pathway of the file to copy and places it into the `C1.INPUT` variable.

`C2=COMMENT/INPUT`

This line takes the full pathway of the destination file and places it into the `C2. INPUT` variable.

`FILE/COPY,C1.INPUT,C2.INPUT,FAIL_IF_DEST_EXISTS`

This line copies the original file to a destination file. This command takes three parameters.

- Parameter 1 is `C1. INPUT` . This is the full path to the file to copy.
- Parameter 2 is `C2. INPUT` , or the full path to the destination file.
- Parameter 3, in this case, aborts the FILE/COPY procedure if it encounters an existing file with the same destination filename. You can set this so that it overwrites existing files of the same name.

Moving a File

The **Insert | File I/O Command | File Move** menu option inserts a command into the Edit window that causes a file move operation to occur upon execution.

The syntax of this command in the Edit window is:

```
File/Move,<oldfilename>,<newfilename>
```

Descriptions for some of the components of this command are:

<oldfilename> - This is the location and name of the file.

<newfilename> - This is the new location and name of the file.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Move command.
3. Press F9.

Sample Code for File Move

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code asks for a file name to move and a location directory and file name to move to. It then performs the file move operation.

```
C1=COMMENT/INPUT,Please type a file you want to move.
,(Include full file path)
C2=COMMENT/INPUT,Please type a destination file name.
,(Include full file path)
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   COMMENT/OPER,File exists to move. File move
commencing.
   FILE/MOVE,C1.INPUT,C2.INPUT
   V2=FILE/EXISTS,C2.INPUT
   IF/V2==0
      COMMENT/OPER,"File doesn't exist at: " + C2.INPUT
      ,The MOVE didn't function properly.
      ROUTINE/END
   END_IF/
ELSE/
   COMMENT/OPER,File MOVE successful.
   ROUTINE/END
END_ELSE/
END_IF/
COMMENT/OPER,Original file doesn't exist. Try again.
```

Code Explanation

Much of this code is similar to that explained in "Sample Code for File ".

Only explanations unique to this example are given here.

`FILE/MOVE,C1.INPUT,C2.INPUT`

This line copies the original file to a destination file. This command takes two parameters.

- Parameter 1 is `C1.INPUT` . This is the full path to the file to move.
- Parameter 2 is `C2.` INPUT, or the full path to the destination file.

Deleting a File

The **Insert | File I/O Command | File Delete** menu option inserts a command into the Edit window that deletes a file when the command executes.

The syntax of this command in the Edit window is:

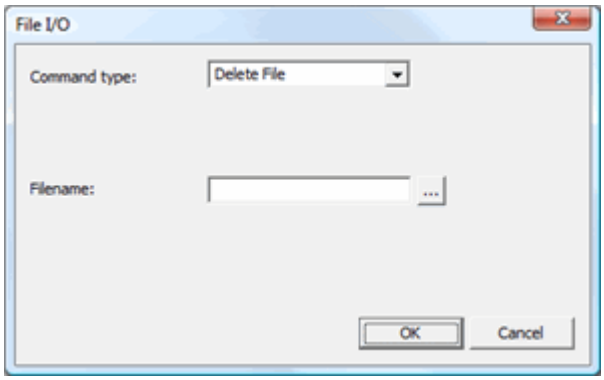
```
File/Delete,<filename>
```

Descriptions for some of the components of this command are:

<filename> - This is the name of the file to delete.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Delete command.
3. Press F9.



Sample Code for File Delete

> The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code asks for a file name and then deletes the file.

```
C1=COMMENT/INPUT,Please type a file you want to delete.
,(Include full file path)
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   COMMENT/OPER,File exists. Ready to delete.
   FILE/DELETE,C1.INPUT
   V2=FILE/EXISTS,
   IF/V2==0
      COMMENT/OPER,File deleted successfully
      ROUTINE/END
   END_IF/
ELSE/
   COMMENT/OPER,File still exists
   ROUTINE/END
END_ELSE/
END_IF/
COMMENT/OPER,File doesn't exist to delete. Choose a file
that exists.
```

Code Explanation

Much of this code is similar to that explained in "Sample Code for File Move ".

Only explanations unique to this example are given here.

`FILE/DELETE,C1.INPUT` - This line deletes the file specified. This command takes one parameter, the name of the file to delete. In this case, `C1.INPUT` .

Checking for a File's Existence

The **Insert | File I/O Command | File Exists** menu option inserts a command into the Edit window that checks for the existence of a file when executed and sets the supplied variable with the result.

The syntax of this command in the Edit window is:

```
<varname> = File/Exists,<filename>
```

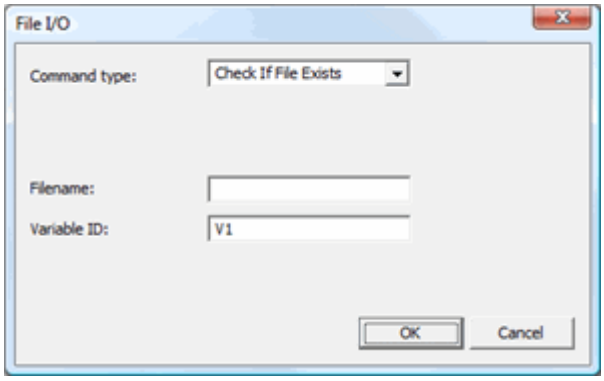
Descriptions for some of the components of this command are:

<varname> - This is the name of the variable that is set to the result of the check that is performed. The variable is set to 1 if the file exists and 0 if the file does not exist.

<filename> - This is the name of the file being checked to see if it exists on the disk.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Exists command.
3. Press F9.



Sample Code for File Exists

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code asks for a file name and then checks for the file's existence.

```
C1=COMMENT/INPUT,Please type a file you want to check.
V1=FILE/EXISTS,C1.INPUT
IF/V1<>0
   COMMENT/OPER,File exists.
END_IF/
ELSE/
   COMMENT/OPER,File doesn't exist
END_ELSE/
```

Code Explanation

Much of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`V1=FILE/EXISTS,C1.INPUT`

This line checks to see if the specified file exists. The file must be placed in the directory where PC-DMIS resides for this code to work; otherwise, the line containing the file must

also contain the full pathway for the file. `V1` receives the result of the file check. It's a non-zero value if it exists; 0 otherwise.

Displaying a File Dialog Box

The **Insert | File I/O Command | File Dialog** menu option inserts a command into the Edit window that brings up an **Open** dialog box during execution. This allows the operator to choose a file name at run time. It stores the name of the file chosen in the specified variable.

The syntax of this command in the Edit window is:

```
<varname> = File/Dialog,<expr>
```

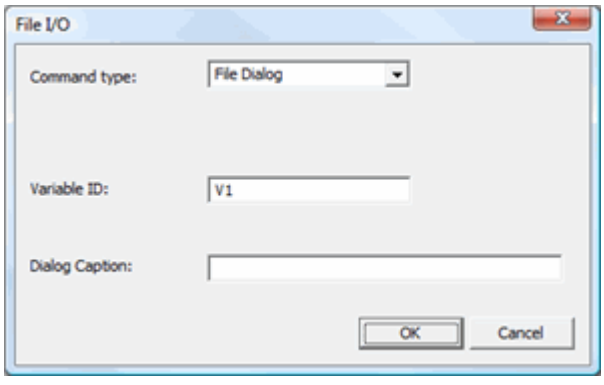
Descriptions for some of the components of this command are:

<varname> - This is the name of the variable that is set to the name of the file chosen by the user in the file dialog box.

<expr> - This is the text that appears on the title bar of the file dialog box.

To access the dialog box associated with this File I / O command:

1. Open the Edit window.
2. Place your cursor on the File Dialog command.
3. Press F9.



Sample Code for File Dialog

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code submits a dialog box that allows you to choose a file to delete.

```
V1=FILE/DIALOG,Choose a file to delete.
V2=FILE/EXISTS,V1
IF/V2<>0
   COMMENT/OPER,File exists. Ready to delete.
   FILE/DELETE,V1
   V3=FILE/EXISTS,
   IF/V3==0
      COMMENT/OPER,File deleted successfully
      ROUTINE/END
   END_IF/
ELSE/
   COMMENT/OPER,File still exists
   ROUTINE/END
END_ELSE/
END_IF/
COMMENT/OPER,File doesn't exist to delete. Choose a file
that exists.
```

Much of this code is similar to that explained in "Sample Code for Read Character" or in "Sample Code for Read Line".

Only explanations unique to this example are given here.

`V1=FILE/Dialog,Choose a file to delete`

This line displays a dialog box with the title "Choose a file to delete". You can browse to a file and when you click **Open**, PC-DMIS gives V1 the full path to the selected file. The rest of the routine deletes the selected file.

Checking for the End of a File or the End of a Line

PC-DMIS allows you to check for the End of a File by using the functions `EOF` or `EOL` in a conditional test.

`EOF` stands for END OF FILE. This function takes a file pointer of type string. When properly placed within a conditional statement, it tests to see if the file pointer has reached the end of the specified file. If it has, then the function returns true.

`EOL` stands for END OF LINE. This function takes a file pointer of type string. When properly placed within a conditional statement, it tests to see if the file pointer has reached the end of a line in the specified file. If it has, then the function returns true. This works best inside of a loop.

The syntax of this command in the Edit window is:

`EOF(<filepointer>)` or `EOL(<filepointer>)`

Descriptions for some of the components of this command are:

<filepointer> - This is the name of the file pointer that you're checking.

Sample code for EOF and EOL

The sample code below should be entered inside the Edit window's Command Mode and not inside the **File I/O** dialog box.

The following code opens test.txt and reads through the file. As long as the end of file hasn't been reached (designated with the code, `WHILE/! EOF )` , PC-DMIS reads through the file character by character, and assigns a character to V1.

If PC-DMIS reaches the end of a line in the file, PC-DMIS shows the last character on that line.

This repeats until PC-DMIS reaches the end of the file. PC-DMIS then shows the text "End of File Reached…".

```
   FPTR=FILE/OPEN,D:\temp\test.txt,READ
   WHILE/!EOF("FPTR")
   V1=FILE/READ_CHARACTER,FPTR
   IF/EOL("FPTR")
   COMMENT/OPER,NO,"End of Line Reached. The last character
is:"
   ,V1
   END_IF/
   END_WHILE/
   COMMENT/OPER,NO,"End of File Reached..."
```

Using Expressions and Variables

Using Expressions and Variables: Introduction

An expression is a user-defined condition used with PC-DMIS's flow control commands. Using flow control statements, you can test these conditions in your measurement routine. If the condition is met or not, you can determine what action PC-DMIS takes.

Expressions are important to making PC-DMIS accomplish your specific tasks. When you use expressions in conjunction with flow control commands, you can unleash even more of PC-DMIS's powerful functionality.

This chapter explains how to create and use expressions inside the PC-DMIS Edit window. When you work with expressions, you should place the Edit window into Command Mode. This allows you to view the Edit window code directly.

The following major topics are covered in this chapter:

- Using Expressions in a Measurement Routine
- Creating Expressions with the Expression Builder
- Using Variables with Expressions
- Understanding Expression Components
- Accessing a Report's Object Properties
- Accessing Information from a Constructed Scan Minimum Circle

If you're looking for information on Reporting Expressions, see the "About Report Expressions" in the "Reporting Measurement Results" chapter.

Using Expressions in a Measurement Routine

The PC-DMIS Edit window allows expressions in most of its editable fields. Editable fields are usually those fields that highlight in yellow when you press the Tab key inside the Edit

window while in Command Mode. Fields that change the type of the feature do not allow expressions.

The box of an Auto feature that specifies the type of the Auto feature, such as a Surface point, Auto Circle, Auto Round Slot, and so on, does not allow expressions.

The subtopics under this topic offer a complete reference to available expressions.

Viewing Expression Values

To see the value of an expression, position the mouse cursor over the expression and leave it in position for at least one second. The expression is evaluated and a small yellow pop-up window that displays the expression and its current value appears just below the mouse cursor.

Keeping Expression Values Only

To immediately solve an expression in the Edit window and keep only the value, follow these steps:

1. Select the expression text in the Edit window.
2. Precede the expression text with a ` (accent grave) character.

Suppose you type the expression`**1/7** in a numeric field. The expression is immediately solved and only the value (**0.143**) is placed in the field.

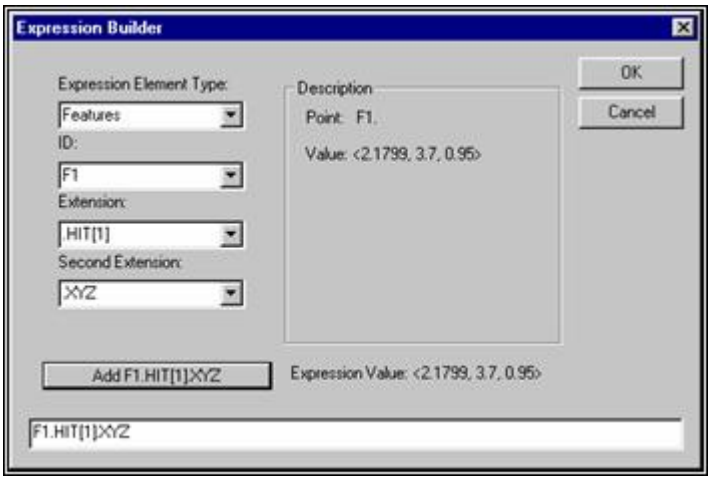
Using Expressions with Branching

The flow control commands use expressions to determine flow of routine execution. Refer to the "Branching by Using Flow Control" chapter for information on when branching may or may not occur.

Using Expressions with File Input / Output

Writing data to an external data file or reading data from an external data file often uses variables and other expressions to effectively manage and store or display that data. Refer to the "Using File Input / Output" chapter for more information.

Creating Expressions with the Expression Builder



Expression Builder Dialog Box

PC-DMIS allows you to create and add expressions into the Edit window by typing them or using the interface of the **Expression Builder** dialog box. The **Edit | Expression** menu option displays the **Expression Builder** dialog box.

This dialog box allows you to create an expression and insert it into an editable field. Pressing the F2 key while the cursor is on a field that allows expressions also displays the **Expression Builder** dialog box.

The **Expressions Builder** dialog lists all of the types of operators and functions that are available for expressions.

Creating an Expression by Typing

To create an expression by typing it directly into the Edit window:

1. Open the Edit window (**View | Edit Window**).
2. Place the Edit window into Command Mode.
3. Press Tab to move your cursor to an editable field where you want to insert the expression. Fields with a yellow highlight are considered "editable".
4. Type the expression.

Creating an Expression with the Expression Builder

You must be in Command Mode for the Expression option to be enabled.

To enter an expression by using the Expression Builder dialog box (**Edit | Expression**):

1. Open the Edit window (**View | Edit Window**).
2. Place the Edit window into Command Mode (**View | Command Mode**).
3. Move the cursor to an editable field where you want to insert the expression.
4. Press the F2 key while the cursor is on a field that allows expressions. The **Expression Builder** dialog box appears. The **Expression Builder** dialog box lists all of the operators, operands, and functions. The following can be referenced through this dialog box:
    - Available expression types
    - Variables
    - Features
    - Dimensions
    - Alignments
    - Comments
5. Select the expression element type from the first drop-down list. Depending on your selection, other combination boxes appear.
6. Select the desired ID from the **ID** drop-down list.
7. Select an extension from the **Extension** drop-down list.
8. Select another extension from the **Second Extension** drop-down list. If the expression is usable, the **Add** button becomes available.
9. Click the **Add** button. The expression appears in an edit box.
10. Click the **OK** button. The expression now appears where your cursor is in the Edit window.

You can also open the **Expression Builder** dialog box from these other dialog boxes:

- The **If Expression** dialog box - Select **Insert | Flow Control Command | If Goto**. Click on the **Expression** button.
- The **Assignment** dialog box - Select **Insert | Assignment**. Click the **Assign To** or **Assign From** button.

Once the expression is created, PC-DMIS automatically inserts the expression at the next legal position in the Edit window.

Checking the Expression for Correctness

When your cursor leaves the field where you added the expression, PC-DMIS attempts to check the expression for correctness. If there is a problem with the expression, an error message that indicates an invalid number may appear, or the expression text may turn red. Also, expressions that refer to non-existent objects appear in red text.

Since the test of expression correctness occurs at the time you leave a field, a field that turned red due to a reference of a nonexistent object, (ex. CIRCLE1.X) will remain red even if the new object (for example, CIRCLE1) is added. The field remains red until the expression is retested for correctness.

To retest the expression for correctness:

1. Move your cursor to the field of the expression.
2. Press the F2 key. The **Expression Builder** dialog box opens again. Any changes to your expression appear in the edit box.
3. Press the ENTER key to close the dialog box.

Expression Element Type

The Expression Element Type drop-down list in the **Expression Builder** dialog box (**Edit | Expression**) lists the various expression element types available to be placed into expressions. These include:

- Functions
- Operators
- Alignments
- Comments

- Dimensions
- Features
- Variables

ID

The **ID** drop-down list in the **Expression Builder** dialog box (**Edit | Expression**) lists the set of items available based on the expression element type selected in the **Expression Element Type** drop-down list.

The list of available items depends on the selected expression element:

- When you select **Functions & Operators** from the **Expression Element Type** drop-down list, the **ID** drop-down list contains a list of the available functions and operators.
- When you select **Features** from the **Expression Element Type** drop-down list, the **ID** drop-down list displays the IDs of the features in the measurement routine.

Extension

The **Extension** drop-down list in the **Expression Builder** dialog box (**Edit | Expression**) becomes available when the item chosen in the **ID** drop-down list requires the addition of an extension in order to form a usable expression element. The **Extension** drop-down list displays available extensions based on the item selected in the **ID** drop-down list.

Suppose you select a feature from the **ID** drop-down list. PC-DMIS shows the possible extensions that you can use to reference data of that feature (such as "X", "Y", "Z", "Diam", "Length", etc.) in the **Extension** drop-down list.

Possible extensions include these measured or theoretical data types:

Measured

- All – All the values of the feature are assigned to the variable. See the example below.
- X – Measured X values of the hits
- Y – Measured Y values of the hits
- Z – Measured Z values of the hits

- XYZ – Measured XYZ values of the hits
- I – Measured I values of the hits
- J – Measured J values of the hits
- K – Measured K values of the hits
- IJK – Measured IJK values of the hits

Theoretical

- TX – Theoretical X values of the hits
- TY – Theoretical Y values of the hits
- TZ – Theoretical Z values of the hits
- TXYZ – Theoretical XYZ values of the hits
- TI – Theoretical I values of the hits
- TJ – Theoretical J values of the hits
- TK – Theoretical K values of the hits
- TIJK – Theoretical IJK values of the hits

Suppose this code snippet is part of a measurement routine:

```
F1=GENERIC/POINT,DEPENDENT,CARTESIAN,$
     NOM/XYZ,<8,9,10>,$
     MEAS/XYZ,<7.98,8.98,9.98>,$
     NOM/IJK,<1,0,0>,$
     MEAS/IJK,<1,0,0>
     ASSIGN/MYFEATURE=F1.ALL
     ASSIGN/V1=MYFEATURE.X
     ASSIGN/V2=MYFEATURE.TX
```

When the measurement routine completes the above code snippet, PC-DMIS assigns the variables with these values:

```
V1 = 7.98
V2 = 8
```

Second Extension

The **Second Extension** drop-down list becomes available only if the item chosen in the **Extension** drop-down list requires the addition of a second extension in order to form a usable expression element.

Suppose you are referencing the nominal value of the X location axis of a dimension named "D1". You would:

1. Choose **D1** from the **ID** drop-down list.
2. Select **X** from the **Extension** drop-down list.
3. Select **Nom** from the **Second Extension** drop-down list.

Add Button

Whenever you select a usable or complete expression element from a list, the **Add** button becomes available. This button displays the text to be added to the expression.

For example, if you chose the following items for your expression:

- Dimensions from the **Expression Element Type** list
- D1 from the **ID** list
- X from the **Extension** list
- Nom from the **Second Extension** list

Then the **Add** button becomes enabled and has the following text: **Add D1.X.NOM**.

When you click the **Add** button, the text then appears in the edit box at the bottom of the dialog box.

When you click the **OK** button, PC-DMIS adds the text from the edit box to the expression field in the Edit window where your cursor is. If you select an item from the Edit window's expression field, and the text to be added contains parenthesis, then the selected item would be placed inside of the parenthesis of the added text.

Edit Box

At the bottom of the **Expression Builder** dialog box (**Edit | Expression**) is an edit box that shows the current expression. You can type the expression directly into this box, or you can use the **Add** button.

Description Area

The **Expression Builder** dialog box (**Edit | Expression**) also contains a **Description** area that gives information about items selected from the drop-down lists. A field next to the **Add** button also shows the current value of the expression.

Invalid expressions have a value of 0.

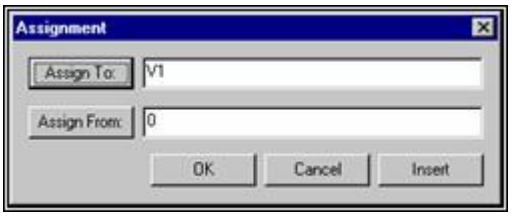
Using Variables with Expressions

Variables are objects that hold values. Variables refer to integer, real, string, or point operands. Variables are essential to using expressions. A variable has a name and a value. The name is used to access the value of the variable. The name is constant; the value can be changed. You can use the `ASSIGN/` command to assign a value to a variable.

For example, the statement `ASSIGN/V1=2` creates a variable with a name of V1 and a value of 2. `ASSIGN/V2=V1+2` accesses the value of V1. If V1 still had a value of 2 when this assign statement was executed, V2 would then have a value of 4.

For more information on variables, see "Variables".

Assigning Values to Variables by Using the Assignment Dialog Box



Assignment Dialog Box

The **Insert | Assignment** menu option displays the **Assignment** dialog box. Use this dialog box to assign a value to a variable or data element of a measurement routine feature, dimension, or alignment. Use of the assignment command requires a basic understanding of PC-DMIS expressions.

Assign To button - This button designates the variable that is receiving the value calculated in the **Assign From** box. Information chosen using the **Assign To** button is placed in the **Assign To** box. This value can be the name of a variable or a reference to a data element of a feature, dimension, or alignment.

The result of solving the mathematical expression for a value is what is meant by the term "evaluated".

Assign From button - This button places the value being assigned into the **Assign From** box. If this box contains an expression, the expression is evaluated at execution time and the result or value of the calculation is assigned to the object specified in the **Assign To** box.

Insert button - This button inserts an assignment command into the measurement routine while keeping the **Assignment** dialog box open. A series of assignment commands can be inserted without closing the dialog box.

Related Topics:

Understanding Expression Components

Accessing a Report's Object Properties

Understanding Expression Components

Expressions have these types of operands:

- Integers
- Real Numbers
- Strings

- Points
- Feature Pointers
- Arrays
- Functions

These are discussed in detail below.

Operand Types

The operands may exist in the form of:

- Literals
- References
- Variables
- Structures
- Pointers
- Arrays

Literals

***Integers:** 1, -6, 209

Reals: 1, -6, 2.4, -0.1, 345.6789

Strings: "Hello World", "47", "CIRCLE 1"

Points: A literal representation is not available for points. However, points can be made from other literals using the MPOINT function: For example, `MPOINT (0,0,1`), `MPOINT (2.2,3.1,4.0)` .

Pointer: The name of a feature enclosed in French brackets: `{CIR1},{LIN2},{F3}`

Arrays: A literal representation is not available for arrays. However, arrays can be created from other literals using the ARRAY function: For example, `ARRAY (3,5,6)` , `ARRAY ("Hello",2.3,9)` . These functions create 3 element arrays with the integer elements 3, 5, and 6 in the first example and the string element "Hello", double element 2.3, and integer element 9 in the second example.

Functions: A literal representation is not available for functions. Functions are defined using the `FUNCTION` keyword and accessed via variable ids. For example, `ASSIGN/ Add2= FUNCTION((X),X+2)` defines a function that takes one argument and adds 2 to

it. The variable Add2 is assigned the function. The function can be called using the variable Add2 as follows. `ASSIGN/Result=Add2(5)` . Result is assigned the value 7.

Numeric Literals are interpreted as real numbers unless the operator or function implies the use of integers. For example, the expression 10 / 8 evaluates to 1.25 instead of 1. Note also that discrete division is also possible via the operand coercion operators. The expression INT(10) / INT(8) does evaluate to 1.

References

References refer to data members of other objects in a measurement routine. References use the ID of an object in the measurement routine followed by a dot and an extension that refers to the data member of the object.

If CIRCLE1 were the name of a measured circle in the measurement routine, `CIRCLE1 .X` would refer to the measured value of the X component of CIRCLE1. All references are evaluated in part coordinates relative to the current alignment.

References of Type Double

The following reference expressions are available:

Valid Extensions for Feature References of Type Double by Example

Format: <Feature Id>.<Extension> -> `CIRCLE1 .X`

CIRCLE1.X Measured X value of CIRCLE1
CIRCLE1.Y Measured Y value of CIRCLE1
CIRCLE1.Z Measured Z value of CIRCLE1

CIRCLE1.TX Theoretical (Nominal) X value of CIRCLE1
CIRCLE1.TY Theoretical (Nominal) Y value of CIRCLE1
CIRCLE1.TZ Theoretical (Nominal) Z value of CIRCLE1

LINE1.SX Measured X value of the Startpoint of LINE1
LINE1.SY Measured Y value of the Startpoint of LINE1
LINE1.SZ Measured Z value of the Startpoint of LINE1

LINE1.TSX Theoretical X value of the Startpoint of LINE1
LINE1.TSY Theoretical Y value of the Startpoint of LINE1
LINE1.TSZ Theoretical Z value of the Startpoint of LINE1

LINE1.EX Measured X value of the Endpoint of LINE1
LINE1.EY Measured Y value of the Endpoint of LINE1
LINE1.EZ Measured Z value of the Endpoint of LINE1

LINE1.TEX Theoretical X value of the Endpoint of LINE1
LINE1.TEY Theoretical Y value of the Endpoint of LINE1
LINE1.TEZ Theoretical Z value of the Endpoint of LINE1

POINT.I Measured I Component of Vector for POINT
POINT.J Measured J Component of Vector for POINT
POINT.K Measured K Component of Vector for POINT

POINT.TI Theoretical I Component of Vector for POINT
POINT.TJ Theoretical J Component of Vector for POINT
POINT.TK Theoretical K Component of Vector for POINT

FEAT1.TYP The type of the feature (i.e. circle, slot, cone). You can use this to change the type of a generic feature (`ASSIGN/Gen1.TYP=Feat1.TYP`).

FEAT1.ALL Refers to all elements of the feature. This is valuable for copying information to a generic feature. (`ASSIGN/Gen1.ALL=Feat1.ALL`)

Surface Vector

EDGE.SURFI

EDGE.SURFJ

EDGE.SURFK

EDGE.TSURFI

EDGE.TSURFJ

EDGE.TSURFK

Angle Vector

CIR.ANGI

CIR.ANGJ

CIR.ANGK

CIR.TANGI

CIR.TANGJ

CIR.TANGK

Radius

CIRCLE1.R

CIRCLE1.TR

CIRCLE1.RAD

CIRCLE1.TRAD

CIRCLE1.RADIUS

CIRCLE1.PR – Polar Radius

CIRCLE1.TPR – Theoretical Polar Radius

CIRCLE1.TRADIUS (Only the First Characters are significant)

Diameter

CIRCLE1.D

CIRCLE1.TD

CIRCLE1.DIAM

CIRCLE1.TDIAM

CIRCLE1.DIAMETER

CIRCLE1.TDIAMETER (Only the first characters are significant)

Angle

CONE.A

CONE.TA

CONE.ANG

CONE.TANG

CONE.ANGLE

CONE.TANGLE

CONE.PA – Polar Angle

CONE.TPA – Theoretical Polar Angle (Only the first characters are significant)

Length

LINE.L

LINE.TL

LINE.LEN

LINE.TLEN

LINE.LENGTH

LINE.TLENGTH (Only the first characters are significant)

Height

CYLINDER.PH – Polar Height

CYLINDER.TPH – Theoretical Polar Height

Radius, Angle, Height

POINT.RAH – Point with Measured Radius, Angle, and Height

POINT.TRAH – Point with Theoretical Radius, Angle, and Height

Area

BLOB1.AREA - Returns the measured Area value for a Blob feature.

BLOB1.TAREA - Returns the Theoretical Area value for a Blob feature.

For example, `ASSIGN/V1=BLOB1 .AREA` returns the measured area value of the BLOB1 feature and assigns it to the V1 variable.

Currently, only the Blob feature works with these area extensions. For information on the Blob feature, see the "Vision Blob" topic in the "PC-DMIS Vision" documentation.

Valid Extensions for Dimension Reference of Type Double by Example

Format: <Dimension ID>.<AXIS>.<Dimension Element> -> `DIM1 .X.NOM`

DIM1.X.NOM	The Nominal Value for X Axis Location of DIM1
DIM1.X.MEAS	The Measured Value for the X Axis Location of DIM1
DIM1.X.MAX	The Max Deviation for X Axis Location of DIM1
DIM1.X.MIN	The Min Deviation for X Axis Location of DIM1
DIM1.X.PTOL	The Plus Tolerance Value for the X Axis Location of DIM1
DIM1.X.MTOL	The Minus Tolerance Value for the X Axis Location of DIM1
DIM1.X.DEV	The Deviation on the X Axis Location of DIM1
DIM1.X.OUTTOL	The Out of Tolerance value on the X Axis Location of DIM1
DIM1.Y.NOM	The Nominal Value for the Y Axis Location of DIM1
DIM1.Z.DEV	The Deviation on the Z Axis Location of DIM1
DIM3.PA.MEAS	The Measured Value for the Polar Angle Location of DIM3
DIM4.M.PTOL	The Plus Tolerance Value for the M Axis of DIM4
DIM4.PTOL	The Plus Tolerance Value for the M Axis of DIM4 (See *Note under "Valid Axes " below).
DIM5.BTOL	The Bonus Tolerance Value where DIM5 is a Position.

Valid Axes:

X, Y, Z, D, R, A, T, V, L, PR, PA, M, PD, RS, RT, S, H, DD, DF, TP

* Dimensions that have only one axis by definition (such as roundness, concentricity, and so on) can omit the axis qualifier. If you use the axis qualifier, note that all of these types of dimensions (that have only one axis) use the M Axis qualifier with the exception of 2D and 3D angle dimensions, which use the A Axis qualifier.

Valid Extensions for Alignment References of Type Double by Example:

Format: <Alignment ID>.<Alignment Axis or Origin>.<Alignment Axis or Origin Component> -> `A1 .ORIGIN.X`

A1.ORIGIN.X	X Component of alignment A1's measured origin
A2.ORIGIN.Y	Y Component of alignment A2's measured origin
A1.ORIGIN.Z	Z Component of alignment A1's measured origin
A1.XAXIS.I	I Component of alignment A1's measured X axis
A1.YAXIS.J	J Component of alignment A1's measured Y axis
A1.ZAXIS.K	K Component of alignment A1's measured Z axis
A1.CORIGIN.X	X Component of alignment A1's origin based on theoretical (C is for CAD) data
A1.CXAXIS.J	J Component of alignment A1's X axis based on theoretical (C is for CAD) data

References of Type Point

The following reference expressions are available:

Valid Extensions for Feature References of Type Point by Example

Format: <Feature ID>.<Extension> -> CIRCLE1.XYZ

`CIRCLE1.XYZ`	Measured centroid of CIRCLE1
`CIRCLE1.TXYZ`	Theoretical centroid of CIRCLE1
`LINE1.SXYZ`	Measured start point of LINE1
`LINE1.TSXYZ`	Theoretical start point of LINE1
`LINE1.EXYZ`	Measured end point of LINE1
`LINE1.TEXYZ`	Theoretical end point of LINE1
`CIRCLE1.IJK`	Measured Vector of CIRCLE1
`CIRCLE1.TIJK`	Theoretical Vector of CIRCLE1
`EDGE.SURFIJK`	Measured Surface Vector of EDGE
`EDGE.TSURFIJK`	Theoretical Surface Vector of EDGE
`AUTOCIR1.ANGIJK`	Measured Angle Vector of AUTOCIR1
`AUTOCIR1.TANGIJK`	Theoretical Angle Vector of AUTOCIR1

Valid Extensions for Alignment Reference of Type Point by Example

Format: <Alignment ID>.<Alignment Axis or Origin> -> A1.XAXIS

A1.ORIGIN	Measured origin of alignment A1
A1.XAXIS	Measured X axis of alignment A1
A1.YAXIS	Measured Y axis of alignment A1
A1.ZAXIS	Measured Z axis of alignment A1
A1.CORIGIN	Theoretical origin of alignment A1
A1.CXAXIS	Theoretical X axis of alignment A1
A1.CYAXIS	Theoretical Y axis of alignment A1
A1.CZAXIS	Theoretical Z axis of alignment A1

References of Type String

References to comments are the only object types that are of type string. Only INPUT comments or YES/NO comments can be referred to via references. These comment types have an ID that you can use to identify the comment.

Format: <Comment ID>.INPUT -> `C1 .INPUT`

C1.INPUT - The input value (from the operator) for comment C1

YES/NO comment types set the input to the appropriate yes or no string based on the current language of PC-DMIS. In the English version of PC-DMIS, if the operator presses the yes button, the string is set to "YES". If the operator presses the no button, the string is set to "NO". When comparing strings to test for "YES" or "NO", the comparison is case sensitive. Thus, comparing against "yes" or "no" always fails even if the YES/NO comment input is set to "YES" or "NO".

Variables

Variables can be of any of the seven operand types: integer, real, string, point, feature pointer, array, and function. Variables come into existence and receive their values and types via the `ASSIGN` statement.

The variable ID can be any alphanumeric string that does not begin with a numeric character. You can use underscores in the variable id as long as the underscore is not the first character.

As long as the measurement routine remains open, PC-DMIS saves variable values between execution runs. This means when execution finishes, PC-DMIS uses the values as the end of the routine when you restart the routine. You may or may not want this behavior. If you want fresh variable values, it's good practice to clear out your values with assign statements at the beginning of your routine. For example, if you use a V1 variable value in some number calculations, you can use `ASSIGN/V1=0` to clear out that variable.

If the Edit window is active, PC-DMIS indicates the current value of the variable whenever the cursor is placed in the field. During execution, variable values change based on execution flow. Position the mouse pointer over the desired variable to find out its current value.

```
ASSIGN/V1=2.2+2
```
Variable V1 is a real number with the value of 4.2

```
ASSIGN/VAR1=CIRCLE1. X
```
Variable VAR1 is a real number with a value equal to the measured value of CIRCLE1.X at the time of assignment.

```
ASSIGN/MYVAR=LINE1. XYZ
```
Variable MYVAR is a point with the same value of the measured centroid of LINE1 at the time of assignment.

```
ASSIGN/SVAR="Hello World"
```
Variable SVAR is a string with the value "Hello World"

In these examples, variables are assigned values. Once a variable is assigned a value, you can use the variable as an operand in any expression field.

Here, V1 is used in a numeric field. It is used as the prehit value of the prehit command:

```
ASSIGN/V1=1/3PREHIT/V1
```

Because expressions can be used in most fields that you can modify, the following expression is also legal and has the same effect: PREHIT / 1/3.

The components of variables of type point can be referred to individually using the dot extension notation used for references.

```
ASSIGN/V1= MPOINT (3,4,5)
```
V1 is of type point with value of 3, 4, 5

```
ASSIGN/XVAR=V1. X
```
XVAR is of type double with the value of 3

```
ASSIGN/YVAR=V1. Y
```
YVAR is of type double with the value of 4

```
ASSIGN/IVAR=V1. I
```
IVAR is of type double with the value of 3

```
ASSIGN/REDUNVAR=V1. XYZ
```
REDUNVAR is of type point with the value of 3, 4, 5

The following extensions are equivalent to each other. Both are provided to clarify the meaning of an expression in a measurement routine.

Given that V1 is of type point.

V1.X is the same as V1.I
V1.Y is the same as V1.J
V1.Z is the same as V1.K

V1.XYZ is the same as V1.IJK and V1 without any extension.

If a variable of type string has a string value equal to the name of the id of a feature, dimension, or alignment, you can use the variable as a reference object:

```
ASSIGN/V1="CIRCLE1"
```

The following operands are possible and valid provided a feature with the name CIRCLE1 exists.

V1.X - The measured X value of CIRCLE1
V1.TX - The theoretical X value of CIRCLE1
V1.Diameter - The measured diameter of CIRCLE1
V1.Radius - The measured radius of CIRCLE1

This type of indirection available on string variables is only available to one level of indirection. The following will not work.

```
ASSIGN/V1="CIRCLE1"
 ASSIGN/V2="V1"
```

V2.X - This evaluates to 0 instead of the current measured value of CIRCLE1.X.

The reference V2.X is *not* flagged as an error with red text even though an expression above it sets its type to a string. The reason it cannot be flagged as an error is because the flow of execution of the measurement routine is not known until execute time.

However, if you use curly brackets the following does work:

```
ASSIGN/V1={CIRCLE1} ASSIGN/V2={V1}
```

V2.X - This gives you the value of CIRCLE1.X.

Consider the following example:

```
ASSIGN/V1="CIRCLE1"
 ASSIGN/V2="V1" IF/CIRCLE1.X>CIRCLE1.TX,GOTO,L2
 L1=LABEL/ ASSIGN/V3=V2.X
 GOTO/LABEL,L3 L2=LABEL / ASSIGN/V2=MPOINT(2,5,7)
 GOTO/LABEL,L1 L3=LABEL/
```

During routine execution, if the value of CIRCLE1.X is greater than the value of CIRCLE1.TX, then the expression V2.X valid and evaluates to 2. Otherwise, the expression V2.X evaluates to 0 since the value of V2 at the time of the ASSIGN for V3 is the string "V1". It is the responsibility of the part programmer to ensure that expressions do as expected in these cases.

You can use almost all of the feature references on the left-hand side of the `ASSIGN` statement to put a value into a measured or theoretical data member of a feature. The only exception is the single I, J, K components of vectors. To assign to vectors, you must assign the complete vector at once with an expression that evaluates to a point. Vector data is normalized as it is input into the feature's vector data members.

```
ASSIGN/CIRCLE1.I=2-illegal
 ASSIGN/CIRCLE1.IJK=MPOINT(2,0,0)-legal(vector
 is normalized to 1,0,0)
```

For information on using variables within dimensions, see the "Dimensioning Variables" topic in the "Using Legacy Dimensions" chapter.

Structures

You can use a variable type called *structures* to place extensions on a variable to identify a sub-element of that variable. You can see that in this code snippet:

```
ASSIGN/V1.HEIGHT=6
 ASSIGN/V1.WIDTH=4.3
 ASSIGN/V1.MODE="CIRCULAR"
 ASSIGN/V1.POINT
 = MPOINT(100.3,37.5,63.1)
```

Where:

- `V1` is the structure
- `HEIGHT`, `WIDTH`, `MODE`, and `POINT` are sub-elements of the structure

Rules for Structures

- Like Variables, structures do not need to be declared.
- Sub-elements of a structure can be any of these variable types:
  - Integer
  - Double
  - Point
  - Feature Pointer
  - Function
  - Array
  - Structure

For example, it is possible to have structure elements that are arrays and array elements that are structures. You can see the valid expressions in the following code snippet:

```
 ASSIGN/
CAR.LEFTSIDE.DOOR[2].QUADRANT[3].JOINT[5].HIT[4]=MPOINT(558.89,910.12,

 COMMENT/OPER,"Current Z Position:
"+CAR.LEFTSIDE.DOOR[2].QUADRANT[3].JOINT[5].HIT[4].Z

 ASSIGN/CURRENTJOINT=LEFTSIDE.DOOR[2].QUADRANT[3].JOINT[5]

 COMMENT/OPER,"Next Hit: "+CURRENTJOINT.HIT[4]
```

Structures with Variables of Type Point

If a variable is of type point, you can still use the .X, .Y, .Z, .I, .J, and .K extensions to get at individual items of the point. You can also use any of the extensions from this example in their structures without being forced to use them as point elements as shown here:

```
ASSIGN/V1.X="Some
 string" ASSIGN/V1.Y=ARRAY(1,3,5,9,7)
 ASSIGN/V1.Z=MPOINT(3,5,7)
```

`COMMENT/REPT,V1.X`	Output is "Some string"
`COMMENT/REPT,V1.Y[2]`	Output is 3, the second element of the array.
`COMMENT/REPT,V1.Z.Y`	Output is 5, the Y value of the MPOINT.

By combining structures with the function capability of the PC-DMIS expression language, it is possible to have dynamic structure references as shown here:

```
ASSIGN/DYNAMICSTRUCT=FUNCTION((X,Y),X.Y)
 C1=COMMENT/INPUT,Please enter in item ASSIGN/
TESTSTR=C1.INPUT
 ASSIGN/FRONT=LEFT(TESTSTR,INDEX(TESTSTR,".")-1)
 ASSIGN/BACK=MID(TESTSTR,INDEX(TESTSTR,"."))
 ASSIGN/RESULT=DYNAMICSTRUCT(FRONT,BACK)
```

This portion of the example asks you to input a variable reference, splits the reference at the first '.', and then assigns `RESULT` to be equal to that reference by using the function `DYNAMICSTRUCT` .

So, if you had typed `V1.Y[4]` for the C1.INPUT variable, `RESULT` would end up with the value of 9 (the fourth element of the array assigned to `V1.Y)`.

The learn time evaluation of expressions has been enhanced to accurately show all elements of a structure or an array.

Pointers

Pointers are also known as "Feature Pointers". See the Glossary term "Feature Pointers" for more information.

Pointers provide a way to reference a feature via a variable or to pass objects using the call sub command. Pointers are similar to indirection via string names. However, the advantage of using pointers is with subroutines. Pointers, unlike strings, when passed in as arguments of a subroutine, allow for direct modification of the object pointed to by the subroutine. Pointers are not used in complex expressions. If it is used in a complex expression, the pointer evaluates to zero.

Consider the following examples.

Pointer Use Example:

In this example, `V1` is defined as a pointer that points to CIR1:

```
ASSIGN/V1={CIR1}
```

In this example, `DIST` is assigned the value of the distance of CIR1 from the origin:

```
ASSIGN/DIST=DOUBLE(V1.XYZ)
```

You can also put an expression between curly brackets in order to obtain a feature pointer. Now the following examples are all legal ways to get the pointer to feature CIR1:

```
ASSIGN/FEATCOUNT=1

ASSIGN/V1={"CIR"+FEATCOUNT}
```

Assigns expression "CIR1" to V1.

```
ASSIGN/V2="CIR1"

ASSIGN/V3={V2}
```

Assigns expression "CIR1" from variable V2 to variable V3.

```
C1=COMMENT/INPUT,Please type a feature name.

ASSIGN/V4={C1.INPUT}
```

This takes the feature name of C1.INPUT and places it into variable, V4.

Subroutine Example:

In the calling routine:

```
CS1=CALLSUB/SUB.PRG,CHANGEX,{CIR1}
```

In the subroutine:

```
GEN1=GENERIC/FEATURE

SUBROUTINE/CHANGEX,ARG1={GEN1}
```

(During execution, when the code passes CIR1 to the routine, CIR1 takes the place of GEN1)

```
ARG1.X=5
```

(Sets the measured X value of CIR1 to 5)

```
END/SUBROUTINE
```

Complex Expression Example:

```
ASSIGN/V1={CIR1}+2
```

{CIR1} evaluates to zero and so entire expression evaluates to 2.

Arrays

Three types of arrays are available: Feature, hit, and variable.

Even though multi-dimensional arrays are displayed as multi-dimensional in the software, you can really only use them as single dimensional arrays until you precede the arrays with an ARRAY INDICES command (see the "Array Indices Object:" topic).

Feature Arrays

When a feature is measured more than once during routine execution because of some kind of loop, the software creates a feature array automatically. The number of elements in the feature array is equal to the number of times that the feature has been executed.

If a measured circle feature is inside a while loop that executes five times, then an array of five measured circles exists. If the id of the measured circle is CIR1, then you can use an array expression to access individual instances of the measured circle object. You use square brackets to indicate the instance you want, like this:

```
ASSIGN/V1=CIR1[3].X
```

V1 is assigned the measured X value of the third instance of the CIR1 circle.

When a feature array exists for a given feature, but array notation is not used in a reference to that feature, the most recent instance is used. From the above example, the reference `CIR1.X` would be the same as `CIR1[5].X`, since the fifth instance would be the most recent instance of the object.

You can use expressions within the square brackets of an array expression:

`CIR1[3].` X and `CIR1[2+1].` X would therefore be equivalent.

This next example uses two While / End While loop command blocks. The first block executes the CIR1 circle five times. The second block uses the V1 variable inside of square brackets, like this, `CIR1[V1].XYZ`, to send the measured centroid of each of the five executions to the Report window:

```
            ASSIGN/V1=1
            WHILE/V1<6
CIR1         =FEAT/CIRCLE,CARTESIAN,IN,LEAST_SQR
              THEO/<40,30,-4.824>,<0,0,1>,30
              ACTL/
<40.002,29.991,-4.836>,<0,0,1>,29.982
              MEAS/CIRCLE,4,ZPLUS
                HIT/
BASIC,NORMAL,<41.984,44.868,-2.885>,<-0.132272,-0.9912
135,0>,<41.972,44.85,-2.891>,USE THEO=YES
                HIT/
BASIC,NORMAL,<51.721,39.36,-5.094>,<-0.781412,-0.62401
55,0>,<51.706,39.375,-5.107>,USE THEO=YES
                HIT/
BASIC,NORMAL,<54.792,32.491,-5.44>,<-0.9861119,-0.1660
821,0>,<54.775,32.474,-5.453>,USE THEO=YES
                HIT/
BASIC,NORMAL,<52.526,21.748,-5.879>,<-0.8350841,0.5501
223,0>,<52.537,21.764,-5.893>,USE THEO=YES
              ENDMEAS/
              ASSIGN/V1=V1+1
            END_WHILE/
            ASSIGN/V1=1
            WHILE/V1<6
              COMMENT/REPT,
              "Centroid of CIR1, instance #" + V1
              CIR1[V1].XYZ
              COMMENT/REPT,
              --------------------
              ASSIGN/V1=V1+1
            END_WHILE/
```

Here is the generated output to the Report window:

Pc	PART NAME : Top Holes - Concentric		May 23, 2022 15:25
	REV NUMBER : Rev1	SER NUMBER : 12345	STATS COUNT : 1

```
Centroid of CIR1, instance #1
<39.994, 30.016, -4.833>
--------------------
Centroid of CIR1, instance #2
<40.039, 30.011, -4.821>
--------------------
Centroid of CIR1, instance #3
<40.032, 30.013, -4.819>
--------------------
Centroid of CIR1, instance #4
<39.991, 30.013, -4.819>
--------------------
Centroid of CIR1, instance #5
<40.016, 30.003, -4.83>
--------------------
```

Arrays also exist on dimensions and alignments that have been executed multiple times in a given execution run. Thus, `Dim1[2].` Nom and `Align1[4].` Origin would be available given that the Dimension "Dim1" has executed at least twice and the alignment "Align1" has executed at least four times.

If a feature array reference is out of bounds (for example, the user asks for `CIR1[2.5]` or `CIR1["Hello, World"]`) the upper or lower bound item is returned. If `CIR1` had 3 instances, then `CIR1[4]` and above would return the value for `CIR1[3],` and `CIR1[0]` and below would return the value for `CIR1[1]` . All expressions between square brackets are coerced to integer, thus 2.5 would become 2 and "Hello World" would become 0.

Array Indices Object

By default, feature arrays are always one-dimensional arrays. If it is more convenient to treat a feature array as a multi-dimensional array, you can do this with an array indices object.

The array indices object allows you to specify upper and lower bounds for multiple array dimensions.

- When you set the upper and lower bounds of the first dimension, PC-DMIS creates a two dimensional array where the first dimension is bound and the second dimension is unbound.
- When you set the upper and lower bounds of the first two dimensions of an array, PC-DMIS creates a three dimensional array. The last dimension is always unbound.

Suppose feature F1 is located inside a nested WHILE loop. The inner WHILE loop executes five times and the outer WHILE loop executes three times. At completion of execution, F1 has been executed 15 times and so 15 instances of F1 exist.

Consider the following example measurement routine segment:

```
ARRAY_INDICES/1..5,..

ASSIGN/V1=1

WHILE/V1<=3

     ASSIGN/V2=1

     WHILE/V2<=5

          F1=FEAT/POINT,RECT

          THEO/V2,V1,0,0,0,1

          ACTL/1,1,0,0,0,1

          MEAS/POINT,1

          HIT/BASIC,V2,V1,0,0,0,1,1,1,0

          ENDMEAS/

          ASSIGN/V2=V2+1

          COMMENT/REPT,"Location of
         F1["+V2+","+V1+"] :"+F1[V2,V1].XYZ

     END_WHILE/

     ASSIGN/V1=V1+1

END_WHILE/
```

This code segment creates a 3 X 5 grid of 15 measured points.

The array indices command has limited the first dimension of the feature array to be between 1 and 5 inclusively. Thus on the inspection report, instead of appearing as F1[1] – F1[15], the objects appear as F1[1, 1] – F1[5, 3], more consistent with the layout of the features. Notice that the comment also refers to the feature array using a two dimensional array syntax.

To insert an array indices object in a measurement routine:

1. Using the keyboard, type "**Array**" on an empty line in the Edit window.
2. Press the Tab key on your keyboard.

If you clear the **Display Brackets for Feature Arrays** check box, the feature will not appear with the bracketed name. See the "Display Brackets for Feature Arrays" description in the "Setup Options ID Setup Tab" topic of the "Setting Your Preferences" chapter.

Hit Arrays

The hits of a given feature are available as an array and can be accessed via expressions using array syntax of the form <FeatID>.Hit[<Array Expression>].<Extension> or the form <FeatID>.RawHit[<Array Expression>].<Extension>. Hit returns probe compensated data when probe compensation is on. RawHit always returns uncompensated data. Valid extensions are X, Y, Z, I, J, K, TX, TY, TZ, TI, TJ, TK, XYZ, TXYZ, IJK, and TIJK

```
Circle1. Hit [1]. XYZ
```
The measured centroid (probe compensated) of hit 1 of "Circle1".

```
Circle1. Hit [2]. IJK
```
The measured vector of hit 2 of "Circle1"

Hit data is available for all objects that have hits, whether the actual hits are displayed in the Edit window or not. Thus, hits can be obtained for scans and Auto features.

Using Hit Arrays to Define Constructed Feature Inputs

You can use Hit arrays to define the inputs for constructed features.

For constructed features, when you use the feature.HIT[start..end] or feature.HITS[start..end] methods, the start and end properties are optional:

- If the start value is not defined, PC-DMIS assumes it to be the value of "1".
- If the end value is not defined, PC-DMIS assumes it to be the total number of hits for the feature.

This is the same if you entered "feature.NUMHITS".

These examples show three different ways to use the same hits from CIR1:

Example 1

In this example, the start and end values are explicitly defined:

```
CIR3    =FEAT/CIRCLE,CARTESIAN,IN,NO
         THEO/
<20.97629,22.90352,0>,<0,0,-1>,20.97629
         ACTL/
<20.96578,22.89023,-0.01243>,<0,0,-1>,20.96578
         CONSTR/
CIRCLE,REV,CIR1.HIT[1..CIR1.NUMHITS]
```

Example 2

In this example, only the start value of "1" is defined. Since the end value is not defined, PC-DMIS defines the end value as "CIR1.NUMHITS":

```
CIR3    =FEAT/CIRCLE,CARTESIAN,IN,NO
         THEO/
<20.97629,22.90352,0>,<0,0,-1>,20.97629
         ACTL/
<20.96578,22.89023,-0.01243>,<0,0,-1>,20.96578
         CONSTR/CIRCLE,REV,CIR1.HIT[1..]
```

Example 3

In this example, neither the start nor end values are defined. Because of this, PC-DMIS defines the start value as "1" and the end value as "CIR1.NUMHITS":

```
CIR3    =FEAT/CIRCLE,CARTESIAN,IN,NO
         THEO/
<20.97629,22.90352,0>,<0,0,-1>,20.97629
         ACTL/
<20.96578,22.89023,-0.01243>,<0,0,-1>,20.96578
         CONSTR/CIRCLE,REV,CIR1.HIT[..]
```

The following sections describe some additional array functions that are useful for finding the minimum or maximum points in a scan:

Assigning a Range of Hits to an Array

You can also assign a range of hits to an array by using this syntax:

<Feature Id>.<Hittype>[<Startnum>..<Endnum>].<Extension>

where

<Feature Id> is the name of the feature.

<Hittype> can be either the word "HIT" for compensated data or "RAWHIT" for uncompensated data. If probe compensation is turned off, the returned values are always uncompensated.

<Startnum> is an expression that identifies the first index value of the range of hits.

<Endnum> is an expression that identifies the second index value of the range of hits.

<Extension> identifies the type of data. Possible extensions include the measured or theoretical data types listed in the next section.

Example of how to use implied Start and End values in an expression.

Suppose you want to find all the hit points that define a plane (PLN1). You can write it out this way:

```
PLN1.HIT[1..PLN1.NUMHITS]
```

A shorter way to do this is to use inferred Start and End hit values. You can write the same code this way:

```
PLN1.HIT[..]
```

In this case, since we did not define the Start and End hit values, PC-DMIS assumes the Start hit value to be 1 and the End hit value to be NUMHITS.

Measured

- X – Measured X values of the hits
- Y – Measured Y values of the hits
- Z – Measured Z values of the hits
- XYZ – Measured XYZ values of the hits
- I – Measured I values of the hits
- J – Measured J values of the hits
- K – Measured K values of the hits

- IJK – Measured IJK values of the hits

Theoretical

- TX – Theoretical X values of the hits
- TY – Theoretical Y values of the hits
- TZ – Theoretical Z values of the hits
- TXYZ – Theoretical XYZ values of the hits
- TI – Theoretical I values of the hits
- TJ – Theoretical J values of the hits
- TK – Theoretical K values of the hits
- TIJK – Theoretical IJK values of the hits

For example:

```
ASSIGN/V1=SCAN1. HIT [1..10]. X
```
V1 is assigned to an array of 10 values, which are the measured X values from the first 10 hits of SCAN1.

```
ASSIGN/V2=SCAN1. HIT [1..SCAN1.NUMHITS]. XYZ
```
V2 is assigned to an array of points from each of the centroids of the hits in the scan.

Sorting Arrays

PC-DMIS allows you to sort arrays in either ascending or descending order. The following two expressions take an array and return a sorted array:

To sort in *ascending* order use:
```
SORTUP (<array>)
```

To sort in *descending* order use:
```
SORTDOWN (<array>)
```

For example:

```
ASSIGN/V1= ARRAY (5,8,3,9,2,6,1,7)
```
V1 is assigned the array of "5,8,3,9,2,6,1,7".

```
ASSIGN/V2= SORTUP (V1)
```
V2 will hold the array values sorted in ascending order: "1,2,3,5,6,7,8,9".

```
ASSIGN/V3= SORTDOWN (V1)
```
V3 will hold the array values sorted in descending order: "9,8,7,6,5,3,2,1".

Returning the Greatest or Least Index Values from an Array

You can input an array into a function and return the index number of the element that has the greatest or the least value by using these functions:

To return the index value of the element with the *greatest* value, use:

```
MAXINDEX (<array>)
```

To return the index value of the element with the *least* value, use:

```
MININDEX (<array>)
```

For example,

```
ASSIGN/V1= ARRAY (5,8,3,9,2,6,1,7)
```
V1 is assigned the array of "5,8,3,9,2,6,1,7".

```
ASSIGN/V2= MAXINDEX (V1)
```
V2 will hold the array's index value of 4. The actual value of that array element is 9.

```
ASSIGN/V3= MININDEX (V1)
```
V3 will hold the array's index value of 7. The actual value of that array element is 1.

You can then use returned index values to get the actual array element value.

Returning Sorted Index Values from an Array

You can input an array into a function, sort the array's values in ascending or descending order, and then return the index values by using these functions:

To return the array's index positions in order of their values sorted from greatest to least use:

```
MAXINDICES (<array>)
```

To return the array's index positions in order of their values sorted from least to greatest use:

```
MININDICES (<array>)
```

For example:

```
ASSIGN/V1= ARRAY (4,8,2,9,5,7)
```
V1 is assigned the array of "4,8,2,9,5,7".

```
ASSIGN/V2= MAXINDICES (V1)
```
V2 will hold an array with these values: "4,2,6,5,1,3".

```
ASSIGN/V3= MININDICES (V1)
```
V3 will hold an array with these values: "3,1,5,6,2,4".

Example of Using Array Functions to Find the Minimum and Maximum Points in a Scan

The main purpose of the hit array functions discussed above is to give you an easy way to find the minimum and maximum points in a scan.

To dimension the point from SCAN1 that has the greatest measured X value, you could use this expression:

```
ASSIGN/MAXPTINDEX=MAXINDEX(SCAN1.HIT[1..SCAN1.NUMHITS].X)

D1=LOCATION OF FEATURE SCAN1.HIT[MAXPTINDEX]
```

To find the three highest points in the Z axis of SCAN2, you could use this expression:

```
ASSIGN/MI=MAXINDICES(SCAN2.HIT[1..SCAN2.NUMHITS].Z)

ASSIGN/
THREEPOINTS=ARRAY(SCAN2.HIT[MI[1]].XYZ,SCAN2.HIT[MI[2]].XYZ
,SCAN2.HIT[MI[3]].XYZ)
```

Variable Arrays

Variable arrays do not need to be declared. Variable arrays come into existence via the assign statement when the expression on the right hand side of the assign statement evaluates to an array or when the left hand side of the assign statement refers to an element in a variable array.

`ASSIGN/V1=Array(3,4,5,6,7)`
Creates 5 element array and assigns it to V1.

`ASSIGN/V2=V1[3]`
Assigns V2 the value of the third element in array V1: 5.

`ASSIGN/V1[4]=23`
Assigns 4th element of array V1 to 23.

Arrays are created and allocated dynamically. Thus an array can be created by using an array reference on the left hand side of an assign statement.

`ASSIGN/V3[5]=8`
Dynamically creates array with 5th element set equal to 8.

When referencing an array element that has never received a value, the array expression will evaluate to 0.

```
ASSIGN/V3[5]=8
```

```
ASSIGN/V4=V3[5]
```
V4 is set equal to the value 8.

```
ASSIGN/V5=V3[6]
```
If the sixth element of V3 has never been set, V5 is set equal to 0.

Like other array types, expressions can be used within the square brackets.

```
ASSIGN/V3[5]=8
```

```
ASSIGN/V4=V3[2+3]
```
V4 is set equal to the value 8.

Variable arrays can have multiple dimensions.

```
ASSIGN/V6= Array ( Array (4,7,2), Array (9,2,6))
```
V6 is set to a 2 by 3 dimensional array where V6[1, 1] equals 4, V6[1, 2] equals 7, V6[1, 3] equals 2, V6[2, 1] equals 9, V6[2,2] equals 2, and V6[2,3] equals 6.

```
ASSIGN/V7=V6[2,1]
```
V7 is set to the value 9.

Variable arrays can have negative indexes:

```
ASSIGN/V8[-3]=5
```
The –3rd index of array V8 is set to 5.

Array assignment will overwrite previous values:

```
ASSIGN/V8="Hello"
```
The variable V8 is equal to the string "Hello"

```
ASSIGN/V8[2]=5
```
V8 is no longer of type string, but of type array, the second element of which has a value of 5.

```
ASSIGN/V8=9
```
V8 is no longer an array, but an integer of value 9.

Arrays can be made up of multiple types:

```
ASSIGN/V9= Array ("Hello",3,2.9,{FEAT1})
```
Creates array V9 with four elements. The first element is a string, the second

element is an integer, the third element is a real number, and the fourth element is a pointer to FEAT1.

Arrays can be increased in size to include more elements:

```
ASSIGN/V10=ARRAY(3,1,5)
```

```
ASSIGN/V10[ LEN (V10)+1]=7
```

The first statement creates an initial array V10 with three elements (3,1, and 5). The second statement then grows the array in V10 by one element and gives the final element a value of 7.

Operators for Expressions

The following basic operators are available inside PC-DMIS:

+ Addition: <*Expression*> + <*Expression*>

Adds the two expressions together. In the case of strings, strings are concatenated.

- Subtraction: <*Expression*> - <*Expression*>

Subtracts the second expression from the first expression.

***** Multiplication: <*Expression*> * <*Expression*>

Multiplies the two expressions.

/ Division: <*Expression*> / <*Expression*>

Divides first expression by the second expression.

^ Exponentiation: <*Expression*> ^ <*Expression*>

Raises the first expression to the power of the second expression.

% Modulo: <*Expression*> % <Expression>

Returns the remainder of one expression divided by the other.

- Additive Inverse -<*Expression*>

Returns the additive inverse of the expression.

! Logical Not: *!*<*Expression*>

Returns the logical not of the expression.

== Equal To: <*Expression*> == <*Expression*>

Evaluates to 1 if expressions are equal. Otherwise, it evaluates to 0. (Two equal signs are used to distinguish from the assignment operator = in the assignment statement).

<> Not Equal To*:* <*Expression*> <> <*Expression*>

Evaluates to 1 if expression are not equal. Otherwise, it evaluates to 0.

> Greater Than*:* <*Expression*> > <*Expression*>

Evaluates to 1 if first expression is greater than second expression. Otherwise, it evaluates to 0.

>= Greater Than or Equal To:<*Expression*> >= <*Expression*>

Evaluates to 1 if the first expression is greater than or equal to the second expression. Otherwise, it evaluates to 0.

< Less Than: <*Expression*> < <*Expression*>

Evaluates to 1 if first expression is less than the second expression. Otherwise, it evaluates to 0.

<= Less Than or Equal To: <*Expression*> <= <*Expression*>

Evaluates to 1 if the first expression is less than or equal to the second expression. Otherwise, it evaluates to 0.

AND Logical And: <*Expression*> *AND* <*Expression*>

Evaluates to 1 if both expressions does not evaluate to 0. Otherwise, it evaluates to 0.

OR Logical Or: <*Expression*> *OR* <*Expression*>

Evaluates to 1 if either expressions does not evaluate to 0. Otherwise, it evaluates to 0.

() Parenthesis: *(* <*Expression*> *)*

Gives evaluation to precedence to expression inside of parenthesis.

Precedence

Expressions are evaluated with the precedence shown below (listed from highest precedence to lowest precedence).

Highest Precedence

- Operands
- (unary minus), !, (), functions (such as ABS, COS, STR, LEN, CROSS, and so on.)

- ^
- *, /, %
- +, -
- ==, <>, <, <=, >, >=
- AND
- OR

Lowest Precedence

Functions

Functions are PC-DMIS specific expressions or user-defined expressions that usually take parameters and then return results. The parameters are substituted into the expression before the expression is evaluated.

Functions List

The following alphabetical list contains all the functions available to PC-DMIS's expression language.

- ABS (mathematical)
- ACOS (mathematical)
- ANGLEBETWEEN (point)
- ARCSEGMENTENDINDEX (miscellaneous)
- ARCSEGMENTSTARTINDEX (miscellaneous)
- ARRAY (array)
- ASIN (mathematical)
- ATAN (mathematical)
- CHR (string)
- CONCAT (string)
- COS (mathematical)
- CROSS (point)

- DEG2RAD (mathematical)
- DELTA (point)
- DIST2D (pointer)
- DIST3D (pointer)
- DOT (point)
- ELEMENT (string)
- EOF (miscellaneous)
- EOL (miscellaneous)
- EQUAL (array)
- EQUAL (string)
- EXP (mathematical)
- FORMAT (string)
- FUNCTION (function)
- GETCOMMAND (pointer)
- GETPROGRAMINFO (string)
- GETROTABDATA (miscellaneous)
- GETSETTING (string)
- GETTEXT (string)
- GETTRACEVALUE (string)
- IF (miscellaneous)
- INDEX (string)
- ISIOCHANNELSET (miscellaneous)
- LEFT (string)
- LEN (array)
- LEN (pointer)
- LEN (string)
- LINESEGMENTENDINDEX (miscellaneous)

- LINESEGMENTSTARTINDEX (miscellaneous)
- LN (mathematical)
- LOG (mathematical)
- LOWERCASE (string)
- MAX (array)
- MID (string)
- MIN (array)
- MPOINT (point)
- ORD (string)
- PCDMISAPPLICATIONPATH (string)
- PCDMISUSERHIDDENDATAPATH (string)
- PCDMISUSERVISIBLEDATAPATH (string)
- PCDMISSYSTEMHIDDENDATAPATH (string)
- PCDMISSYSTEMVISIBLEDATAPATH (string)
- PCDMISSYSTEMREPORTINGPATH (string)
- PROBEDATA (miscellaneous)
- QUALTOOLDATA (miscellaneous)
- RAD2DEG (mathematical)
- RIGHT (string)
- ROUND (mathematical)
- SETROTABDATA (miscellaneous)
- SIN (mathematical)
- SQRT (mathematical)
- SYSTEMDATE (string)
- SYSTEMTIME (string)
- SYSTIME (string)
- TAN (mathematical)

- TUTORELEMENT (miscellaneous)
- UNIT (point)
- UPPERCASE (string)

String Functions

The following functions are used with text strings.

CHR

Character Conversion: *CHR(<Integer>)*

This function returns a string, which consists of the character corresponding to the ASCII decimal value.

CONCAT

This function concatenates all of the strings specified in expressions 1 through N together into one string: `CONCAT (<expression1>, <expression2>, & <expressionN>)`

ELAPSEDEXECUTIONTIME

Formatted Elapsed Execution Time: *ELAPSEDEXECUTIONTIME()*

This function returns the time that has elapsed since the measurement routine or mini routine started to execute. The elapsed execution time is the time spent during the DCC portion of the execution; it does not track time for pauses due to the attention required by the user. Such pauses include execution pauses during comment execution or PC-DMIS messages, and error messages that may stop the execution altogether.

You can record the elapsed execution time at any point in the measurement routine or mini routine by assigning the function to a variable, such as:

```
ASSIGN/V1=ELAPSEDEXECUTIONTIME()
```

The default format of the returned time is "hh:mm:ss". You can measure the elapsed execution time in other formats as well:

- Use `ASSIGN/V1=FORMAT(ELAPSEDEXECUTIONTIME(),"hh:mm:ss")` or `ASSIGN/V1=ELAPSEDEXECUTIONTIME()` to get the time in hours, minutes, and seconds.
- Use `ASSIGN/V1=FORMAT(ELAPSEDEXECUTIONTIME(),"mm:ss")` to get the time in minutes and seconds.
- Use `ASSIGN/V1=FORMAT(ELAPSEDEXECUTIONTIME(),"ss")` to get the time in seconds.

ELEMENT

Delimited substring location: *ELEMENT(<Integer>, <String1>, <String2>)*

This function returns the nth substring (element) from string2 using string1 as the delimiting text that divides the elements in string2.

Suppose string2 is "6, 12, 8, 4, 5" and string1 is a comma character ",". The five elements that can be individually retrieved with the element command are "6", "12", "8", "4", and "5".

EQUAL

Case-insensitive string comparison: *EQUAL(<String1>, <String2>)*

This function compares two strings (ignoring the case) to determine if they are identical. It returns an integer set to 1 if the strings are the same, and 0 if they are not.

FORMAT

Format: FORMAT(<String>,<Integer,double, or point>)

This function takes two expressions and returns a formatted string similar to using the *sprintf* function inside C++.

- Expression 1 should be a *string* type that contains one or three format specifiers. If it is a different type, the expression evaluator attempts to coerce it to a string. The string should contain *one* format specifier if Expression 2 is an integer or double types and *three* format specifiers (see paragraphs below) if Expression 2 is a point type.
- Expression 2 is expected to be of type *integer*, *double*, or *point*. If a different type is used, the value of the expression is 0.

Format Specifier for FORMAT Function:

The format specifier should have the same syntax as a format specifier used in the *sprintf* function used in the C++ programming language.

A format specifier consists of optional and required fields and has the following syntax:

%[flags] [width] [.precision] type

Each field of the format specifier is either a single character or a number that signifies a particular format option. The simplest format specifier uses only the percent sign and a type character (for example, %d). If a percent sign is followed by a character that has no meaning as a format field, the character is copied to STDOUT. For example, to print a percent-sign character, use %%.

The optional flag, width, and precision fields, which appear before the type character, control other aspects of the formatting. These are described below:

flags

These *optional characters* control output justification and the printing of signs, blanks, decimal points, and octal or hexadecimal prefixes. More than one flag can appear in a format specifier.

Here are the possible flags:

–

Meaning: Left align the result within the given field width.

Default: Right align.

+

Meaning: Prefix the output value with a sign (+ or –) if the output value is of a signed type.

Default: Sign appears only for negative signed values (–).

0

Meaning: If the width is prefixed with 0, zeros are added until the minimum width is reached. If 0 and – appear, the 0 is ignored. If 0 is specified with an integer format (i, u, x, X, o, d) the 0 is ignored.

Default: No padding.

blank (' ')

Meaning: Prefix the output value with a blank if the output value is signed and positive; the blank is ignored if both the blank and + flags appear.

Default: No blank appears.

#

Meaning 1: When used with the o, x, or X type, the # flag prefixes any nonzero output value with 0, 0x, or 0X, respectively.

Default 1: No prefix appears.

Meaning 2: When used with the e, E, or f type, the # flag forces the output value to contain a decimal point in all cases.

Default 2: Decimal point appears only if digits follow it.

Meaning 3: When used with the g or G format, the # flag forces the output value to contain a decimal point in all cases and prevents the truncation of trailing zeros.

Default 3: Decimal point appears only if digits follow it. Trailing zeros are truncated. This is ignored when used with d, i, or u.

width

This second optional field, or argument, controls the minimum number of characters printed. It is a non-negative decimal integer.

- If the number of characters in the output value is less than the specified width, blanks are added to the left or the right of the values — depending on whether the – flag (for left alignment) is specified — until the minimum width is reached.
- If the width is prefixed with 0, zeros are added until the minimum width is reached (not useful for left-aligned numbers).
- The width specification never causes a value to be truncated. If the number of characters in the output value is greater than the specified width, or if the width is not given, all characters of the value are printed (subject to the precision specification listed below).

precision

This third optional field, or argument, specifies the number of characters to be printed, the number of decimal places, or the number of significant digits. Unlike the width specification, the precision specification can cause either truncation of the output value or rounding of a floating-point value. It is a non-negative decimal integer preceded by a period (.).

type

This required character determines whether the associated argument is an integer, a double, or a point. The list of available types includes:

d - signed decimal integer

i - signed decimal integer

o - unsigned octal integer

u - unsigned decimal integer

x - unsigned hexadecimal integer, using "abcdef"

X - unsigned hexadecimal integer, using "ABCDEF"

e - double in exponential form [-]d.dddd e [sign]ddd

E - same as e, except uses E to introduce the exponent

f - double with the form [-]dddd.dddd

g - formats to either the e or f format depending on which is more compact

G - same as g, except uses E to introduce the exponent

FORMAT Example

This example shows several statements using the FORMAT function inside a measurement routine:

`ASSIGN/ V1=PROBEDATA("OFFSET")`	V1 becomes type point representing the offsets of the current probe. Using the values from the measurement routine used for this example, V1 becomes: <-1.8898, 1.8898, 5.704>
`ASSIGN/V3=FORMAT("%.5f, %.5f,%.5f",V1)`	V3 becomes type string. The string is formatted using the point object of variable V1. V3 now has: -1.88976, 1.88976, 5.70403
`ASSIGN/V4=1.123456789`	V4 becomes type double.
`ASSIGN/V5=FORMAT("%.5f ",V4)+FORMAT("%.6f ",V4)+FORMAT("%.7f ",V4)+FORMAT("%.8f",V4)`	V5 becomes type string with this value: 1.12346 1.123457 1.1234568 1.12345679
`ASSIGN/V6A="The value of V4 is: "+FORMAT("%.8f",V4)`	V6A becomes type string with value of: The value of V4 is: 1.12345679
`ASSIGN/V6B=FORMAT("The value of V4 is: %.8f",V4)`	The expression result remains the same as V6A above.
`ASSIGN/V7=4444`	V7 becomes type double since all numbers are assumed double unless coerced to an integer.
`ASSIGN/ V8=FORMAT("%o",INT(V7))`	V8 becomes type string with this value: 10534
`ASSIGN/ V9=FORMAT("%u",INT(-1))`	V9 becomes type string with this value: 4294967295

`ASSIGN/ V10=FORMAT("%x",INT (2143))`	V10 becomes type string with this value: 85f
`ASSIGN/ V11=FORMAT("%X",INT (9567))`	V11 becomes type string with this value: 255F
`ASSIGN/ V12=FORMAT ("%e",0.0005432)`	V12 becomes type string with this value: 5.432000e-004
`ASSIGN/ V13=FORMAT ("%E",145.3421)`	V13 becomes type string with this value: 1.453421E+002
`ASSIGN/V14=FORMAT(", %6d,",INT(1))`	`V14 becomes type string with this value: ,     1,`
`ASSIGN/V15=FORMAT(", %-6d,",INT(1))`	`V15 becomes type string with this value: ,1     ,`

GETSETTING

This function allows you to return various settings of PC-DMIS based on the string parameter inserted.

GETSETTING(<*String*>)

You can use these string parameters:

- "DCC Mode" – Returns a 1 if PC-DMIS is in DCC Mode, 0 otherwise.
- "Manual Mode" – Returns a 1 if PC-DMIS is in Manual Mode, 0 otherwise.
- "Current Alignment" – Returns a string of the current alignment.
- "Current Workplane" – Returns a string of the current workplane.
- "Workplane Value" – Returns a numerical value of the current workplane.

- "PreHit" – Returns the current prehit value as a double precision number.
- "Retract" – Returns the current retract value as a double precision number.
- "Check" – Returns the current check value as a double precision number.
- "Touch Speed" – Returns the current Touch Speed value as a double precision number.
- "Move Speed" – Returns the current Move Speed value as a double precision number.
- "Fly Mode" – Returns a 1 if PC-DMIS uses the Fly Mode, 0 otherwise.
- "Ph9 present" – Returns a 1 if the Ph9/Ph10 is present, 0 otherwise.
- "Manual CMM" – Returns 1 if the CMM is a manual CMM, 0 otherwise.
- "LangStr(<Number or ID>)" – Returns a string from PC-DMIS's resources in the current language from a resource ID number or from one of these IDs:

  "Yes", "No", "Oper", "Rept", "Input", "Doc", "YesNo", "Readout", "Internal", "External", "Rect ", "Polr ", "Out", "In", "Least_Sqr", "Min_Sep", "Max_Insc", "Min_CircSc", "Fixed_Rad", "Workplane", "Xaxis", "YAxis", "ZAxis", "Xplus", "Xminus", "YPlus", "YMinus", "ZPlus", "ZMinus", "Point", "Plane", "Line", "Circle", "Sphere", "Cylinder", "Round_Slot", "Square_slot", "Cone", or "None".

  If the value you use is a positive number, PC-DMIS pulls the string from its resource.dll file. If you use a negative number, PC-DMIS pulls the string from its strings.dll file (the strings table).
- "Extended Sheet Metal" – Returns a 1 if the **Show Extended Sheet Metal Options** check box is selected inside the **Setup Options** dialog box, 0 otherwise.
- "LastHitMove(X)" – Returns the X value of the most recent HIT /BASIC or MOVE/POINT command. PC-DMIS must be in DCC Mode for this to work.
- "LastHitMove(Y)" – Returns the Y value of the most recent HIT/BASIC or MOVE/POINT command. PC-DMIS must be in DCC Mode for this parameter to work.
- "LastHitMove(Z)" – Returns the Z value of the most recent HIT/BASIC or MOVE/POINT command. PC-DMIS must be in DCC Mode for this parameter to work.

You can use the GETSETTING function to determine whether PC-DMIS is in Manual Mode or DCC Mode as shown below:

`ASSIGN/DCCMODEVAR=GETSETTING("DCC Mode")` - This gives the variable DCCMODEVAR the value of 1 if PC-DMIS is in DCC Mode, otherwise 0.

`ASSIGN/MANMODEVAR=GETSETTING("Manual Mode")` - This gives the variable MANMODEVAR the value of 1 if PC-DMIS is in Manual Mode, otherwise 0.

You can use the GETSETTING function to determine the current workplane as shown below:

`ASSIGN/WORKPLANE_ID=GETSETTING("Current Workplane")` - This gives the variable WORKPLANE_ID the string value of the current workplane (ZPLUS, ZMINUS etc.).

`ASSIGN/WORKPLANE_VALUE=GETSETTING("Workplane Value")` - This gives the variable WORKPLANE_VALUE a numerical value representing the workplane. The workplanes have these values associated with them: ZPLUS = 0, ZMINUS = 3, XPLUS = 1, XMINUS = 4, YPLUS = 2, or YMINUS = 5.

GETTEXT

This function returns the current text from the specified data field:

`GETTEXT (<String or Integer>,<Integer>,<Pointer>)`

This function has three fields.

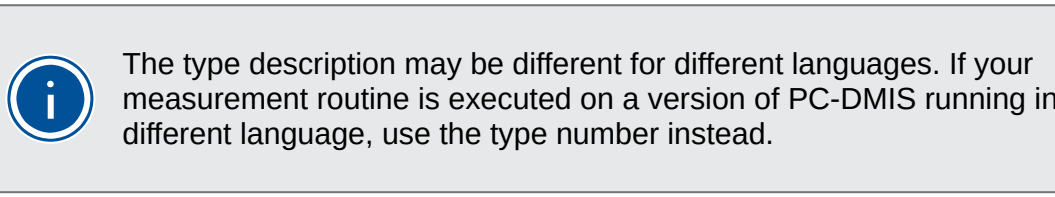
First Field—Data Field Number or Description

The first field can be either a string description of the data field, indicated by item (A) in the image below or the data field number, indicated in item (C) in the image below.

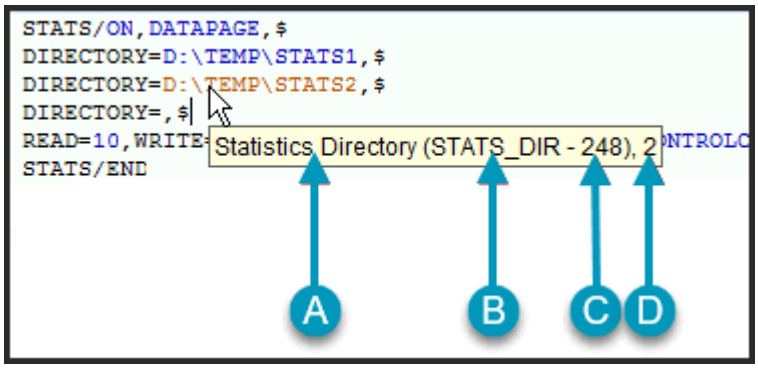
Item (B) in the image below is not used in this function but is sometimes used in automation or in report expressions.

To obtain these values:

1. Place PC-DMIS in Command Mode. Right-click anywhere in the Edit window. A shortcut menu appears.
2. From the shortcut menu, select **Change Pop-up Display** and then **Data Type Information**.
3. Position the mouse over a data field in the Edit window. The type description, type number, and type index for that data item display.



The type description may be different for different languages. If your measurement routine is executed on a version of PC-DMIS running in a different language, use the type number instead.



Sample Data Type Information Showing (A) Type Description, (B) Type String Identifier, (C) Type Number, and (D) Type Index

Second Field—Type Index

The second field is the type index, indicated as (D) in the above image. This field is usually zero unless you have more instances of the same type of field in the same command, such as multiple DIRECTORY fields shown in the above image. The correct value for this field can be obtained in the same manner as described for the first field.

Third Field—Command Pointer

The third field is a command pointer. It points to the command containing the field from which the text is being obtained. You can use the command pointer notation (i.e., {F15}) to specify this field, or you can use the GETCOMMAND expression as shown below:

`ASSIGN/V1=GETTEXT("Best Fit Math Type",0,{F15})` - This command assigns V1 the current value of the best fit math type toggle of feature F15.

`ASSIGN/V2=GETCOMMAND("Comment","TOP",1)` - V2 is assigned a pointer to the first comment from the top of the measurement routine.

`ASSIGN/V3=GETTEXT("Comment Type",1,V2)` - V3 is assigned the value of the Comment Type toggle field. If the first comment in the measurement routine is a comment to be displayed to the operator, the value of V3 will be the string "OPER".

See the "Pointer Functions" for information on the GETCOMMAND expression used for setting a pointer to a command.

GETTEXTEX

This function returns the current text from the specified data field:

`GETTEXTEX (<String or Integer>,<Integer>,<String>,<Pointer>)`

This function has four fields.

First Field—Data Field Number or Description

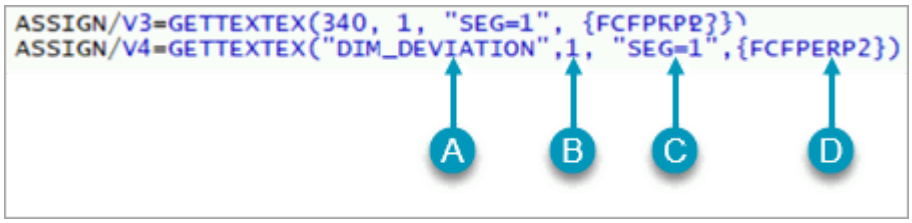
The first field can be either a string description of the data field or the data field number, indicated in item (A) in the image below.

> If you use the Type String Identifier instead of the Numeric Identifier (Item (A) in the image below), PC-DMIS will automatically convert it to the correct numeric value.
>
> For example, if you pass in the string identifier "DIM_DEVIATION", internally, PC-DMIS will convert it to the numeric value 340. When you then hover over the command in the Edit window, the pop-up command shows the text string as well as the actual numeric identifier value. In this example then, if you hover the cursor over the Edit window command, the command pop-up will show `(DIM_DEVIATION - 340), 1, SEG=1` .
>
> You can also just pass in the numeric value if known.

To obtain these values:

1. Place PC-DMIS in Command Mode and then right-click anywhere in the Edit window. A shortcut menu appears.
2. From the shortcut menu, select **Change Pop-up Display** and then **Data Type Information**.
3. Position the mouse over a data field in the Edit window. The type description, type number, and type index for that data item display. Hover over an Extended D_Type to show the content string after the colon.

The type description may be different for different languages. If your measurement routine is executed on a version of PC-DMIS running in a different language, use the type number instead.



*Sample Data Type Information Showing (**A**) Type String or Numeric Identifier, (**B**) Type Index, (**C**) Content String, and (**D**) Command Pointer*

Second Field - Type Index

The second field is the type index, indicated as B in the above image. This field is usually zero unless you have more instances of the same type of field in the same command, such as multiple DIRECTORY fields shown in the above image. You can

obtain the correct value for this field in the same manner as described for the first field.

Third Field—Content String

The third field is the content string of the extended D_TYPE, indicated as C in the image above.

Fourth Field—Command Pointer

The fourth field is a command pointer, indicated as D in the image above. It points to the command that the expression extracts data from. You can use either the command pointer notation (i.e., {FCFPERP2}) or the GETCOMMAND expression as shown below:

`ASSIGN/V1= GETTEXTEX ("DIM_DEVIATION",1,"SEG=1", {FCFPERP2})` - This command assigns V1 the current value of the deviation from feature 1, segment 1, dimension FCFPERP2.

See the "Pointer Functions" for information on the GETCOMMAND expression used for setting a pointer to a command.

The GETTEXTEX function adds support for the Extended DTypes which contain a CONTENT string. Currently, only the PC-DMIS Geometric Tolerance commands use Extended DTypes.

GETPROGRAMINFO

This function returns the measurement routine's information based on the parameters passed in: `GETPROGRAMINFO (<String>, <Optional String>)`

This function has at most two strings as parameters. For most items, you only need the first parameter. The string fields are not case sensitive.

First Field—String

The first field is a string input detailing what information to return.

CADMODELFILE - Returns the full path to the file name of the CAD model that you imported into the measurement routine.

CADMODELFILENAME - Returns the name of only the CAD model (not the path) that you imported into the measurement routine.

DATE - Returns the current date.

DRAWING - Like REVISION, this also returns the revision number as defined in the heading.

ELAPSEDTIME - Returns the time that has passed since the beginning of the execution.

FILENAME - Returns the measurement routine's file name (.prg).

NUMMEAS - Returns the number of dimensions executed.

NUMOOT - Returns the number of out of tolerance dimensions executed.

PARTNAME - Returns the part name as defined in the measurement routine header.

PARTPATH - Returns the full pathway to the measurement routine file.

PCDMISVERSION - Returns a string value of the actual installed version of the PC-DMIS software.

PRGSCHEMA - Returns an integer of the PC-DMIS schema number of the measurement routine file. This is an internal value used by PC-DMIS to indicate the commands and options that are serialized.

PRGVERSION - Returns a string value of the PC-DMIS version number of the measurement routine file. You can save a measurement routine file to be compatible with a specific version. For more information, see "Save As" in the "Using Basic File Options" chapter.

PROBEFILE - Returns the name of the current probe file in use.

REPORTNAME - Returns the current output file name.

REVISION - Returns the revision number as defined in the heading.

SERIALNUM - Returns the serial number as defined in the header.

SEQNUM - Like STATSCOUNT, this string also returns the current stats count number.

SHRINK - Returns the global scale factor.

STATSCOUNT - Returns the current stats count.

TEMP - Returns the temperature for the optional second input string. See "Second Field—Optional String" below.

TIME - Returns the current time.

TIPID - Returns the name of the current tip in use.

Second Field—Optional String

The second field is an optional string input. It's only needed if TEMP is used in the first input field. The possible strings below come from the Temperature Compensation command. For more information, see "Compensating For Temperature" in the "Setting Your Preferences" chapter.

HIGH_THRESHOLD - Returns the high threshold temperature

LOW_THRESHOLD - Returns the low threshold temperature

REF_TEMP - Returns the reference temperature

TEMPP - Returns the temperature for the part sensor

TEMPX - Returns the temperature for the X axis sensor

TEMPY - Returns the temperature for the Y axis sensor

TEMPZ - Returns the temperature for the Z axis sensor

Example

```
$$ NO, This code sample displays the number of total
dimensions and the number of out-of-tolerance dimensions.
      ASSIGN/V1=GETPROGRAMINFO("NUMMEAS")
      ASSIGN/V2=GETPROGRAMINFO("NUMOOT")
      COMMENT/REPT
      "Total Dimensions: "+V1
      "Total Out of Tolerance: "+V2
$$ NO,This code sample returns the temperature on the Z
sensor axis.
      ASSIGN/V3=GETPROGRAMINFO("TEMP", "TEMPZ")
      COMMENT/REPT
      "Temperature on the Z axis: "+V3
```

GETTRACEVALUE

Read Trace Value: *GETTRACEVALUE(<string>)*

This function takes a single string parameter. It returns a value from a `TRACEFIELD` command in the measurement routine.

<string> represents a case-sensitive string of the trace name whose value you want to return.

If you have multiple tracefields with the same trace name, this function returns the value of the most recent tracefield above this function. If a tracefield doesn't contain a value, this function returns a value of 0. In the example below, "Operator" is the tracefield name in the measurement routine:

```
ASSIGN/V2=GETTRACEVALUE("Operator")
```

INDEX

Substring Location*: INDEX(<String>, <String>)*

This function returns the location of the second string within the first string. The first letter of string is 1. A return value of zero indicates that the sub string is not found in the string.

For an example of this function, see the "Sample Code for Read Line" topic in the "Using File Input / Output" chapter.

LASTEXECUTIONTIME

Formatted Last Execution Time: *LASTEXECUTIONTIME()*

This function returns the last execution time that PC-DMIS recorded and stored in the <*name of measurement routine*>.MiniRoutines.xml file. The last execution time appears in the **Execution** dialog box. The time is returned in "hh:mm:ss" format.

LEFT

Left number of characters of a string: *LEFT(<String>, <n>)*

This function returns a string consisting of a number of left-most characters specified by the second expression (n) from the string specified in the first expression (String).

The first expression (String) is coerced to type string. The second expression (n) is coerced to type integer.

For an example of this function, see the "Sample Code for Read Line" topic in the "Using File Input / Output" chapter.

LEN

Length of string: *LEN(<String>)*

This function returns the number of characters of the string.

LOWERCASE

Create lowercase string: *LOWERCASE(<String>)*

This function returns a string that is the lowercase equivalent of the expression string.

MID

Middle n characters of a string: *MID(<String>, <Integer>, <Optional Integer>)*

This function returns a substring consisting of the characters of the string specified in the first parameter starting at the position specified by the second parameter for a length of n characters as specified by the third parameter. If the third parameter is not supplied, the rest of the string is returned.

For an example of this function, see the "Sample Code for Read Line" topic in the "Using File Input / Output" chapter.

ORD

Ordinal Conversion: *ORD(<String>)*

This function returns the integer ASCII value of first letter of the string (0-255).

PCDMISAPPLICATIONPATH

Full pathway display: PCDMISAPPLICATIONPATH*()*

This function returns the string value containing the full path to the application directory where PC-DMIS is installed. This directory contains the main executable and other necessary program files to run PC-DMIS.

PCDMISUSERHIDDENDATAPATH

Full pathway display: PCDMISUSERHIDDENDATAPATH*()*

This function returns the string value containing the full path of the hidden user data directory used by PC-DMIS. See "Understanding File Locations" for the files contained in this directory.

PCDMISUSERVISIBLEDATAPATH

Full pathway display: PCDMISUSERHIDDENDATAPATH*()*

This function returns the string value containing the full path of the visible user data directory used by PC-DMIS. See "Understanding File Locations" for the files contained in this directory.

PCDMISSYSTEMHIDDENDATAPATH

Full pathway display: PCDMISSYSTEMHIDDENDATAPATH*()*

This function returns the string value containing the full path of the hidden system data directory used by PC-DMIS. See "Understanding File Locations" for the files contained in this directory.

PCDMISSYSTEMVISIBLEDATAPATH

Full pathway display: PCDMISSYSTEMVISIBLEDATAPATH*()*

This function returns the string value containing the full path of the visible system data directory used by PC-DMIS. See "Understanding File Locations" for the files contained in this directory.

PCDMISSYSTEMREPORTINGPATH

Full pathway display: PCDMISSYSTEMREPORTINGPATH*()*

This function returns the string value containing the full path to the Reporting directory used by PC-DMIS. This directory contains the report and label templates used by the Report window.

RIGHT

Right n characters of string: *RIGHT(<String>, <Integer>)*

This function returns a string consisting of the n rightmost characters specified by integer from the string.

SYSTEMDATE

System Date: SYSTEMDATE(<Date Format String>)

This function returns the date-formatted string with the current date details filled in. For example, the command SYSTEMDATE("MM’/’dd’/’yy") returns the string "02/12/14" if the current date is February 12, 2014.

Use the following string elements to create the date string. Elements must be in the same case as shown below (MM instead of mm). Non-date characters (such as spaces) that appear between date format string elements appear in the output string in the same location as the input string. Characters in the input string delimited by single quotes appear in the same location in the output string without the single quotes.

d - Day of the month as digits. No leading zero for single-digit dates.

dd - Day of the month as digits. Leading zero used for single-digit dates.

ddd - Three-letter abbreviation for the day of the week.

dddd - Full name for the current day of the week.

M - Month as digits with no leading zeros for single-digit months.

MM - Month as digits with leading zero for single-digit months.

MMM - Month as three-letter abbreviation.

MMMM - Full name of month.

y - Year as digits with no leading zeros for single-digit years.

yy - Year as digits with leading zero for single-digit years.

yyyy - Year represented by four digits.

SYSTEMTIME

Formatted System Time: *SYSTEMTIME(<Time Format String>)*

This function returns the time-formatted string with the current time details filled in. For example, the command SYSTEMTIME("hh:mm:ss tt") returns the time in a formatted string, like this "11:29:40 PM".

Use the following string elements to create the time string. Elements must be in the same case as shown below (**tt** instead of **TT**). Non-time characters (such as spaces) appearing between time format string elements will appear in the output string in the

same location as the input string. Characters in the input string delimited by single quotes will appear in the same location in the output string without the single quotes.

h - Hours with no leading zero for single-digit hours; 12-hour clock

hh - Hours with leading zero for single-digit hours; 12-hour clock

H - Hours with no leading zero for single-digit hours; 24-hour clock

HH - Hours with leading zero for single-digit hours; 24-hour clock

m - Minutes with no leading zero for single-digit minutes

mm - Minutes with leading zero for single-digit minutes

s - Seconds with no leading zero for single-digit seconds

ss - Seconds with leading zero for single-digit seconds

t - One character time marker string, such as A or P

tt - Multi-character time marker string, such as AM or PM

SYSTIME

System Time: *SYSTIME()*

This function returns a string with the current system time. This function differs from the SYSTEMTIME function described above. It automatically returns the day, date, and time, followed by the year.

Example: "Wed February 12 13:50:21 2014"

The returned string, showing the current system time, is adjusted to local time zone settings.

UPPERCASE

Creates uppercase string: *UPPERCASE(<String>)*

This function returns a string that is the uppercase equivalent of string.

Mathematical Functions

ABS

Absolute Value: *ABS(<Double>)*

Returns the absolute value of the input.

EXP

Exponential: *EXP(<Double>)*

Returns the exponential of the expression.

LOG

Log Base 10: *LOG(<Double>)*

Returns the log base 10 of the expression.

LN

Natural Log: *LN(<Double>)*

Returns the natural logarithm of the expression.

ROUND

Rounding:*ROUND(<Double>)*

Returns the input rounded to the nearest integer.

SQRT

Square Root: *SQRT(<Double>)*

Returns the square root of the input.

Trigonometry Functions

By default, each of the trigonometry functions takes and returns radians. If you want values in degrees, use the RAD2DEG function described below.

ACOS

ArcCosine: *ACOS(<Double>)*

Returns the arc cosine of the expression. For example, ACOS(5.0) returns 0. In general, ACOS(<expression>) returns the arc cosine of the value of the expression.

ASIN

ArcSine: *ASIN(<Double>)*

Returns the arc sine of the input.

ATAN

ArcTangent: *ATAN(<Double>)*

Returns the arc tangent of the input.

COS

Cosine: *COS(<Double>)*

Returns the cosine of the input.

DEG2RAD

Degrees to Radians: *DEG2RAD(<Double>)*

Returns the input divided by 360 and multiplied by 2p. This converts from degrees to radians.

RAD2DEG

Radians to Degrees: *RAD2DEG(<Double>)*

Returns the input multiplied by 360 and divided by 2p. This converts from radians to degrees.

SIN

Sine: *SIN(<Double>)*

Returns the sine of the input.

TAN

Tangent: *TAN(<Double>)*

Returns the tangent of the input.

Functions where the input is out of range (like ACOS, ASIN, LOG, LN, SQRT and so on, that would cause the computer to crash) return 0.

Point Functions

ANGLEBETWEEN

Angle Between: *ANGLEBETWEEN(<vector>, <vector>)*

Returns the value of the angle in degrees between the two vectors. The two parameters must be expressions that evaluate to a vector type. To get the vector from a feature, for example, you need to use the feature ID followed by the .IJK extension. You can see this in the code snippet below:

```
F1          =GENERIC/POINT,DEPENDENT,CARTESIAN,$
             NOM/XYZ,<3,3,3>,$
             MEAS/XYZ,<3,3,3>,$
             NOM/IJK,<1,0,0>,$
             MEAS/IJK,<1,0,0>
F2          =GENERIC/POINT,DEPENDENT,CARTESIAN,$
             NOM/XYZ,<10,10,10>,$
             MEAS/XYZ,<10,10,10>,$
             NOM/IJK,<0,0,1>,$
             MEAS/IJK,<0,0,1>
             ASSIGN/V1=F1.IJK
             ASSIGN/V2=F2.IJK
             ASSIGN/V3=ANGLEBETWEEN(V1,V2)
             COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,
             "The angle between "+V1+" and "+V2+" is: "+V3
```

CROSS

Cross Product: *CROSS(<Point>, <Point>)*

Return value is of type point and is the unit vector in the direction of the cross product of the first and second expressions.

DELTA

Vector Offset*: DELTA(<Point>, <Point>, <Double>)*

The function takes the first expression (Point) and calculates a new point in the direction of the second expression (vector) at an offset of the third expression. For example, `DELTA ( MPOINT (0,0,0), MPOINT (1,0,0),10)` returns the point 10,0,0.

DOT

Dot Product: *DOT(<Point>, <Point>)*

Returns the dot product of the two points (vectors).

UNIT

Unit Vector: *UNIT(<Point>)*

Returns the point divided by its length. For example, `UNIT ( MPOINT (0,0,0))` returns the point 0,0,1.

MPOINT

Point Coercion: *MPOINT(<Expression1>, <Expression2>, <Expression3>)*

Coerces the three expressions to a type Point as shown in this code snippet:

```
ASSIGN/V1= MPOINT (2.5,3.6,4)
```

Where:

V1.X = 2.5

V1.Y = 3.6

V1.Z = 4.0

See "Point Coercion".

Pointer Functions

DIST2D

2d Distance: DIST2D(<FEAT1>, <FEAT2>, <FEAT3>)

> The features must be inside curly braces.

This calculates the distance between the first two arguments in the command (Feat1 and Feat2), perpendicular to the third argument (Feat3).

- If the third argument is a plane, PC-DMIS calculates the distance between the first two arguments perpendicular to the plane.
- If the third argument is a line or cylinder, PC-DMIS calculates the distance between the first two arguments perpendicular to the third argument in the active workplane.

For example, if you have the XY plane as your third argument, it will have a Z+ vector (0,0,1) and the distance reported will only be in the Z axis.

Example

```
ASSIGN/V3=DIST2D({CIR1},{CIR2},{PLN1})
            COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,
            V3
```

DIST3D

3D Distance: DIST3D(<FEAT1>, <FEAT2>)

Calculates the 3D distance between Feat1 and Feat2.

> The features must be inside curly braces.

Example

```
ASSIGN/V3=DIST3D({CIR1},{CIR2})
            COMMENT/OPER,NO,FULL SCREEN=NO,AUTO-CONTINUE=NO,
            V3
```

GETCOMMAND

Obtains a pointer to the command specified by the parameters:
`GETCOMMAND (<Integer or String>, <String>,<Integer>`

First Parameter-Command Info Field

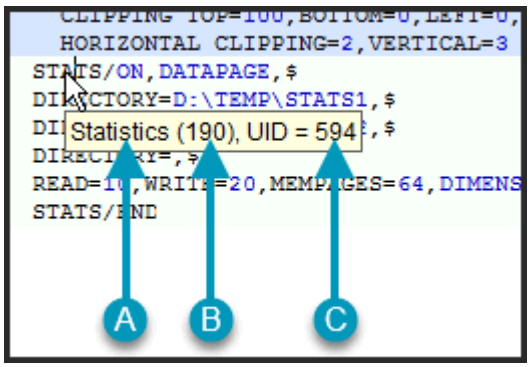
The first parameter is the command info field. It specifies the command type for which to search. The following can be passed in:

- A command description string. See (A) in the graphic below.
- A command type number. See (B) in the graphic below.
- The unique number identifier. See (C) in the graphic below.

If the unique id of the command is passed in, no other arguments are necessary.

To obtain the command description string, the command type number, and the command's unique number identifier:

1. Right-click in the Edit window
2. Choose **Change Pop-up Display | Command Information** (PC-DMIS must be in Command Mode).
3. Position the mouse over the desired command. PC-DMIS displays the command description, type number, and unique number identifier for that command in the pop-up.



A. *Command Description String*

B. *Command Type Number*

C. *Unique Number Identifier (UID)*

Second Parameter - Search Direction

The second parameter is the search direction. Legal values include:

Value	Description
UP	This value signifies that the search should start at the current command and proceed upwards
DOWN	This value signifies that the search should start at the current command and proceed downwards.
TOP	This value signifies that the search should begin at the start of the measurement routine in a downwards direction
BOTTOM	This value signifies that the search should begin with the last object in the measurement routine in an upwards direction

Third Parameter - Which Instance to Find

The third parameter indicates which instance of the command should be found if multiple instances of the same command are in the measurement routine.

If the measurement routine has two instances of a STATS/ON command and you would like to obtain a pointer to the second instance from the top, pass in " 2 " as the third parameter and pass in " TOP " as the second parameter as shown here.

```
ASSIGN/V1= GETCOMMAND ("Statistics", "TOP" ,2)
```

You can use the GETCOMMAND function to supply the third parameter to the GETTEXT string function. See "String Functions" for information on GETTEXT.

LEN

Pointer loop count: *LEN(<POINTER>)*

Returns the number of times a pointer has been in a loop. For example, if feature CIR1 is in a loop that iterates 10 times, you can store how many times CIR1 has been measured in a variable with an `ASSIGN` statement like this: `ASSIGN/V1= LEN ({CIR1})`

Array Functions

ARRAY

Create Array: *ARRAY(<EXPRESSION1>, <EXPRESSION2>, <EXPRESSION3>, …)*

Creates an array object with the array elements indicated by the expression parameters. The array elements are numbered with a base index of 1.

AVERAGE

Average array elements: *AVERAGE(<ARRAY>)*

Returns the average value of elements in the array.

EQUAL

Element by Element Array Comparison: *EQUAL(<ARRAY>, <ARRAY>)*

Compares the two arrays, element by element, to determine if the arrays hold the same elements. If the two arrays are not the same size, or if any of the elements in one array do not match the corresponding element in the other array, the function returns 0. Otherwise, the function returns 1.

LEN

Array element count: *LEN(<ARRAY>)*

Returns the number of elements in the array.

MAX

Largest array element: *MAX(<ARRAY>)*

Returns the largest element in the array. Items in the array are compared numerically or alphabetically.

MIN

Smallest array element: *MIN(<ARRAY>)*

Returns the smallest element in the array. Items in the array are compared numerically or alphabetically.

SUM

Sum array elements: *SUM(<ARRAY>)*

Returns the sum of elements in the array.

Miscellaneous Functions

ARCSEGMENTENDINDEX

This function returns the index number of the ending point of a specified arc segment from a scan: ARCSEGMENTENDINDEX*(<ID>, <index>,<tol1>,<tol2>)*

<ID> - The first parameter is a string value of the ID of the scan on which this function pulls out the index number of the ending point for the arc. This parameter can either be the ID in quotation marks or any expression that when coerced to type string ends up being the ID of a scan.

<index> - The second parameter is the index number for the arc from which you want to get the ending point number. This is a one-based value. For example, the arc index number would be 3 if you want the ending point number for the third arc in the scan.

<tol1> - The third parameter is the general feature tolerance. It is a maximum form error used to break up the scan into lines and arcs.

<tol2> - The fourth parameter is the refine tolerance. Generally, this tighter tolerance is used to drop points from either end of the feature until the form error of the segment is within this tolerance.

Once you have the Start and End indices for an arc, you can use these points inside a constructed feature to construct a separate arc feature. For a similar example, see "Example of a Line Feature Created from a Scan Segment".

ARCSEGMENTSTARTINDEX

This function returns the index number of the starting point of a specified arc segment from a scan: ARCSEGMENTSTARTINDEX*(<ID>, <index>,<tol1>,<tol2>).*

<ID> - The first parameter is a string value of the ID of the scan on which this function pulls out the index number of the start point for the arc. This parameter can either be the ID in quotation marks or any expression that when coerced to type string ends up being the ID of a scan.

<index> - The second parameter is the index number for the arc from which you want to get the start point number. This is a one-based value. For example, the arc index number would be 3 if you want the start point number for the third arc in the scan.

<tol1> - The third parameter is the general feature tolerance. It is a maximum form error used to break up the scan into lines and arcs.

<tol2> - The fourth parameter is the refine tolerance. Generally, this tighter tolerance is used to drop points from either end of the feature until the form error of the segment is within this tolerance.

There are two additional parameters, which control whether an identified arc segment in a scan is acceptable. These can only be changed with the PC-DMIS Settings Editor. Any arc segment with a radius less than the value for the `MinimumArcSegmentRadiusInMM` entry is rejected. The default value for this parameter is 2 mm. Similarly, any arc segment with a radius greater than the value for the `MaximumArcSegmentRadiusInMM` entry is rejected. The default value for this parameter is 2000 mm (it should not be necessary to change this value).

Once you have the Start and End indices for an arc, you can use these points inside a constructed feature to construct a separate arc feature. For a similar example, see "Example of a Line Feature Created from a Scan Segment".

EOF and EOL

For information on these functions, see "Checking for the End of a File or the End of a Line" in the "Using File Input / Output" chapter.

FUNCTION

Creates a function: FUNCTION*((<PARAM1>, <PARAM2>…), <EXPRESSION>)*

Creates a function that takes the number of parameters indicated by the parameter list and substitutes those parameters into the expression.

- The first item when using the `FUNCTION` keyword is the parameter list.
- This list consists of parameter names separated by commas.
- The parameter list is also surrounded by parenthesis.
- The second item is the expression.
- The expression contains the parameter names where the parameters should be substituted when the function is called.

For an example, see the "Generic Function Example" topic.

GETROTABDATA

This function returns the center, angular position, and vector values for the specified rotary table.

GETROTABDATA(<*PARAMETER*>[,<*TABLE*>])

The function returns the values for the following configurations:

- Single rotary table
- Dual (independent) rotary tables
- Stacked rotary tables

The data that the function returns matches the data in the **Rotary Table Setup** dialog box (**Edit | Preferences | Rotary Table Setup**). For more information about this dialog box, see "Defining the Rotary Table".

CENTER

- "CENTER" - Returns the rotary table's current XYZ center.
- "CENTER","V" - Returns the current XYZ center for rotary table V in a dual-table or stacked-table configuration.
- "CENTER","W" - Returns the current XYZ center for rotary table W in a dual-table or stacked-table configuration.

Examples:

`ASSIGN/ V1=GETROTABDATA("CENTER")`	V1 is set to the rotary table's current XYZ center.
`ASSIGN/ V1=GETROTABDATA("CENTER","V")`	V1 is set to the current XYZ center for rotary table V.
`ASSIGN/ V1=GETROTABDATA("CENTER","W")`	V1 is set to the current XYZ center for rotary table W.

ANGULAR POSITION

- "ANGLE" - Returns the rotary table's current angular position.
- "ANGLE","V" - Returns the current angular position for rotary table V in a dual-table or stacked-table configuration.
- "ANGLE","W" - Returns the current angular position for rotary table W in a dual-table or stacked-table configuration.

Examples:

`ASSIGN/ V2=GETROTABDATA("ANGLE")`	V2 is set to the rotary table's current angular position.
`ASSIGN/ V2=GETROTABDATA("ANGLE","V")`	V2 is set to the current angular position for rotary table V.
`ASSIGN/ V2=GETROTABDATA("ANGLE","W")`	V2 is set to the current angular position for rotary table W.

VECTOR

- "VECTOR" - Returns the rotary table's current IJK vector.
- "VECTOR","V" - Returns the current IJK vector for rotary table V in a dual-table or stacked-table configuration.
- "VECTOR","W" - Returns the current IJK vector for rotary table W in a dual-table or stacked-table configuration.

Examples:

`ASSIGN/ V3=GETROTABDATA("VECTOR")`	V3 is set to the rotary table's current IJK vector.
`ASSIGN/ V3=GETROTABDATA("VECTOR","V")`	V3 is set to the current IJK vector for rotary table V.
`ASSIGN/ V3=GETROTABDATA("VECTOR","W")`	V3 is set to the current IJK vector for rotary table W.

The [TABLE] argument is optional. If you do not specify table V or W, PC-DMIS does one of the following:

- If you are using a single-table or stacked-table configuration, it returns the values for rotary table W.
- If you are using a dual-table configuration, it returns the values for the rotary table that is activated on the **Active Rotary Table** toolbar. For more information about the toolbar, see "Active Rotary Table Toolbar".

PC-DMIS has two internal table definitions to accommodate dual-table and stacked-table configurations. For a single-table configuration, the second table definition is not actually used. Because it does exist internally, an error will not occur if you specify table V in a single-table configuration; however, this is not recommended. The values that the function returns are not generally useful because the table does not actually exist.

IF

Conditional expression evaluation: IF*(<EXPRESSION1>, <EXPRESSION2>, <EXPRESSION3>)*

If expression1 evaluates to true (non-zero), then this function returns the value of expression2; otherwise, this function returns the value of expression3.

ISIOCHANNELSET

This expression takes two parameters. The first parameter indicates which I/O channel will be checked (the range of numbers that is available is based on the machine being used). The second parameter determines whether the software will query the Arm1 or Arm2 machine. If the second parameter is set to 1 (one), it will query the Arm2 controller. If the second parameter is not present (or is set to zero), then the IO Channel will query the Arm1 controller. The Arm1 controller is your only option if you are not in multiple arm mode.

If an invalid probe data type, tip id, probe file name, or channel number is supplied, the expression evaluates to 0.

Example:

`ASSIGN/ V4= ISIOCHANNELSET (3,0)`	V4 will equal 1 (evaluate to true) when the channel is set, otherwise it will equal 0 (evaluate to false).

LINESEGMENTENDINDEX

This function returns the index number of the ending point of a specified line segment from a scan: LINESEGMENTENDINDEX*(<ID>, <index>,<tol1>,<tol2>).*

<ID> - The first parameter is a string value of the ID of the scan on which this function pulls out the index number of the ending point for the line segment. This parameter can either be the ID in quotation marks or any expression that when coerced to type string ends up being the ID of a scan.

<index> - The second parameter is the index number for the line segment from which you want to get the end point number. This is a one-based value. For example, the line segment index number would be 3 if you want the end point number for the third line in the scan.

<tol1> - The third parameter is the general feature tolerance. It is a maximum form error used to break up the scan into lines and arcs.

<tol2> - The fourth parameter is the refine tolerance. Generally, this tighter tolerance is used to drop points from either end of the feature until the form error of the segment is within this tolerance.

Once you have the Start and End indices for a line segment, you can use these points inside a constructed feature to construct a separate line feature. For an example, see "Example of a Line Feature Created from a Scan Segment".

LINESEGMENTSTARTINDEX

This returns the index number of the starting point of a specified line segment from a scan: LINESEGMENTSTARTINDEX*(<ID>, <index>,<tol1>,<tol2>).*

<ID> - The first parameter is a string value of the ID of the scan on which this function pulls out the index number of the start point for the line segment. This can either be the ID in quotation marks or any expression that when coerced to type string ends up being the ID of a scan.

<index> - The second parameter is the index number for the line segment from which you want to get the start point number. This is a one-based value. For example, the line segment index number would be 3 if you want the start point number for the third line in the scan.

<tol1> - The third parameter is the general feature tolerance. It is a maximum form error used to break up the scan into lines and arcs.

<tol2> - The fourth parameter is the refine tolerance. Generally, this tighter tolerance is used to drop points from either end of the feature until the form error of the segment is within this tolerance.

There is an additional parameter that controls whether an identified line segment in a scan is acceptable. This can only be changed with the PC-DMIS Settings Editor. Any line

segment of a length less than the value for the `MinimumLineSegmentLengthInMM` entry is rejected. The default value for this parameter is 2 mm.

Once you have the Start and End indices for a line segment, you can use these points inside a constructed feature to construct a separate line feature. See "Example of a Line Feature Created from a Scan Segment" for an example.

PROBEDATA

This function returns data about the current or specified probe : PROBEDATA*(<OPTPROBEDATATYPE>, <OPTTIPID>, <OPTPROBEFILENAME>)*

This function takes up to three optional parameters. You only need to provide commas between the parameters if you use more than one parameter. You do not need to use commas between empty parameters. For example, to obtain the current probe's diameter you would use `ASSIGN/V1= PROBEDATA("DIAM")` .

OPTPROBEDATATYPE - Optional parameter which specifies what probe data the expression should return. If this parameter is not supplied, the current tip ID is returned. This parameter is of type string. Any expression that evaluates to a valid string expression can be put in the first expression slot. Valid string expressions (not case sensitive) for the first parameter include the following. These are string expressions and should be inside double quotation marks:

"**A**" - Tip A Angle. Returns type double.

*"**B***" - Tip B Angle. Returns type double.

*"**C**"* **-** The C angle of a CW43 light probe head. Returns type integer.

*"**Date**"* - Date the tip was last qualified. Returns type string.

"**Diam"** *or* **"Diameter**" - Measured tip diameter. The first four letters are required "Diam", but could include more letters up to the full name of "Diameter". Returns type double.

*"**ID**"* - Tip ID. Default parameter. Returns type string.

"**Offset**" - Measured tip X,Y,Z offset. Returns type point.

"PrbRdv" - The probe's radial deviation. Returns type double.

"**Rotation**" - This is the rotation about the tip vector in radians. Returns type double.

*"***Standarddeviation***"* - The probe's standard deviation. Returns type double.

*"**Thick"** or **"Thickness**"* - Measured tip thickness. The first five letters are required "Thick", but could include more letters up to the full name of "Thickness". Returns type double.

"Time" - Time the tip was last qualified. Returns type string.

*"**Vector**"* - Tip Vector. Returns type point.

> Adding a "**T**" in front of "**Diameter**", "**Offset**", or "**Thickness**" returns the theoretical information (for example, **TDIAMETER**, **TOFFSET**, and **TTHICKNESS**).

OPTTIPID - This optional parameter specifies the tip to be used when obtaining the probe data specified in the first expression. If not supplied, the current tip is used. This parameter should be type string.

OPTPROBEFILENAME - This optional parameter specifies the probe filename to be used in obtaining the probe data. If this is not supplied, the current probe file is used.

Examples:

`ASSIGN/V1= PROBEDATA ()`	V1 is set to current tip id (i.e. "T1A0B0")
`ASSIGN/ V2= PROBEDATA ("TOFFSET","T1A45B0")`	V2 is set to the theoretical probe offset for the tip T1A45B0
`ASSIGN/ V3= PROBEDATA ("Date","T1A90B90","MYPROB")`	V3 is set to a string representing the date tip T1A90B90 of the probe file MYPROB was last qualified.

QUALTOOLDATA

This function returns data about the current or specified calibration tool. It has this syntax:

QUALTOOLDATA(<*TOOLINFO*>, <*TOOLID*>, <*FACENUMBER*>)

This function takes up to three parameters. It needs at least one parameter to return any data:

The first parameter, <*TOOLINFO*>, is a *string* that specifies the type of information to return about the calibration tool. If you don't pass this parameter, this function returns the name of the current or specified calibration tool.

- "***CTE***" or "***COEFFICIENTOFTHERMALEXPANSION***" - Either one of these strings returns the coefficient of thermal expansion as a double value.
- "***DIAM***" - This string returns the diameter of the tool as a double value.
- "**ID**" - This string returns the name of the tool as a string value.
- "**LENGTH**" - This string acts the same as "DIAM". It also returns the diameter of the tool as a double value.
- "**OVERRIDEIJK**" - This string returns the search override IJK vector as a point value.
- "**POLYDIAM**" - This string returns the diameter of the specified polyhedral face as a double value.

- "**POLYIJK**" - This string returns the IJK vector of the specified polyhedral face as a point value.
- "**POLYXYZ**" - This string returns the XYZ center of the specified polyhedral face as a point value.
- "**SHANKIJK**" - This string returns the IJK vector of the shank as a point value.
- "**TYPE**" - This string returns the type of tool as an integer value (0 for a sphere, 1 for an Arm2 sphere, 2 for a polyhedral, 3 for an Arm2 polyhedral).
- "***WIDTH***" - This parameter is no longer used.
- "**XYZ**" - This string returns the XYZ location of the tool as a point value.

  The second parameter, <*TOOLID*>, is a *string* that specifies the name for the calibration tool for which you would like to receive information. If you don't pass this parameter, PC-DMIS assumes you want information from the current calibration tool. The string is not case sensitive.

  The third parameter, <*FACENUMBER*>, is a parameter that you only need when you work with a polyhedral calibration tool and only when the first parameter is "POLYXYZ", "POLYIJK", or "POLYDIAM". This is an integer value that specifies the face of the polyhedral tool to use in order to obtain data.

The calibration tool does not have a global setting that automatically applies to all the probes in a measurement routine. When you calibrate a probe for the first time, you need to select the calibration tool to use. PC-DMIS saves this calibration tool information for each probe. When you recalibrate the same probe, PC-DMIS will use the same calibration tool.

Examples:

`ASSIGN/ VDIAM=QUALTOOLDATA("DIAM ","SPHERE_1_IN")`	Gives the variable VDIAM the diameter of the tool SPHERE_1_IN.
`ASSIGN/ VID=QUALTOOLDATA("ID")`	Gives the variable VID the current tool's name.
`ASSIGN/ VTYPE=QUALTOOLDATA("TYPE")`	Gives the variable VTYPE the current tool's type.
`ASSIGN/ VPOLYDIAM=QUALTOOLDATA(" POLYDIAM","POLYTEST",3)`	Gives the variable VPOLYDIAM the diameter of face 3 on the polyhedral tool, POLYTEST.

SETROTABDATA

This function sets the center or vector to the new input value:
SETROTABDATA(<*PARAMETER*>,<*NewValue*>[,<*TABLE*>])
This function works in the following configurations:

- Single rotary table
- Dual (independent) rotary tables
- Stacked rotary tables

CENTER

- "CENTER", - Sets the rotary table's XYZ center to the new value.
- "CENTER",,"V" - Sets the XYZ center for rotary table V in a dual-table or stacked-table configuration to the new value.
- "CENTER",,"W" - Sets the XYZ center for rotary table W in a dual-table or stacked-table configuration to the new value.

Examples:

`ASSIGN/ V1= SETROTABDATA("CENTER",NewValue)`	V1 is a return code (1=success, 0=failure).
`ASSIGN/ V1= SETROTABDATA("CENTER",NewValue,"V")`	V1 is a return code (1=success, 0=failure).
`ASSIGN/ V1= SETROTABDATA("CENTER",NewValue,"W")`	V1 is a return code (1=success, 0=failure).

VECTOR

- "VECTOR", - Sets the rotary table's IJK vector to the new value.
- "VECTOR",,"V" - Sets the IJK vector for rotary table V in a dual-table or stacked-table configuration to the new value.
- "VECTOR",,"W" - Sets the IJK vector for rotary table W in a dual-table or stacked-table configuration to the new value.

Examples:

`ASSIGN/ V2= SETROTABDATA("VECTOR",NewValue)`	V2 is a return code (1=success, 0=failure).
`ASSIGN/ V2= SETROTABDATA("VECTOR",NewValue,"V")`	V2 is a return code (1=success, 0=failure).
`ASSIGN/ V2= SETROTABDATA("VECTOR",NewValue,"W")`	V2 is a return code (1=success, 0=failure).

The [TABLE] argument is optional. If you do not specify table V or W, PC-DMIS does one of the following:

- If you are using a single-table or stacked-table configuration, it sets the new value for rotary table W.
- If you are using a dual-table configuration, it sets the new value for the rotary table that is activated on the **Active Rotary Table** toolbar. For more information about the toolbar, see "Active Rotary Table Toolbar".

PC-DMIS has two internal table definitions to accommodate dual-table and stacked-table configurations. For a single-table configuration, the second table definition is not actually used. Because it does exist internally, an error will not occur if you specify table V in a single-table configuration; however, this is not recommended. The values that the function sets are not generally useful because the table does not actually exist.

TUTORELEMENT

This function takes one parameter, either a number or a string (a string would be the ID of a feature).

TUTORELEMENT(<*PARAMETER*>)

This function works with the variable type, *Structures*. For explanations of structure and sub-elements, see "Structures".

Examples:

`ASSIGN/ E=TUTORELEMENT(1)`	Creates a single Tutor Element Structure
`ASSIGN/ WM=TUTORELEMENT(n)`	For any number over 1, this creates an array of `n` Tutor Element Structures
`ASSIGN/ CIR1E=TUTORELEMENT ("CIR1")`	Copies Data from feature CIR1 into the Tutor Element Structures.

The TutorElement structure currently has the following sub-elements:

Sub-Element	Description
ID	String of the Feature's ID
TYPE	INTEGER (FTYPE)
X, Y, Z	X, Y, and Z coordinate values
PR	Polar Radius
PA	Polar Angle
CX	I
CY	J
CZ	K
DM	Diameter 1
DM2	Diameter 2
DS	Distance from the origin
A	Angle
AXY	Angle in the XY plane
AYZ	Angle in the YZ plane

AZX	Angle in the ZX plane
F	Form Error
SDEV	Standard Deviation
TP	Position

Function Examples

Below are some different examples of functions that may help you in creating and using your own functions:

- Generic Function Example
- Functions Passed as Variables Example
- Function with Multiple Parameters Example
- Functions Creating Other Functions Example
- Functions As Members of an Array Example
- Functions Defined Recursively Example

Generic Function Example

```
ASSIGN/MYFUNC= FUNCTION ((X,Y,Z),X*3+Y*2+Z)
```

Creates a user-defined function and assigns it to the variable MYFUNC. The function takes three parameters, X,Y, and Z.

- X is multiplied by 3.
- Y is multiplied by 2.
- Z holds the passed value.

The total of X + Y + Z is what gets returned when values are passed into the function, as shown here:

```
ASSIGN/V1=MYFUNC(7,2,5)
```
This assigns V1 the value 30 by evaluating the parameters passed into the function MYFUNC(7,2,5).

- 7 is the parameter and is substituted where X occurs in the expression portion of the function definition. Thus, X*3 becomes 7*3, or 21.
- 2 is substituted where Y occurs. Thus Y*2 becomes 2*2, or 4.
- 5 is substituted where Z occurs.

The values are then all added together (21 + 4 + 5) and passed to V1.

Functions Passed as Variables Example

Functions can be passed as variables. The following example builds on the Generic Function Example above:

```
ASSIGN/NEWFUNC=MYFUNC
```
Sets the variable NEWFUNC to have the same function that MYFUNC has.

```
ASSIGN/V3=NEWFUNC(12,2,3)
```
Assigns V3 to have the value 43 from the evaluated expressions within the function (36 + 4 + 3).

Function with Multiple Parameters Example

Functions can have multiple parameters:

```
ASSIGN/ADDANDDOUBLE= FUNCTION ((A,B),2*(A+B))
```
Creates a function and assigns it to the variable ADDANDDOUBLE. The function takes two parameters, adds them together and then multiplies the result by 2.

```
ASSIGN/V2=ADDANDDOUBLE(4,5)
```
Assigns V2 the value 18. The parameters 4 and 5 are substituted into the expression portion of the function, thus becoming 2*(4+5).

Functions Creating Other Functions Example

Functions can create other functions.

```
ASSIGN/COMPOSE= FUNCTION ((F,G), FUNCTION ((X),G(F(X))))
```
Assigns COMPOSE to be a function that takes two functions as parameters and creates a new function using the two functions.

```
ASSIGN/ADD2= FUNCTION ((X),X+2)
```
Assigns ADD2 to be a function that adds two to the parameter passed in.

```
ASSIGN/ADD3= FUNCTION ((X),X+3)
```
Assigns ADD3 to be a function that adds three to the parameter passed in.

```
ASSIGN/ADD5= COMPOSE (ADD2,ADD3)
```
Assigns ADD5 to be a function composed of the functions ADD2 and ADD3.

```
ASSIGN/V5= ADD5 (3)
```
Assigns V5 to have the value V8.

Functions As Members of an Array Example

Functions can be members of an array.

```
ASSIGN/ANARRAY= ARRAY (3,FACTORIAL,"Hello World",ADD5)
```
Assigns ANARRAY to be an array of 4 elements : a number (3), a function (FACTORIAL), a string, ("Hello World"), and a function (Add5).

```
ASSIGN/V6=ANARRAY[2](4)
```
The second element of ANARRAY is the function FACTORIAL. The parameter 4 is passed in to this function and the result of 24 is assigned to V6.

```
ASSIGN/V7=ANARRAY[2](ANARRAY[4](ANARRAY[1]))
```
From the inside out: The first element of ANARRAY (3) is passed to the function of the fourth array element (Add5). The result, 8, is passed to the function of the second array element (FACTORIAL) and assigned to V7. V7 receives a value of 40320.

Functions Defined Recursively Example

Functions can be defined recursively, meaning they can be defined to call themselves.

```
ASSIGN/FACTORIAL= FUNCTION ((X), IF (X<=1,1,X* FACTORIAL (X-1))
```
Creates a function called factorial that takes one parameter. If the parameter is less than or equal to 1 it evaluates to 1, otherwise it evaluates to X multiplied by the FACTORIAL of X-1.

```
ASSIGN/V4= FACTORIAL (5)
```
Assigns V4 the value of 120 (5*4*3*2*1).

Example of a Line Feature Created from a Scan Segment

This topic provides an example of how to use PC-DMIS expression language, specifically the line segment functions, to export start and end point numbers for line segments within a scan and then to create your own line feature by using the extracted points within a constructed feature. You can use the same principles covered in this example to create an arc segment from a scan as well.

Suppose your measurement routine has a scan feature named SCN1 that looks like this:

```
SCN1=FEAT/SCAN,LINEAROPEN,SHOW HITS=NO,SHOWALLPARAMS=YES
   EXEC MODE=RELEARN, NOMS MODE=FIND
NOMS,CLEARPLANE=NO,SINGLE POINT=NO,THICKNESS=0

FINDNOMS=5,SELECTEDONLY=NO,USEBESTFIT=NO,PROBECOMP=YES,AVOID
ANCE MOVE=NO,DISTANCE=0,CAD Compensation=NO
   DIR1=VARIABLE,
   HITTYPE=VECTOR
   INITVEC=0,-1,0
   DIRVEC=1,0,0
   CUTVEC=0,0,1
   ENDVEC=0,-1,0
   PLANEVEC=-1,0,0
   POINT1=100,0,-5
   POINT2=70,0,-5
   MEAS/SCAN
   BASICSCAN/LINE,SHOW HITS=NO,SHOWALLPARAMS=YES
   <100,0,-5>,<70,0,-5>,CutVec=0,0,1,DirVec=1,0,0
   InitVec=0,-1,0,EndVec=0,-1,0,THICKNESS=0
   FILTER/NULLFILTER,
   EXEC MODE=RELEARN
   BOUNDARY/PLANE,<70,0,-5>,PlaneVec=-1,0,0,Crossings=2
   HITTYPE/VECTOR
   NOMS MODE=FINDNOMS,5
   ENDSCAN
   ENDMEAS/
```

To create a line from this scan you will need to use the LINESEGMENTSTARTINDEX and LINESEGMENTENDINDEX functions to pull out the data, like this:

```
   ASSIGN/LINESTARTINDEX=LINESEGMENTSTARTINDEX("SCN1",1,0.4,
0.1)
   ASSIGN/LINEENDINDEX=LINESEGMENTENDINDEX("SCN1",1,0.4,
0.1)
```

This tells PC-DMIS to go to the scan named "SCN1", and from its first line segment pull out the starting and ending index values that fall within the defined tolerances. It then assigns those index values to variables named LINESTARTINDEX and LINEENDINDEX.

Once you have the start and ending index values for the line segment assigned to variables, you can use those variables within a constructed line, like this:

```
LIN4=FEAT/LINE,RECT,UNBND
   THEO/100.225,0,-5.011,1,0,0
   ACTL/100.225,-0.005,-5.011,1,-0.0000388,0
   CONSTR/LINE,BF,2D,SCN1.HIT[LINESTARTINDEX..LINEENDINDEX],
,
   OUTLIER_REMOVAL/OFF,3
   FILTER/OFF,WAVELENGTH=0
```

Notice that in the highlighted code from the line feature above, PC-DMIS uses the starting and ending numbers you pulled out of the scan to create the feature:
`SCN1. HIT[LINESTARTINDEX..LINEENDINDEX]`

Operand Coercion

Operands can be coerced to other types using any of the coercion operators:

Integer Coercion

INT(<Expression>) - Coerces value of expression to type integer.

INT(4)	Evaluates to 4
INT(4.5)	Evaluates to 4
INT("Hello World")	Evaluates to 0
INT("2")	Evaluates to 2
INT("2.2")	Evaluates 2
INT("3 Blind Mice")	Evaluates to 3
INT("The 3 Blind Mice")	Evaluates to 0
INT("3, 4, 5")	Evaluates to 3
INT(MPOINT(0, 0, 1))	Evaluates to the distance of the point from the origin, in this case 1
INT(MPOINT(3, 4, 5))	Distance evaluates to 7.0711. This expression evaluates to 7.

Double Coercion

DOUBLE(<Expression>) - Coerces value of expression to type double

`DOUBLE(4)`	Evaluates to 4.0
`DOUBLE(4.5)`	Evaluates to 4.5
`DOUBLE("A String")`	Evaluates to 0.0
`DOUBLE("3.5")`	Evaluates to 3.5
`DOUBLE("3.5 inches")`	Evaluates to 3.5
`DOUBLE("The circle measures 3.5 inches in diameter ")`	Evaluates to 0.0
`DOUBLE(MPOINT(0,0,1))`	Evaluates to 1.0
`DOUBLE(MPOINT(3,4,5))`	Evaluates to 7.0711

String Coercion

STR(<Expression>) - Coerces value of expression to type string

`STR(4)`	Evaluates to "4"
`STR(4.5)`	Evaluates to "4.5"
`STR("Hello World")`	Evaluates to "Hello World"
`STR(MPOINT(3,4,5))`	Evaluates to "3, 4, 5"

Point Coercion

MPOINT(<Expression1>, <Expression2>, <Expression3>) - Coerces values of expressions to type point after coercing each expression to type double.

`MPOINT(1, 1, 1)`	Evaluates to point 1.0,1.0,1.0
`MPOINT(1.1, 1.1, 1.1)`	Evaluates to point 1.1, 1.1, 1.1
`MPOINT("1", "1", "1")`	Evaluates to point 1.0,1.0,1.0
`MPOINT(3, 4.5, "5.6")`	Evaluates to point 3.0, 4.5, 5.6

`MPOINT(MPOINT(1, 0, 0),`
`MPOINT(0,1,0),`
`MPOINT(3,4,5))`
	Evaluates to 1.0, 1.0, 7.0711

Operand Coercion and Mixed Type Expressions

The expression evaluator automatically coerces variables in mixed-type expressions. If the result of an expression is not what is expected because of automatic coercion, use of the coercion operators in some cases yields the desired result. Following are examples of automatic coercions in mixed-type expressions.

"CIR" + 1
Evaluates to "CIR1"

"2" + 2
Evaluates to 4

"The Value of 2+2 is " + 2 + 2
Evaluates to "The Value of 2+2 is 22". This is because PC-DMIS evaluates expressions left to right, and the left part of the expression is a string.

"The Value of 2+2 is " + (2 + 2)
Evaluates to "The Value of 2+2 is 4"

LINE1.XYZ > 2
Evaluates to 1 if the distance of the centroid of LINE1 from the origin is greater than 2

LINE1.XYZ > LINE2.XYZ
Evaluates to 1 if the centroid of LINE1 is further from the origin than the centroid of LINE2

LINE1.XYZ = LINE2.XYZ
Evaluates to 1 if the centroids of LINE1 and LINE2 are the same (no coercion occurs in this case)

DOUBLE(LINE1.XYZ) = DOUBLE(LINE2.XYZ)
Evaluates to 1 if the centroids are the same distance from the origin

11% 3.1
Evaluates to 2 (% is the modulo operator designed to work with integers. It returns the remainder from discrete division. 11%3 = 2.)

CIRCLE1.HIT [3.2].X
Evaluates to measured X value of third hit of Circle1. The argument 3.2 is automatically coerced to an integer with a value of 3.

ID Expressions

Many of the PC-DMIS commands use feature IDs as parameters. For example, constructed features use IDs to indicate which features are to be used as inputs for the constructed feature. ID expressions allow the user to refer to a specific instance of a feature, a group of similarly named features, an instance of a feature inside a call to a subroutine, or a feature in an external measurement routine.

Feature Array ID

Use a feature array id to refer to a specific instance of a feature or to a range of feature instances. For example, if the feature "Circle1" were located in a while loop that looped five times, then five instances of the circle would exist upon exiting the loop. To refer to an individual instance of the five instances of "Circle1", use feature array syntax as described under "Feature Arrays:" where "Circle1[1]" would refer to the first instance, "Circle1[2]" would refer to the second instance, etc.

To refer to a range of instances use the **..** notation. "Circle1[1..3]" refers to the 1st through 3rd instances of Circle1. "Circle1[3..5] refers to the 3rd through 5th instances of Circle1. "Circle1[1..5]" would refer to the 1st through 5th instances of Circle1. When a range of feature is referred to, the set is treated and behaves as a constructed set.

ID Wild Cards

Use ID Wild Cards to refer to a set of similarly named features. The two wild card characters are "*****" and "**?**". (See "Selecting Features Using Meta-character matching" in the "Editing the CAD Display" chapter for additional information.)

The asterisk '*' character refers to 0 or more instances of any character. To refer to the set of all features that start with the letters "CIR", use the expression ID "CIR*". This syntax creates a set of features that include all features with Ids that start with "CIR", such as "CIRCLE1", "CIRCLE2", "CIR3", or "CIR".

If CIR3 has several executions only the most recent measurement is used. To get the different instances of the executions, the following expression could be used: CIR?[1..3]

The question mark '?' character refers to a single instance of any character.

The ID expression "MY???1" would create a set of features that are six characters long, begin with "MY" and end with "1", such as "MYCIR1", "MYCON1", "MYLIN1", or "MYFT21".

IDs for Features Within Subroutines, Basic Scripts, or External Routines

Subroutines can be located within the current measurement routine or in an external measurement routine. When the subroutine is located in the same routine as the call to the subroutine, you can use the feature array ID syntax explained under "Feature Arrays:" to refer to individual instances of a feature created in the subroutine. However, when the subroutine is located in an external measurement routine, you can use the following syntax to refer to any features created in the subroutine: "<Call Sub ID>:<FeatID>".

For example, if a feature named "F1" were located in an external subroutine that was called from a Call Sub command with the id "CS1", then the ID expression "CS1:F1" could be used to refer to that feature.

This example merely illustrates the use of the syntax CS1.F1 and is not intended for use.

Routine 1: PLUS1.PRG

```
SUBROUTINE/PLUS1, A1 = 0, A2 = 0, A3 = 0

F1 =FEAT/POINT,RECT

THEO /A1+1,A2+1,A3+1,0,0,1

ACTL/3,1,1,0,0,1

MEAS/POINT,1

HIT /BASIC,A1+1,A2+1,A3+1,0,0,1,0,0,0

ENDMEAS/

ENDSUB/
```

Routine 2: TEST.PRG

```
CS1 =CALLSUB/PLUS1,D:\V30\WINDEBUG\PLUS1.PRG: 3,3,3,,

DIM D1= LOCATION OF POINT CS1:F1 UNITS=IN,$

GRAPH=OFF TEXT=OFF MULT=10.00 OUTPUT=BOTH

AX NOMINAL +TOL -TOL MEAS MAX MIN DEV OUTTOL

X 3.0000 0.0000 0.0000 3.0000 3.0000 3.0000 0.0000
0.0000

----#----

END OF DIMENSION D1
```

Basic scripts create and delete objects dynamically. Use the syntax "<Basic Script ID>:<Feat ID>" to refer to a feature created by a basic script. For example, if a basic script with ID "BS1" creates a feature with ID "F2", use the ID expression "BS1:F2" to refer to that feature.

You can use the attach command to attach external routines to PC-DMIS. To refer to features in the attached routine, use the following syntax: "<Attach Routine ID>:<Feat ID>". To refer to feature "F3" in the attached measurement routine with ID "GEAR1", use the expression, "GEAR1:F3". (For more information, see "Attaching an External Measurement Routine" in the "Adding External Elements" chapter.)

ID Expression Combinations

You can use array ID Expressions, Wild Card ID Expressions, and external subroutine, basic script, and external measurement routine ID Expressions in combination. For example, to refer to the third instance of all features that start with the letters "CIR" in an external measurement routine attached with the ID "BOLTPAT", use the ID expression " `BOLTPAT:CIR*[3]` ".

Also, you can use ID Expressions in regular expressions. Thus, the measured centroid of the above set of features could be assigned to variable with the following expression:

```
ASSIGN/V1=BOLTPAT:CIR*[3]. XYZ
```

Also, ID Expressions can be used in regular expressions. Thus, the measured centroid of the above set of features could be assigned to variable with the following expression:

```
ASSIGN/V1=BOLTPAT:CIR*[3]. XYZ
```

Accessing a Report's Object Properties

You can create your own custom report and label templates. PC-DMIS uses them to display report data inside a Report window (see **View | Report Window**). You use template editors to create the templates. The editors utilize a Visual Basic-like interface that you can use to insert, relocate, and size special components called "objects".

Each object consists of "properties" that define how it appears and what information it holds. Some of these properties are common to all other objects, some are common to only related objects, and others are unique to that specific object.

The PC-DMIS Expression Language can query the currently loaded report and store property values of a particular object in a variable. It can obtain values of type String, Integer, and Real by using this syntax:

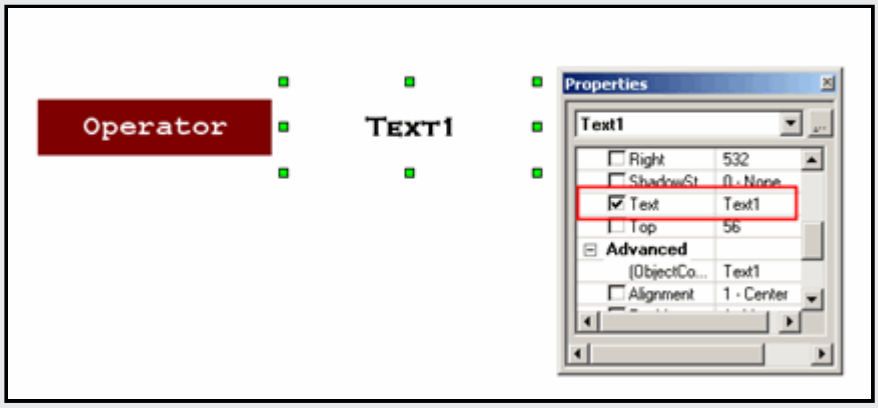
Property Query Syntax

```
ASSIGN/V1=Report.<Object Name>.<Property Name>
```

Report is a reference to the currently loaded report. <Object Name> is the object's unique name. <Property Name> is a valid property name for that object.

Example

Suppose that your report template has a text object called "Text1" that you want to use in the final report to display the operator's name. PC-DMIS stores the actual string of characters that represent the operator's name in the **Text** property of the object. By default, the text property (displayed text) initially has the value of "Text1" (see the figure below). Because this is a user-assigned property, the property's value changes when you type in the name during execution.



Properties dialog box that shows a selected object and the property to query

To use the Expression Language code to query this Text object's "Text" property and obtain the keyed-in data, you would use the following command:

```
ASSIGN/V1=Report.Text1.Text
```

In this code:

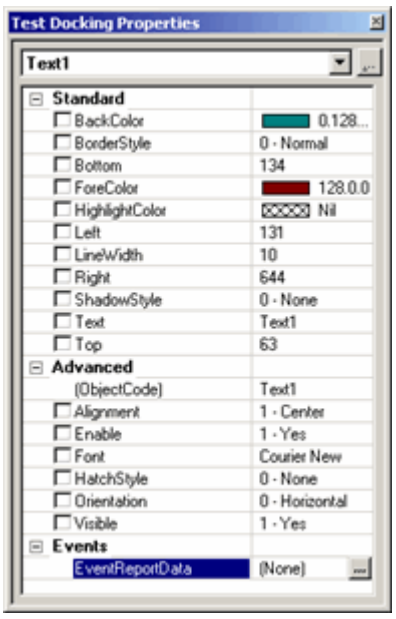
"Report" tells the code to look at the report loaded in the Report window.

"Text1" tells it to look for the object named "Text1".

"Text" tells it to look for the "Text" property within that object. The value of the "Text" property then gets passed into the V1 variable, which you could then further manipulate or display using the PC-DMIS Expression Language.

Finding Properties

To find the properties that are associated with a particular object, access the Report template in the Report template editor (**File | Reporting | Edit | Report Template**), select the object, and then right-click on the object to display its property sheet.



Property sheet for a Text object

The property sheet contains two columns. The left column displays the property name. The right column displays the current value. Be sure to use the exact property name in your expression code.

When you query property values, you may find that some properties return a seemingly useless numerical value. Generally, this happens when the property has a set list of available options. PC-DMIS returns an internal value for the selected property that does not relate to the displayed property.

For example, the **Text** object has an **Orientation** property with these values:

- 0 - Horizontal
- 1 - Vertical up
- 2 - Vertical down

However, if you obtain the value using PC-DMIS expression language, the software will instead return the following:

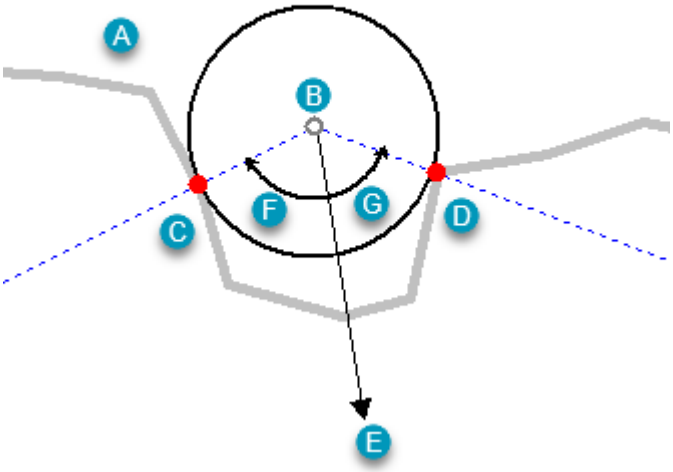
- 0 (for Horizontal)
- 900 (for Vertical up)
- -900 (for Vertical down)

It may require some trial and error to determine what returned values correspond with the value displayed on the property sheet.

Accessing Information from a Constructed Scan Minimum Circle

You can use PC-DMIS expressions to pull information from a circle feature that is constructed with a given radius at a minimum point along a linear scan. See the "Constructing a Circle at a Scan's Minimum Point" topic in the "Constructing New Features from Existing Features" chapter for more information.

When you construct a scan minimum circle feature, the circle ultimately uses a vector (termed the down vector) to contact the scan line. It only contacts the line in two places called contact points (CONTACTPOINT1 and CONTACTPOINT2). PC-DMIS can then use these points to determine the angles from the down vector to these contact points (CONTACTANGLE1 and CONTACTANGLE2). For example, consider this diagram:



A *- The scan line to which the circle is constructed.*
B *- The final XYZ position of the circle's centroid.*
C *- The contact point to the left of the Down Vector. It is termed CONTACTPOINT1.*
D *- The contact point to the right of the Down Vector. It is termed CONTACTPOINT2.*
E *- The Down Vector.*
F *- The angle from the Down Vector to CONTACTPOINT1. It is termed CONTACTANGLE1.*
G *- The angle from the Down Vector to the CONTACTPOINT2. It is termed CONTACTANGLE2.*

The expressions detailed below only function with this type of constructed circle feature. You can also use CONTACTPOINT2 in the syntax below to return the equivalent information using the second contact point instead.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . XYZ
```
Returns the XYZ point information for the circle's first contact point with the line, CONTACTPOINT1.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . X
```
Returns the X information for CONTACTPOINT1.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . Y
```
Returns the Y information for CONTACTPOINT1.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . Z
```
Returns the Z information for CONTACTPOINT1.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . IJK
```
Returns the IJK vector from CONTACTPOINT1 to the circle's centroid.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . I
```
Returns the I value from the above CONTACTPOINT1 IJK vector.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . J
```
Returns the J value from the above CONTACTPOINT1 IJK vector.

```
ASSIGN/V1=CIR1. CONTACTPOINT1 . K
```
Returns the K value from the above CONTACTPOINT1 IJK vector.

```
ASSIGN/V1=CIR1. CONTACTANGLE1
```
Returns the angle from the Down Vector to CONTACTPOINT1.

```
ASSIGN/V1=CIR1. CONTACTANGLE2
```
Returns the angle from the Down Vector to CONTACTPOINT2.

```
ASSIGN/V1=CIR1. CONTACTANGLE
```
Returns the sum of the absolute values of CONTACTANGLE1 and CONTACTANGLE2. It should be no greater than 180 degrees.

Adding External Elements

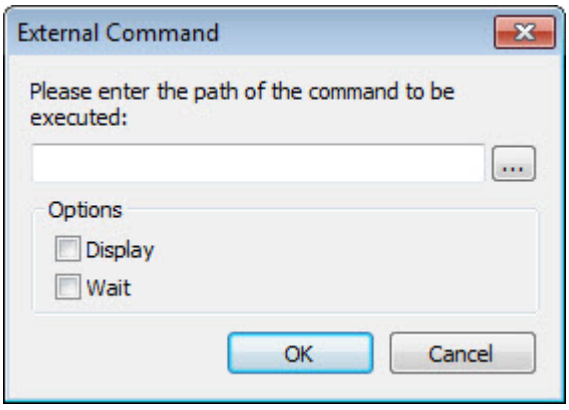
Adding External Elements: Introduction

This chapter discusses the various external elements that you can insert into your measurement routines. These include external applications, BASIC scripts, measurement routines, and other objects that further enhance your measurement routine's capability.

The main topics discussed in this chapter include:

- Inserting an External Command
- Inserting BASIC Scripts
- Attaching an External Measurement Routine
- Inserting External Objects

Inserting an External Command



External Command Dialog Box

Use the **Insert | External Command** menu option to insert a command into the Edit window that, when marked and executed, runs an external executable or batch file from the measurement routine.

- Normal DOS commands can be executed if they are put into a batch file.
- The command must be marked in the Edit window for PC-DMIS to execute the external command.
- A valid path and file name must be used.

- PC-DMIS halts the execution of the measurement routine and displays a message when it encounters an `EXTERNALCOMMAND/DISPLAY` command during execution. Click **OK** to continue measurement routine execution.

To insert an External Command

Select the **Insert | External Command** menu option. The **External Command** dialog box appears.

1. Specify an external command in the dialog box. To do this, either type the complete path for the file into the available box, or use the **...** button to find the file.
2. Select the **Display** check box if you want PC-DMIS to display a message informing you that measurement routine execution is paused when running an external command. Execution remains paused until you click **OK** on the message. See "Display Check Box" topic for more information.
3. Click **OK**. The command gets inserted into the Edit window.

The Edit window command line for this option reads:

```
EXTERNALCOMMAND/DISPLAYSTATE ; path name
```

DISPLAYSTATE - This toggle field controls whether or not PC-DMIS pauses execution and displays a message notifying you of an external execution. This field switches between `DISPLAY` and `NO DISPLAY`.

path name - This string represents the path and file name of the executable or batch file.

Display Check Box

If you select the **Display** check box in the **External Command** dialog box (**Insert | External Command**), PC-DMIS displays a message that lets you know that routine execution is paused in order to run the external command. PC-DMIS pauses execution until you click **OK** on the message.

Note that this message only appears if you have turned it on inside the **Warnings Display Options** dialog box.

To get this message to appear:

1. Press F5 to access the **Setup Options** dialog box.
2. From the **General** tab, click the **Warnings** button. The **Warnings Display Options** dialog box appears.

3. Select the **OK Execution paused in order to spawn a process** check box. Select **OK** to continue execution.

If you do not select the **Display** check box, PC-DMIS runs the specified external command without displaying any sort of message. PC-DMIS continues execution while the executable or batch file runs simultaneously. When not checked, the **Wait** check box becomes enabled. For more information, see "Wait Check Box".

Wait Check Box

The **Wait** check box in the **External Command** dialog box (**Insert | External Command**) is only available when the **Display** check box is not selected.

- When the **Wait** check box is selected, the execution of the measurement routine is paused until the external command finishes its operation.
- When the **Wait** check box is not selected, the measurement routine continues execution even if the external command hasn't finished its operation.

... button

The **...** button in the **External Command** dialog box (**Insert | External Command**) displays an **Open** dialog box. You can select the file name for the external command. Once you open the file, PC-DMIS inserts the full path into the **External Command** dialog box.

Creating an External Command as a Menu Item or Toolbar Item

PC-DMIS enables you to customize toolbars and menus to accept new menu items tied to .EXE, .BAT, and .BAS files. For information on how to do this, see the "Customizing the User Interface" topic in the "Navigating the User Interface" chapter.

Example - Using the External Command to Display a File

This example shows how to create a batch file (an external program) that works with the `EXTERNALCOMMAND` to open and display a file. For example, suppose you want to display an image file, but you don't want to use the **Insert | Report Command | External Object** menu option. You can use a command line (or DOS) prompt inside of a batch file to do this.

The advantage or disadvantage (depending on your needs) to this approach is that the image does not appear in the report at the end of measurement routine execution.

Step 1: Create a batch file to display the image:

1. Open a text editor, such as Notepad. On the first line, enter this command:

   ```
   start <PATHWAY>
   ```

   where <PATHWAY> is the complete path to the picture (for example, start d:\temp\mypart.gif).

   This command tells Windows to open the image file using its default image viewer application.

2. Save the file and give it a .bat filename extension.

Step 2: Link the External Command dialog box to the batch file:

1. Select **Insert | External Command**. The **External Command** dialog box appears.
2. Click the **...** button. An **Open** dialog box appears.
3. Under **Files of Type** list at the bottom of the **Open** dialog box, change the type to **BAT Files (*.bat)**.
4. Navigate to and select your batch file.
5. Click the **Open** button. The **Open** dialog closes, and the **External Command** dialog box now contains the path to the batch (.bat) file.

Step 3: Control what happens, and insert the command:

1. Choose either to display or not display a message that pauses the execution of the measurement routine. See "Display Check Box" for more information.

   - Select the **Display** check box from the **External Command** dialog box to display a message.
   - Clear the **Display** check box from the **External Command** dialog box to not display a message. When not checked, the **Wait** option becomes enabled. See "Wait Check Box" topic for more information.

2. Ensure that the **Link** check box is cleared.
3. Click **OK**. The `EXTERNALCOMMAND` gets inserted into the Edit window.

   ```
   EXTERNALCOMMAND/DISPLAY ; E:\BATCH\TEST.BAT
   ```

Step 4: Execute the routine:

1. Mark the resulting command line in the Edit window.
2. Execute your measurement routine.

PC-DMIS runs the specified batch program, displays the picture and, depending on what you selected in the **External Command** dialog box, it either pauses or continues the measurement routine while you view the picture.

Inserting BASIC Scripts

The Basic Language Extension to PC-DMIS provides a powerful extension to the software's functionality. Basic language scripts or applications may be written from within PC-DMIS (or imported from elsewhere) and linked to a button on a user-definable toolbar, allowing the execution of powerful macros. The version of Basic featured in supported versions of PC-DMIS provides all of the features of a high-level language, including custom dialogs (created using the built in Dialog Editor), ODBC support, and OLE support.

The ability to create and use basic scripts comes as a standard feature in PC-DMIS CAD and PC-DMIS CAD++. It is not available in PC-DMIS Pro.

The Edit window command line for a sample script reads:

```
 SCRIPT/FILENAME = C:\PCDMISW\sample.bas
FUNCTION/Main,SHOW=YES,,
ENDSCRIPT/
```

The **FILENAME=** field lets you specify the pathway to the BASIC file (.bas filename extension) that you want to insert and execute with the measurement routine.

The **FUNCTION/Main** field runs the "Main" subroutine. You can change this to specify a different subroutine or function in the BASIC file to run.

The **SHOW=** field lets you determine whether or not commands automatically generated by your Basic Script appear in your measurement routine after execution.

- When you set SHOW=NO, generated commands will not appear in Summary Mode, Command Mode, or DMIS Mode (commands still appear in the inspection report, however). Also, PC-DMIS will not save any generated commands with the measurement routine.
- When you set SHOW=YES, then generated commands appear in the measurement routine and PC-DMIS will save generated commands with the measurement routine. The default is SHOW=YES for measurement routine compatibility from previous versions.

The PC-DMIS *Basic Language Reference Manual* completely describes this add-on package. If you did not receive a copy of the documentation with your Basic Language package, contact your PC-DMIS software support representative.

To Insert a Basic Script as a Command

The **Insert | Basic Script** menu option displays the **Insert Basic Script** dialog box.

You can use this dialog box to add a basic script object to the measurement routine. Basic script objects contain the name of the basic script that should be executed when the basic script object is executed. Execution of the measurement routine does not continue until the basic script has executed. If the basic script creates any objects while executing, those objects will be inserted into the measurement routine and executed.

Objects inserted by Basic Scripts are highlighted in a different color than other objects to indicate that the basic script created them. For more information on basic scripts, see the PC-DMIS BASIC documentation.

To Insert a Basic Script as a Menu or Toolbar Item

You can use PC-DMIS to customize toolbars and menus to accept new menu items tied to .EXE, .BAT, and .BAS files. For information on how to do this, see the "Customizing the User Interface" topic in the "Navigating the User Interface" chapter.

Passing Variables To and From BASIC Scripts

From PC-DMIS code, you can only pass variables *to* BASIC scripts and not *from* BASIC scripts.

The only variable types that you can pass into BASIC scripts from PC-DMIS are:

- Integer
- String
- Double

Variables are passed as comma-separated values (known as arguments). You define these in the FUNCTION/ statement within the SCRIPT/FILENAME code block. The FUNCTION/ statement specifies not only the arguments, but also the subroutine within the script that takes the passed values. By default, this line initially reads:

```
FUNCTION/Main,SHOW=YES,,
```

This means that the subroutine named "Main" takes the arguments. `SHOW=YES` shows or hides the closing `ENDSCRIPT/` statement for the command block.

Notice the two commas following `SHOW=YES` above. These indicate a field where you may define your first argument. Type the value in the field and press the Tab key on your keyboard. PC-DMIS automatically adds "ARG1=" before the first value to indicate that it's the first argument. An additional comma is also added to allow for additional arguments. Any additional arguments have "ARG2=" and "ARG3=" and so on added as prefixes once you define the values.

Consider the examples below.

PC-DMIS variables only hold values during measurement routine execution; at learn time, PC-DMIS variables always have a value of zero.

Example 1: Using the Function Line to Pass Variables

The following command executes a BASIC script named TEST.BAS. It also, upon execution, passes the variables defined from the `FUNCTION/` line into the TEST.BAS script:

```
CS2=SCRIPT/FILENAME= D:\PROGRAM FILES\PCDMIS35\TEST.BAS
FUNCTION/ShowVars,SHOW=YES,ARG1=3,ARG2="Hello",ARG3=2.5,,
STARTSCRIPT/
ENDSCRIPT
```

Now here's the TEST.BAS script; on execution, it displays the passed in variables in their respective message boxes:

```
Sub ShowVars(IntVar As Integer, StrVar As String, DoubleVar
As Double)
   msgbox "The passed integer variable is " & IntVar
   msgbox "The passed string variable is " & StrVar
   msgbox "The passed double variable is " & DoubleVar
End Sub
```

Example 2: Using the GetVariableValue and SetVariableValue Methods to Pass Variables

The following example first uses PC-DMIS code to receive an integer value from the user and assigns it to the V1 variable.

```
C1=COMMENT/INPUT,Please type an integer value.
ASSIGN/V1=INT(C1.INPUT)
COMMENT/OPER,BEFORE SCRIPT: Variable is:
,V1
```

It then calls a BASIC script named TEST2.BAS.

```
CS1=SCRIPT/FILENAME=D:\PROGRAM FILES\PCDMIS35\TEST2.BAS
FUNCTION/Main,SHOW=YES,,
STARTSCRIPT/
ENDSCRIPT/
```

Here is TEST2.BAS:

```
Sub Main
  Dim App As Object
  Set App=CreateObject("PCDLRN.Application")
  Dim Part As Object
  Set Part=App.ActivePartProgram
  Dim Var As Object
  Set Var=Part.GetVariableValue("V1")
  Dim I As Object
  If Not Var Is Nothing Then
    Var.LongValue=Var.LongValue+1
    Part.SetVariableValue "V1",Var
    MsgBox"V1 is now: "&Var.LongValue
  Else
    Msgbox"Could Not find variable"
  End If
End Sub
```

This script takes V1 variable and, using the `GetVariableValue` and `SetVariableValue` automation methods, increments the V1 by one and then sets the new value for V1 in the measurement routine.

PC-DMIS then displays the changed variable in an operator comment.

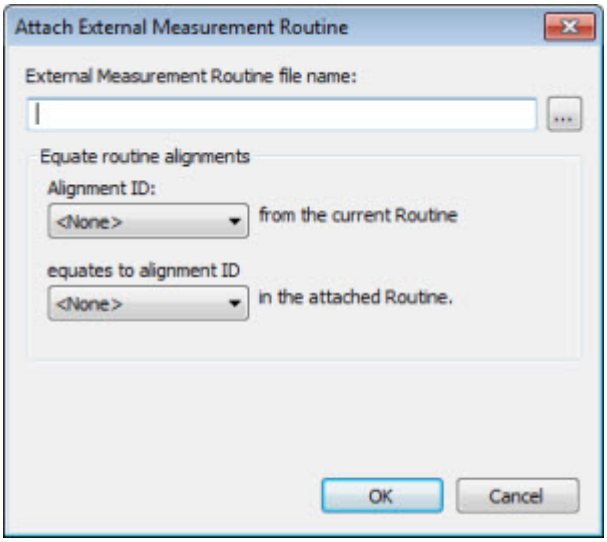
```
COMMENT/OPER,AFTER SCRIPT:Variable is now
,V1
```

Attaching an External Measurement Routine

When you attach an external measurement routine, you can only attach measurement routines created in valid versions of PC-DMIS.

For this version of PC-DMIS, the measurement routine must be created in PC-DMIS version 2021.1 up to this version of PC-DMIS.

The **Insert | Attach Measurement Routine** menu option opens the **Attach External Measurement Routine** dialog box. You can use this dialog box to attach a measurement routine to the current measurement routine. This does not mean that PC-DMIS actually attaches and executes all the command in the specified measurement routine. Instead, it attaches a pointer to the measurement routine, which enables you to access its dimensional and feature data. See "Using a Pointer to Reference Data".



Attach External Measurement Routine Dialog Box

This dialog box contains these options:

External Measurement Routine file name
Use this box to type the path for the measurement routine you are going to attach. If you prefer, you can use the **Browse** button (...), which also places the file name path into the box.

Equate Routine Alignments
Use this area to share alignments between two measurement routines by selecting them from the available lists. See "Equating an Alignment" in the "Creating and Using Alignments" chapter for additional information.

The **Attach Measurement Routine** menu option is especially useful when working in multiple arm mode. With this option, you can attach a measurement routine from the Arm2

system onto the Arm1 measurement routine. Attaching the measurement routine allows the sharing of data so that both arms can run off the same alignment, pass feature data from one measurement routine to another for calculations, and generally work in a cooperative manner. (See the "Using Multiple Arm Mode" chapter for more information on multiple arm mode.)

Using a Pointer to Reference Data

When you access an attached measurement routine, you often want to use data from that measurement routine. When you reference this data, you *must* reference the pointer (or variable) that contains the location of your measurement routine.

Suppose you're attaching TEST_A.prg to your current measurement routine. Your attach command would look something like this:

```
CS1=ATTACH/C:\PCDMISW\TEST_A.PRG, Machine=

EQUATE/LOCAL ALIGNMENT = A1, ATTACHED ALIGNMENT = A1
```

Notice the pointer, `CS1` . This pointer is used to extract data from the attached measurement routine.

Now, suppose you wanted to display the measured X value for feature F1 from TEST_A.prg inside your current measurement routine. You would use a statement like this.

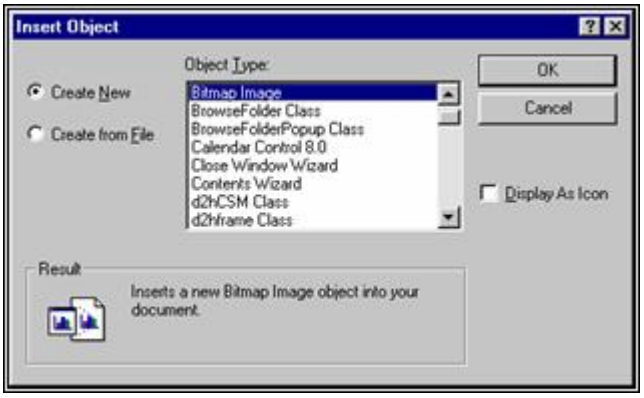
```
COMMENT/OPER,"The X value for F1 from the attached
measurement routine is:"

,CS1:F1.X
```

The code, `CS1:F1.X` , essentially tells PC-DMIS to look at TEST_A.prg, find feature F1, and display the X value. This is the way pointers are referenced inside PC-DMIS.

PC-DMIS also lists features from an attached measurement routine in the Construction or Dimension dialog box. PC-DMIS will display the attached measurement routine’s ID pointer within the Feature List box. A plus sign (+) appears to the left of the pointer. Clicking this plus sign expands or collapses a view of all the features in the attached measurement routine. After you expand the view to show all the features of the attached measurement routine, you can select any of those features to use in the construction or dimension process.

You cannot select the attached routine’s pointer. You can only select the expanded IDs associated with the pointer.

Inserting External Objects



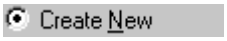
Insert Object Dialog Box

The **Insert | Report Command | External Object** menu option enables you to enter various types of objects from other applications on your system into the Edit window. Types of objects that can be inserted depend on the applications installed on the computer system. Examples of objects that can be inserted include graphics, sound clips, movie clips, midi clips, documents, worksheets, and database tables.

Use external objects to import an instructional video, audio instructions, or written instructions into your measurement routine.

Be sure to place the Edit window in Command Mode when working with external objects.

Create New



The **Create New** option (**Insert | Report Command | External Object**) enables you to create a new file of a particular object type at a specified location in the Edit window. The object types are chosen from the **Object Type** list. Objects can only be placed just after or just before a feature in the measurement routine.

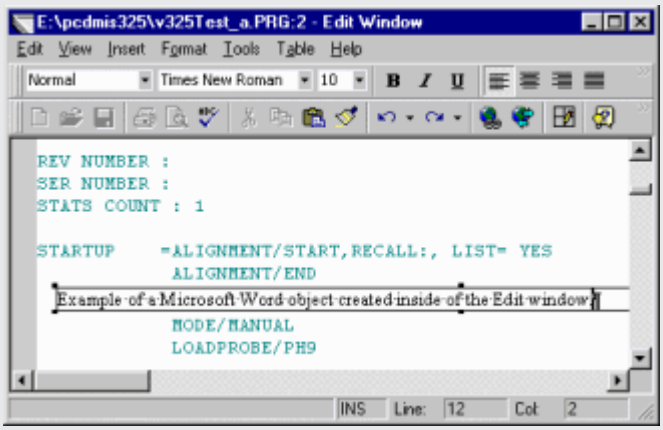
To create a new object and place it in the Edit window:

1. Select the **Create New** option.
2. Select the desired object type from the **Object Type** list. The object will appear in a box, bounded by small rectangular points within your Edit window. Use your mouse to drag the box to another location or resize the box using the small rectangles.
3. Double-click within the object**.** You will see that the application that runs that particular object opens in the Edit window.

4. Modify the object by using the tools specific to the inserted application.
5. When you are done, click on the portion of the Edit window that is *outside* of the inserted object.

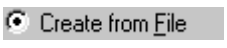
Suppose you want to include a *new* word document in the Edit window:

1. Select the **Create New** option.
2. Select a word document object from the **Object Type** list.
3. Drag the object box with the mouse to a desired location within the Edit window.
4. Resize the box to the desired size.
5. Type the information within the word document object.
6. Click outside of the object to return to normal PC-DMIS Edit window functions. The word document will remain in the Edit window.



Example of a word document created in the Edit window. Notice how the Edit window's menus and toolbars are replaced with the menus and toolbars of the application that corresponds to the object you are creating.

Create from File



The **Create from File** option (**Insert | Report Command | External Object**) enables you to insert a previously-created file as an object into the Edit window.

To insert a previously-created object:

1. Select the **Create from File** option.

2. Type the directory path that contains the file you want to place as an object. You can also click the **Browse** button to navigate to the correct directory.
3. Select the file to insert as an object.
4. Click the **OK** button. The object now appears in the Edit window. Use your pointer to drag the box to another location or resize the box using the small rectangles.

Suppose you want to insert a previously-written word document into the Edit window that includes a set of instructions about operating the CMM.

To do that:

1. Select the **Create from File** option.
2. Navigate to the directory that contains the document.
3. Click the **OK** button. The document appears within the object box in the Edit window.
4. Use your pointer to move the object box to where you want it in the Edit window and resize it.

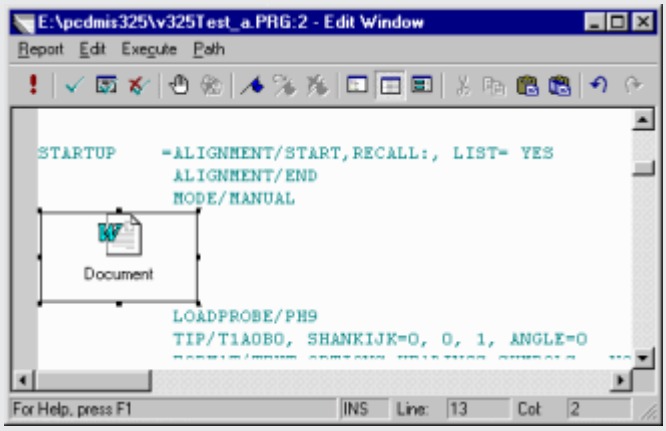
Link

When you select the **Create from File** option (**Insert | Report Command | External Object**), you have the opportunity to "link" your object to the Edit window. When you link your object back to its original file, any changes made to the original file are also applied to the external object in the Edit window.

Display As Icon

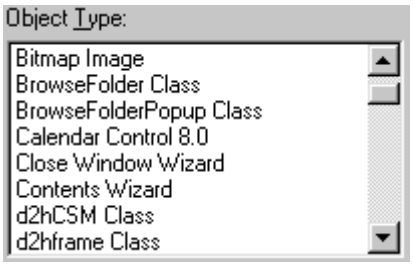
The **Display As Icon** check box (**Insert | Report Command | External Object**) enables you to display the imbedded object as an icon instead of displaying the information outright. Once you double-click on the icon within the Edit window, it will then activate.

Suppose you place a word document in the Edit window and select the **Display As Icon** check box. The word document appears as an icon. However, if you double-click on the icon, the imbedded application opens and displays the information contained in the word document.



Example of a word document object displayed as an icon

Object Type List



Object Type List

The **Object Type** list (**Insert | Report Command | External Object**) contains the object types available on your computer system. These vary from computer to computer depending on what application programs you have installed on your hard drive.

To choose an object type:

1. Use the scroll bar or UP ARROW and DOWN ARROW keys to find the object.
2. Select the object.
3. Click the **OK** button.

Marking External Objects for Execution

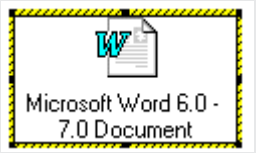
By default, external objects print and do not execute when you execute a measurement routine. However, with some objects, the desired action may be to execute and not print. For example, when embedding a graphic, the desired action may be to print the graphic to the report; while when embedding a sound or movie clip, the desired action may be to execute the object to play the clip at execution time.

External objects have four modes of execution:

1. Print - Don’t execute
2. Don’t print - Don’t execute
3. Don’t Print - Execute
4. Print - Execute

To switch between the different modes, press F3 while the insertion point is on the same line as the external object in the Edit window. PC-DMIS will use hash lines and borders to indicate the objects current mode of execution. See the examples below.

- Hash lines - diagonal lines that cross the graphic. When there are hash lines, the object will not print to the report.
- Hashed borders - diagonal lines that cross the border around the graphic. When a hashed border outlines the object, the object will execute.

Example	Behavior
No Hash Lines - No Hashed Border 	• Print • Don't Execute
Hash Lines - No Hashed Border 	• Don't Print • Don't Execute
Hash Lines - Hashed Border 	• Don't Print • Execute
No Hash Lines - Hashed Border 	• Print • Execute

The execute action is always the same action that occurs when an external object is double-clicked. For media clips, this default action is usually *play*. For most other objects, the default action is *edit*.

Play - Sound clips will play sound. Movie clips will play their movie, etc.

Edit - Objects that have a default action of edit will usually not be objects that would need to be executed. However, some application objects also have the ability to run custom basic scripts upon being activated. A Microsoft Word document or Excel spreadsheet are examples of this type of external object. With these types of objects and the PC-DMIS automation commands, it is possible to change the object using data from the measurement routine. For example, an Excel graph could be inserted inside a measurement routine that has a basic script that, when activated, would pull data out of the measurement routine using the PC-DMIS automation commands and adjust the graph accordingly before it was printed on the inspection report.

Displaying Files in a Different Way

An alternate way to display files is to use the PC-DMIS `EXTERNALCOMMAND` Edit window command. See "Example - Using the External Command to Display a File" for information.

Using Multiple Arm Mode

Using Multiple Arm Mode: Introduction

The Multiple Arm Mode was created to drive multiple arm CMMs. The Multiple Arm Mode is available as an add-on package. Currently, this version only supports a multiple arm mode of two arms. The two arms must be connected to separate computers with a version of PC-DMIS and the correct PC-DMIS license on each computer. These arms share a common alignment.

Later versions of PC-DMIS enable you to drive up to four machine arms from one version of PC-DMIS, using any combination of one to four computers.

While the measurement process for multiple arms is comparable to single arm units, PC-DMIS must be able to distinguish the arm that is taking the measurement when using a multiple arm system. The topics in this chapter explain how to do this.

The main topics in this chapter describe how to setup a multiple arm CMM, create measurement routines using multiple arms, and share the measurement routine among the participating arms. These topics are:

- Setting up a Multiple Arm CMM
- Creating a Measurement Routine Using Multiple Arm Mode
- Running the Arm1 Measurement Routine on Arm2
- Dialog and Message Boxes in Multiple Arm Mode
- Dual Arms with Wrists Calibration Example

PC-DMIS must be installed on all systems prior to running in Multiple Arm Mode.

In addition, the **World Axis** value in your PC-DMIS license on each machine must be configured appropiately to enable operation in Multiple Arm Mode, as follows:

- The primary computer's PC-DMIS license setting should detail how many secondary arms that will be driven by the primary computer. So, a primary computer driving three secondary arms would need to specify a value of 3.
- Each secondary computer's LMS license needs to be properly configured or the portlock should specify a value of 1.

Setting Up a Multiple Arm CMM

To setup a multiple arm CMM, follow these steps:

Step 1: Install PC-DMIS on All Computers

The first step needed to make this option accessible is to install PC-DMIS on all the computers that will drive the multiple arms. (See the documentation on software installation procedures if necessary.)

The Multiple Arm option must be programmed in all port locks. This should be available on all computers.

Step 2: Determine the Arm1 System

Determine the Arm1 system. In most cases, any arm controller can be used. However, if the multiple arm system has a rotary table, then the Arm1 system must be the controller that controls the rotary table.

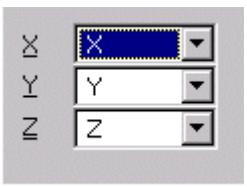
Label the participating arms in some way. Most users usually call them "ARM1" and "ARM2".

Step 3: Match the CMM Axes for Each Arm

The CMM axes for each arm must match. That is, the X+, Y+, and Z+ axes for all arms must go in the same direction.

If you need to change the axes assignments and directions:

1. Make sure you're running PC-DMIS in Online mode.
2. On the Arm2 computer, select **Edit | Preferences | Machine Interface Setup**. The **Machine Options** dialog box appears.
3. Select the **Axes** tab. This shows the X,Y, and Z axes combination boxes.



4. Use the **X**, **Y**, or **Z** list to reassign the axes of the connected CMM so that they match the axes of Arm1. Usually, only the X and Y axes will need modification.

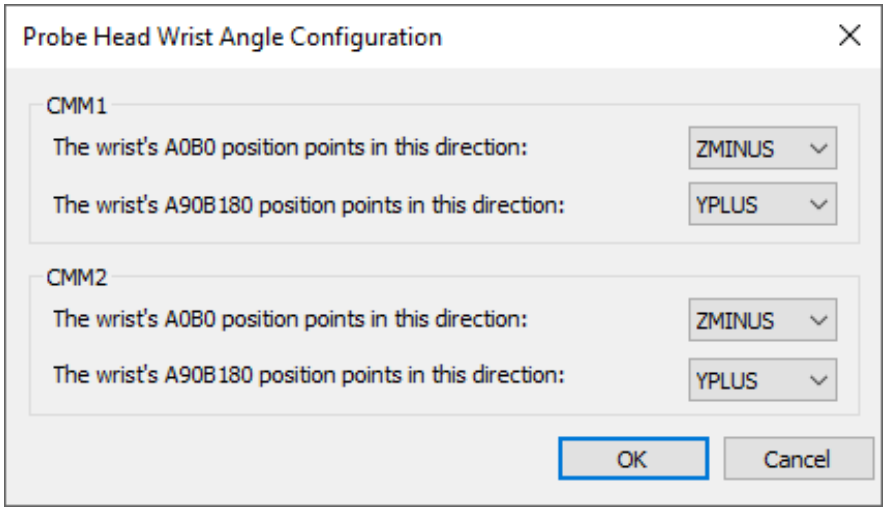
5. Click **Apply** to keep the changes.
6. Once the dialog box closes, exit PC-DMIS.
7. Restart PC-DMIS, and verify that the changes are correct by moving the axes for both arms in the same directions. Make sure the axis counters increment accordingly.

The numerical values in the counters will not match until you complete the multiple arm calibration.

Step 4: Configure Probe Head Mount Orientation

Once PC-DMIS is loaded on all systems, and the axes match for both arms, configure your probe head mount orientation:

1. Select **Edit | Preferences | Setup** to display the **Setup Options** dialog box.
2. Select the **Part/Machine** tab.
3. Click the **Probe Head Orientation** button. The **Probe Head Wrist Angle Configuration** dialog box appears.

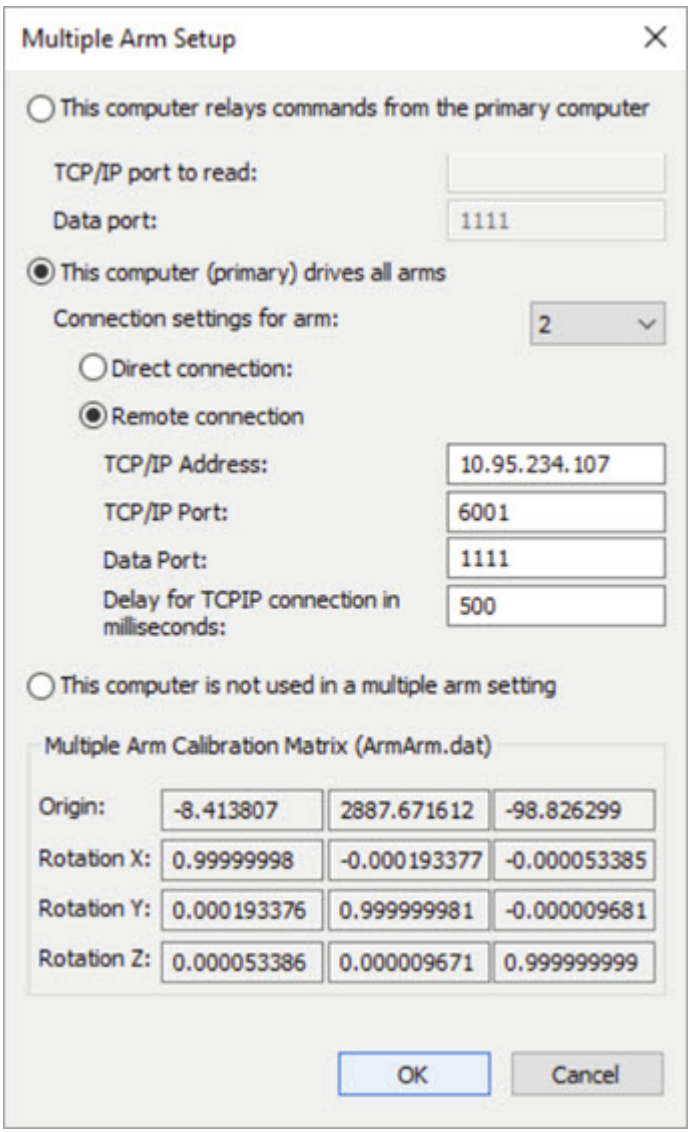


Probe Head Wrist Angle Configuration Dialog Box

4. Change the mount orientation as needed for each CMM arm.

Step 5: Setup the Multiple Arm Connection

The next step is to set up the multiple arm mode for the current computer. Select **Edit | Preferences | Multiple Arm Setup**. This menu option displays the **Multiple Arm Setup** dialog box.

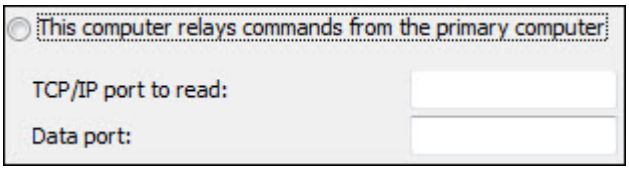


Multiple Arm Setup Dialog Box

Use this dialog box to determine if the current computer is the primary computer and drives the other arm or arms, or if the current computer is not the computer but relays commands from the primary computer.

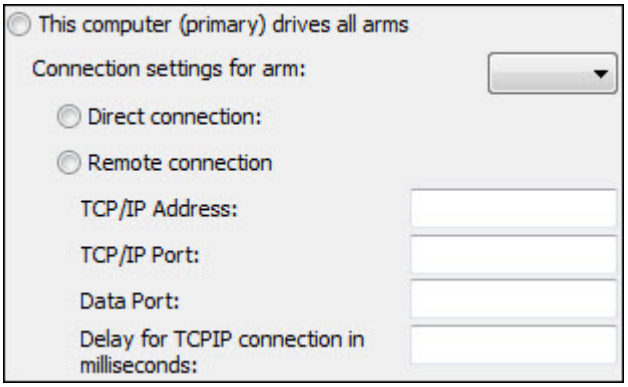
When you finish making changes to this dialog box and click **OK**, PC-DMIS displays a warning message that says that you need to restart PC-DMIS for your changes to take effect.

This computer relays commands from the primary computer



The **This computer relays commands from the main computer** option in the **Multiple Arm Setup** dialog box (**Edit | Preferences | Multiple Arm Setup**) enables the computer to be used with a non-primary arm (Arm2 or higher). In this case, it relays commands from the primary computer. You can choose to connect to the primary computer by defining the proper TCP/IP port and data port.

This computer (primary) drives all arms



The **This computer (primary) drives all arms** option in the **Multiple Arm Setup** dialog box (**Edit | Preferences | Multiple Arm Setup**) enables the computer to be the main computer that drives all the arms. You can choose to connect the computer to the other arms using either a direct connection or a remote connection.

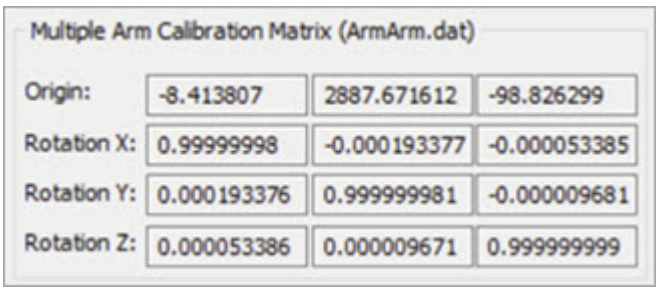
If you choose the **Remote connection** option, you need to fill out the available fields to establish the connection:

1. Select the arm from the **Connection settings for arm** list.
2. Type the IP address in the **TCP/IP Address** box.
3. Type the IP address's port number in the **TCP/IP Port** box.
4. Type the data port number in the **Data Port** box.
5. Specify the TCP/IP connection delay in the **Delay for TCPIP connection in milliseconds** box. This value is the number of milliseconds that PC-DMIS delays before attempting a TCP/IP connection to the specified arm from the current computer.

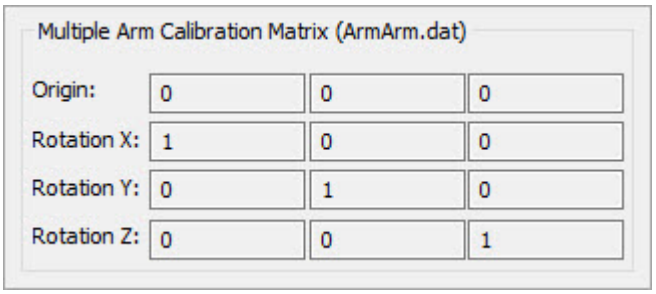
This computer is not used in a multiple arm setting

If the computer is not used in a multiple arm setup, select the **This computer is not used in a multiple arm setting** option in the **Multiple Arm Setup** dialog box (**Edit | Preferences | Multiple Arm Setup**).

Multiple Arm Calibration Matrix (ArmArm.dat)



This area displays the calibration data from the ArmArm.dat file which PC-DMIS creates after a multiple arm calibration. If PC-DMIS does not find an ArmArm.dat file, PC-DMIS populates this area with the identity matrix:



Each time you perform the multiple arm calibration routine, PC-DMIS updates the ArmArm.dat file with the new calibration values and populates the matrix with the new calibration data values.

Step 6: Connect the Computers

Now, you need to connect the computers so they can communicate with each other. You can either use a null modem serial cable between the computers, or, if the computers are connected to a network, you can use the network to communicate between them. The "Step 5: Setup the Multiple Arm Connection" topic enables you to configure these communication settings.

Once the computers are connected:

1. Start up PC-DMIS on the Arm2 and higher systems. Do *not* create a new measurement routine or activate a measurement routine. These systems are now ready.

2. Start up PC-DMIS on the Arm1 (primary) system. Create a new measurement routine (or activate one that is already available). If you are creating a new measurement routine, PC-DMIS automatically opens the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).

3. Either select or create a probe file that describes a probe on Arm1. Make sure that you add an AB angle for the tip you plan to use to calibrate the relationship between the two arms. *Don't calibrate the probe at this time*.

Step 7: Enter Multiple Arm Mode

Once you have set up the computers and are in a measurement routine, the **Operation | Enter Multiple Arm Mode** menu option becomes selectable from the Arm1 (primary) computer. Select this option.

PC-DMIS displays a check mark to the left of the **Operation | Enter Multiple Arm Mode** option. PC-DMIS also displays the **Active Arms** toolbar.

When PC-DMIS enters multiple arm mode, it attempts to establish a link between the multiple computer systems. This link coordinates the activities of all the arms.

Troubleshooting

If PC-DMIS cannot establish the link among the computers after it enters multiple arm mode, an error message appears and informs you of the arm that isn't responding. The following may cause this communication problem:

- PC-DMIS is not running on a computer
- An online measurement routine is active on an Arm2 or higher computer
- The cable connections (or other network connections) between the computers are not functioning
- Settings entered in the **Multiple Arm Setup** dialog box (**Edit | Preferences | Multiple Arm Setup**) are incorrect

Once you have a link between the multiple computers, you can calibrate the relationship between the arms.

Exiting the active measurement routine also removes PC-DMIS from multiple arm mode.

Step 8: Calibrate the Multiple Arm System

This step walks you through the calibration of the multiple arm system.

Prior to calibration and after PC-DMIS has been started on the computer systems, you must define any probes that will be used in the measurement process. PC-DMIS uses the theoretical data from the probes to calibrate the multiple arm system.

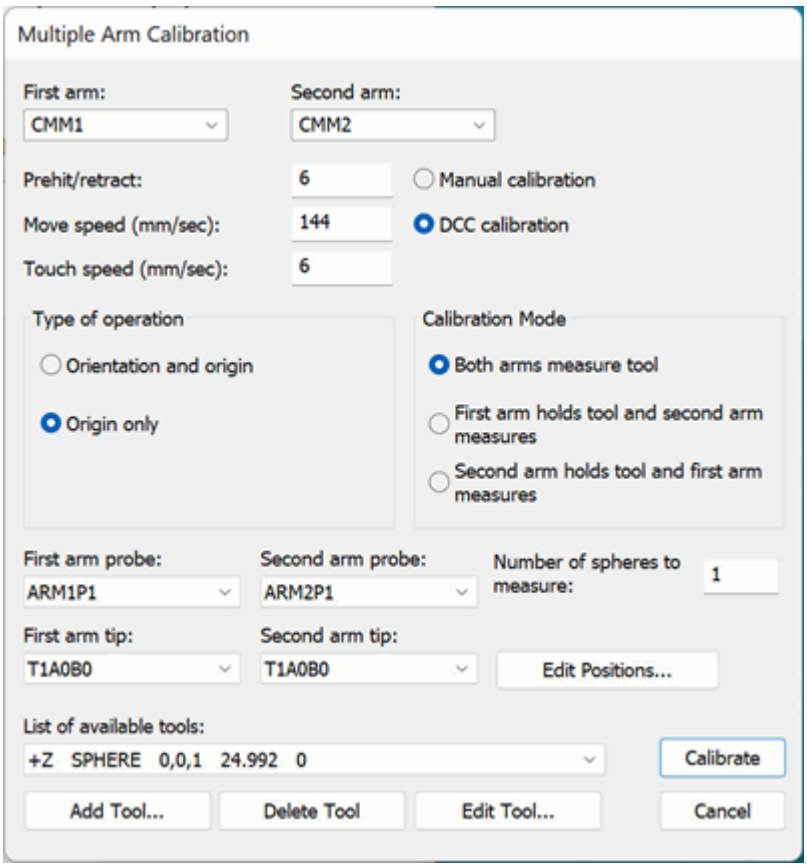
Do not calibrate the probes at this time. Only ensure they are properly defined and that you add an AB angle for the tip you plan to use to calibrate the multiple arm system.

In your measurement routine, you should have multiple `LOADPROBE` commands, one for each arm.

If you've attempted a multiple arm calibration previously then PC-DMIS has created *armtoarm.dat* files. It stores these where you installed PC-DMIS on both the Arm1 and Arm2 computers. You should delete or rename these files before proceeding so that data from previous calibration attempts doesn't negatively affect your current calibration process.

Follow these calibration procedures:

Part 1 of Calibration Procedure



Multiple Arm Calibration Dialog Box

1. Select the **Operation | Calibrate/Edit | Multiple Arm Mode** menu option (only available in Online mode). This displays the **Multiple Arm Calibration** dialog box.

2. Ensure that the correct probe files and tips are selected from the arm lists in the dialog box.

3. Ensure that the calibration tool you are using is available from the **List of available tools** list. The tool is only needed if you selected the **Both arms measure tool** option.

4. Define the **Prehit / Retract**, **Move speed**, and **Touch speed** values as desired. During calibration, these values supersede the same values listed in the **Measure Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button). In addition, they are shared by both arms to ensure that they operate using the same conditions during calibration:

    - **Prehit / Retract** - This defines a distance value away from the part or calibration tool. The speed of PC-DMIS decreases to the defined **Touch speed** while within this distance. It remains at the **Touch speed** until the hit is taken and the distance is reached again. At that point, PC-DMIS returns to the defined **Move speed**.

    - **Move speed** - This defines the speed that PC-DMIS uses to move the probe into position to take hits during multiple arm calibration.

    - **Touch speed** - This defines the speed that PC-DMIS uses to take hits during multiple arm calibration.

    Depending on the state of the **Display absolute speeds** check box on the Part/Machine tab in the **Setup Options** dialog box (**Edit | Preferences | Setup**), the above **Move speed** and **Touch speed** boxes can either accept an absolute speed (mm/sec) or a percentage of the machine's defined top speed.

5. Choose what you want to calibrate by selecting either the **Orientation and origin** option or the **Origin only** option.

    - Selecting the **Orientation and origin** option creates a 3D transformation between the two arms to compensate for any out-of-squareness between the two arms. You must do this *at least once* (normally, this is done periodically every few months).

    - Selecting **Origin only** corrects only the origin between the two arms. You should do this type of calibration more frequently, based on probe calibration procedures. When calibrating your probe, PC-DMIS asks if you moved your tool. If you normally indicate to PC-DMIS that you haven't moved the tool, then you don't have to adjust your arm-to-arm origin. If you answer that you have

moved the tool, then after calibrating your probe tips, you should return to this dialog box and choose the **Origin only** type of arm-to-arm calibration.

With **Origin only**, make sure you are using calibrated tips.

6. Choose how you want to perform the calibration by selecting either **Manual calibration** or **DCC calibration**.

   - If you selected the **Manual calibration** option, PC-DMIS prompts you to measure each sphere position using the CMM 's jog box. Once you have taken the first hit on the top of the sphere, PC-DMIS takes the rest of hits in DCC Mode.
   - If you selected the **DCC calibration** option, you are prompted to supply the sphere positions. Once entered, the computer performs the calibration's motion control.

     WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

     Click the **Edit Positions** button to change the location of each sphere position by typing the X, Y, and Z coordinates. You may find it helpful to read the position of the active arm to fill in these three values. Be aware that clicking the **Done** button on your jog box can read the arm's current position.

You must first do at least one origin arm-to-arm calibration in Manual Mode to establish the basic relationship between the two arms. When you complete an arm-to-arm calibration, PC-DMIS generates an *armarm.results* file stored in the directory defined through the **Set Search Path** menu option (see the "Specifying Search Paths" topic in the "Setting your Preferences" chapter). This text file is viewable using any text editor. It shows how beneficial the spheres are once you have done the initial fit. In particular, it displays the "fitment error".

Fitment Error: During calibration, PC-DMIS transforms values from one coordinate system to another, attempting to "fit" a plane through all the spheres measured by a given arm. The values of this "fitment" is what is stored in the armarm.results file. This information may help to show the overall accuracy of the calibration.

7. Type a number in the **Number of spheres to measure** box. This value determines the number of spheres PC-DMIS measures for each arm. If you type a number of spheres greater than 1, PC-DMIS averages the measurements to create the origin.

- If you use the **Manual calibration** option, PC-DMIS prompts you to manually measure these positions.
- If you use the **DCC calibration** option, PC-DMIS automatically drives each arm to measure these positions. The minimum number of spheres is three.

Ensure that the spheres aren't in the same position. Otherwise, the calibration finishes with incorrect results.

8. Use the available options below to determine how PC-DMIS uses the arms to measure the tools.

Both arms measure tool

- If you select this option and you use **Manual calibration**, PC-DMIS prompts you to measure each sphere position with both arms.
- If you select this option and you use **DCC calibration**, PC-DMIS drives both arms to measure a sphere at each of the positions defined in the **Edit Calibration Positions** dialog box. Be sure to have the desired number of sphere positions actually present on the CMM since there is not enough time to physically move the sphere around between measurements.

First arm holds tool and second arm measures

- If you select this option and you use **Manual calibration**, PC-DMIS prompts you to move Arm1 to each position and then measure the spherical tool with Arm2.
- If you select this option and you use **DCC calibration**, PC-DMIS moves Arm1 to each of the given calibration positions and then commands Arm2 to measure the sphere at that position. For this option, you need a special sphere mounted onto the end of the arm.

Second arm holds tool and first arm measures

- If you select this option while using **Manual calibration**, PC-DMIS prompts you to move Arm2 to each position and then measure the spherical tool with Arm1.
- If you select this option while using **DCC calibration**, PC-DMIS moves Arm2 to each of the given calibration positions and then commands Arm1 to measure the sphere at that position. For this option, you need a special sphere mounted into the end of the arm.

9. Click **Calibrate** once this button becomes enabled. It remains disabled until you select all needed calibration parameters.

   If you select **Both arms measure tool**, you must provide the following items before the **Calibrate** button becomes available:

- A valid Arm1 probe file name and tip angle.
- A valid Arm2 probe file name and tip angle.
- A valid tool specified in the list of available tools.

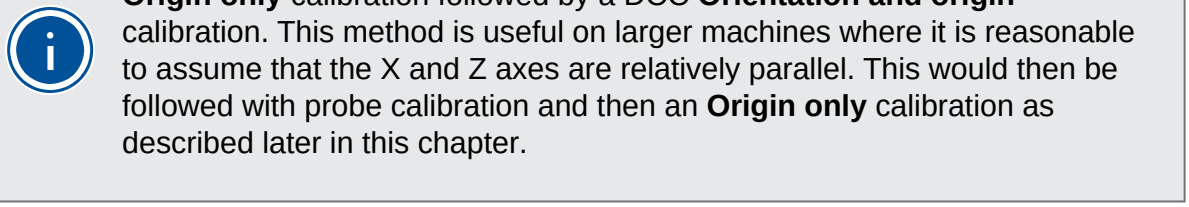
If you select **First arm holds tool and second arm measures**, you must provide the following items before the **Calibrate** button becomes available:

- A valid Arm1 probe file name and tip angle.
- A valid Arm2 probe file name and tip angle.
- The Arm1 tip type must be of type FIXEDBALL as specified in the probe.dat.

If you select **Second arm holds tool and first arm measures**, you must provide the following items before the **Calibrate** button becomes available:

- A valid Arm1 probe file name and tip angle.
- A valid Arm2 probe file name and tip angle.
- The Arm2 tip type must be of type FIXEDBALL as specified in the probe.dat.

10. Once you click this button, PC-DMIS begins the requested calibration. This creates an alignment between Arm1 and Arm2 by leveling, rotating, and setting the origin.
    - If you selected the **Manual calibration** option, you must begin by taking one point on the top of the tool. PC-DMIS automatically measures the rest of the points. Once you have measured the tool in the current position, PC-DMIS prompts you to move it to a new location on the table.
    - If you selected the **DCC calibration** option, PC-DMIS measures each of the given calibration sphere positions. Make sure the tool locations on the table are *not* co-linear (in a line). Position the tool locations as far as possible from each other, with at least one of the positions being raised up in the Z axis.



An alternate way to calibrate DCC machines involves performing a Manual **Origin only** calibration followed by a DCC **Orientation and origin** calibration. This method is useful on larger machines where it is reasonable to assume that the X and Z axes are relatively parallel. This would then be followed with probe calibration and then an **Origin only** calibration as described later in this chapter.

Part 2 of Calibration Procedure

Once you have completed the calibration described in Part 1, you need to calibrate the Arm1 and Arm2 probe files on the same calibration tool. This resets the origin relationship

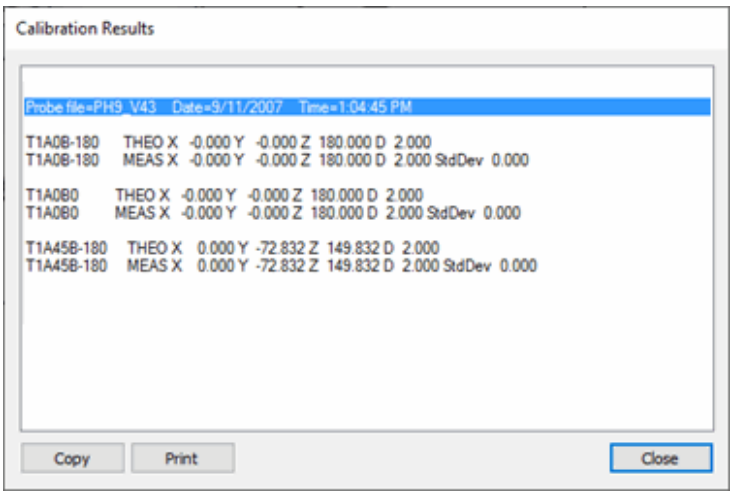
between the two tips. It doesn't change the level or the rotation of the alignment, only the origin. If you are using a PHS wrist, you need to do a wrist calibration with both arms on the common tool.

Follow this procedure:

1. Click the **Arm 1 Active** button on the **Active Arms** toolbar (**View | Toolbars | Active Arms**). Most users assign this button to the primary computer.
2. Access the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).
3. If PC-DMIS asks you if you want to load a new probe file, click **No**.
4. Calibrate the Arm1 probe (or do a wrist calibration if you are using a PHS). PC-DMIS prompts if the tool has been moved.
5. Click **Yes**, and follow the on-screen prompts.
6. When you complete the calibration, exit the **Probe Utilities** dialog box.
7. Click the **Arm 2 Active** button on the **Active Arms** toolbar. Most users assign this button to the Arm2 computer.
8. Access the **Probe Utilities** dialog box to calibrate the Arm2 probe (or do a wrist calibration if you are using a PHS).
9. This time when PC-DMIS prompts if the tool has been moved, click **No**.

Once you have calibrated both probe files in multiple arm mode, you are done with the calibration of the multiple arms. PC-DMIS copies the Arm2 file, the tool data, and the arm-to-arm transformation data to the Arm2 computer. This allows you to run Arm2 on its own as if it were an extension of the Arm1 coordinate system, or you can always run them together in multiple arm mode.

Viewing Calibration Results



If needed, you can access the calibration results by clicking on the **Results** button of the **Probe Utilities** dialog box. This displays the **Calibration Results** dialog box, which displays information about the calibrated tips of the related probe file. You can view the Arm2 calibration results in this same way on your Arm2 computer.

Performing an Automatic Calibration

In addition to the usual multiple arm calibration, you can perform an *automatic* calibration of the arms. PC-DMIS provides a command that automatically calibrates the current probe during measurement routine execution. PC-DMIS begins the calibration routine once it executes the command. To insert this command, select the **Insert | Calibrate | AutoCalibrate Multiple Arm** menu option.

The following command block is inserted into the Edit window:

```
AUTOCALIBRATE/MULTIPLEARM, ARM_THAT_MEASURES=BOTH,
MEASURE_MODE=DCC, OPERATION TYPE=ORIGIN,
QUALTOOL_ID=SPHERE3, MEASURE_AT_CENTER 1=<0,0,0>
ARM 1=CMM1, ARM 2=CMM2
```

A description of the items in this command block is given here:

ARM_THAT_MEASURES=
Indicates which arm performs the measurement (not which arm holds the tool). Choices are `BOTH` , `ARM2` or `ARM1` .

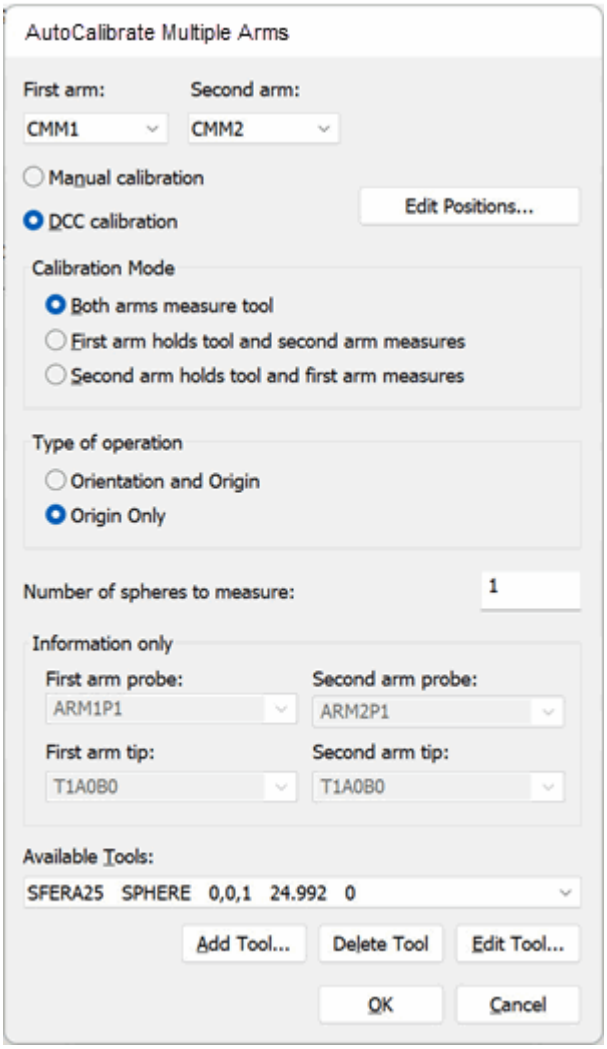
MEASURE_MODE=
Indicates whether the calibration is in `MANUAL` or `DCC` mode.

OPERATION_TYPE=
Indicates if calibration is for `ORIENTATION AND ORIGIN` or `ORIGIN` only.

QUALTOOL_ID=
Specifies the name of the qualification tool used.

MEASURE_AT_CENTER 1=
The X, Y, Z location to be used for measurement when using `CALIBRATION_MODE=DCC` . This is not present if the command is using `CALIBRATION_MODE=MANUAL` .

Press F9 to open the **AutoCalibrate Multiple Arms** dialog box.



AutoCalibrate Multiple Arms Dialog Box

Most of the items in this dialog box are the same as those used in the **Multiple Arm Calibration** dialog box (**Operation | Calibrate/Edit | Multiple Arm Mode**). The differences are listed below:

You cannot specify the probe and tip for Arm1 or Arm2. They are shown in the lists on this dialog box for information purposes only. The `AUTOCALIBRATE/MULTIPLEARM` command gets them from the content of the measurement routine in which the command is used.

First Arm Probe
For display only. This is determined by the `LOADPROBE` command for Arm1 that precedes the `AUTOCALIBRATE/MULTIPLEARM` command

First Arm Tip
For display only. This is determined by the `TIP` command for Arm1 that precedes the `AUTOCALIBRATE/MULTIPLEARM` command

Second Arm Probe
For display only. This is determined by the `LOADPROBE` command for Arm2 that precedes the `AUTOCALIBRATE/MULTIPLEARM` command

Second Arm Tip
For display only. This is determined by the `TIP` command for Arm2 that precedes the `AUTOCALIBRATE/MULTIPLEARM` command

Step 9: Calibrate the Multiple Arm Probe Files

Now that the system has been calibrated, you need to calibrate the multiple arm probe files.

For this calibration, you can:

- Calibrate any arm in any order.
- Use different calibration tools to calibrate the different arms.
- Calibrate Arm2 from the Arm2 computer and Arm1 from the Arm1 computer.
- Calibrate the Arm2 probe on the same calibration tool as the Arm1 probe.
- Only calibrate one arm at a time.

After this calibration, PC-DMIS synchronizes the probe files from the multiple computers the next time you enter multiple arm mode on the Arm1 computer.

Arm1 Probe File Calibration

If you want to calibrate the Arm1 probe file:

1. Select the **Arm 1 Active** icon from the **Active Arms** toolbar (**View | Toolbars | Active Arms**).
2. Access the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).
3. Follow existing procedures in the "Calibrating Probe Tips" topic of the "PC-DMIS CMM" documentation for information on how to calibrate probe files.

Arm2 Probe File Calibration

If you want to calibrate the Arm2 probe file:

1. Select the **Arm 2 Active** icon from the **Active Arms** toolbar (**View | Toolbars | Active Arms**).
2. Access the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**).
3. Follow the existing procedures in the "Calibrating Probe Tips" topic of the "PC-DMIS CMM" documentation for information on how to calibrate probe files.

If you want to use a calibration tool different from the one used for the Arm1 probe file, select the SPHERE(ARM 2) tool during Arm2 probe calibration.

To select the SPHERE(ARM 2) tool:

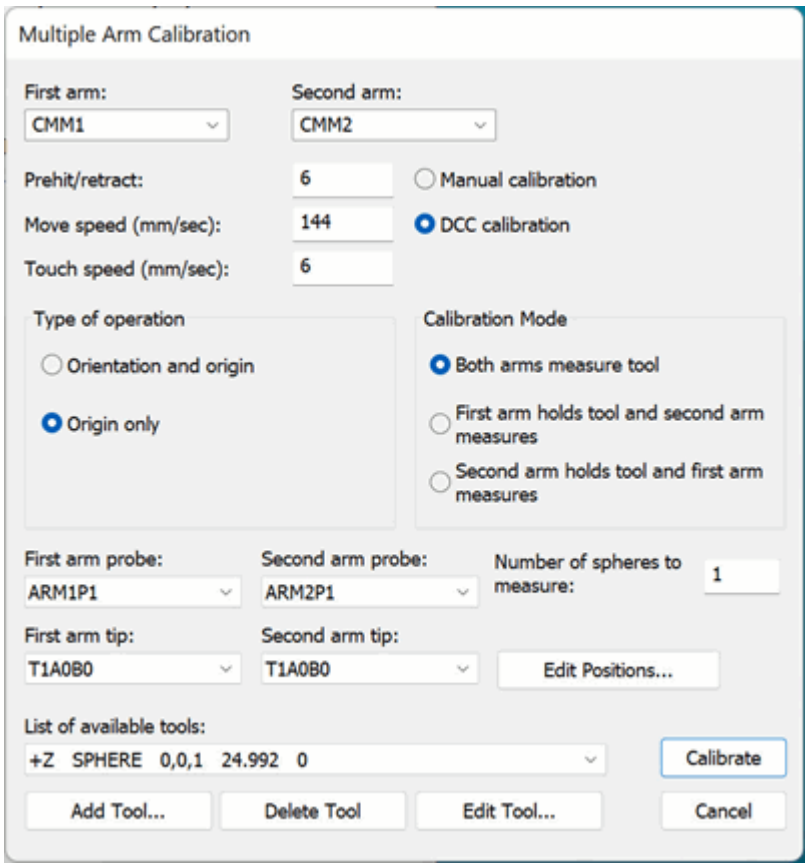
1. Select **Insert | Hardware Definition | Probe**.
2. Click **Measure**. The **Measure Probe** dialog box appears.
3. Click **Edit Tool**. The **Edit Tool** dialog box appears.
4. Select **SPHERE(ARM 2)** from the **Tool Type** list.
5. Click **OK** to confirm your choice. The SPHERE(ARM 2) information appears in the bottom of the **Measure Probe** dialog box.

Step 10: Set the Multiple Arm Origin

The final step in calibrating the multiple arm system is to set the origin between the two arms. You need to do this with calibrated probe files on both Arm1 and Arm2.

To set the multiple arm origin:

1. Open the **Multiple Arm Calibration** dialog box (**Operation | Calibrate/Edit | Multiple Arm Mode**).



Multiple Arm Calibration Dialog Box

2. Select the **Origin only** option.
3. In the **Number of spheres to measure** box, type the number of spheres you want to measure to establish the origin. If more than one sphere position is measured, PC-DMIS averages the positions to establish the origin.
4. Select the **Both arms measure tool** option.
5. Select the correct probe files and tips.
6. Select either the **DCC calibration** or **Manual calibration** option. If you select **DCC calibration**, make sure you define the correct sphere position using the **Edit Calibration Positions** dialog box.
7. From the **List of available tools**, select a tool that defines the correct diameter and orientation of the calibration tool to be measured.
8. Click the **OK** button.
    - If you selected the **Manual calibration** option, PC-DMIS prompts you to measure a single point on the sphere with the multiple arms. The rest of the points are then measured in DCC Mode around the sphere.
    - If you selected the **DCC calibration** option, PC-DMIS drives the arms to the sphere positions that are defined in the **Edit Calibration Positions** dialog box.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Once you have calibrated all the probe files in multiple arm mode and set the arm-to-arm origin, the calibration process is complete.

After you finish your multiple arm calibration and perform a wrist calibration (see "Wrist Calibration" in the "Using a Wrist Device" chapter), you need to repeat the Origin Only wrist operation as discussed early on in "Step 8: Calibrate the Multiple Arm System".

PC-DMIS copies the Arm2 probe file, tool data, and arm-to-arm transformation data to the Arm2 computer. Arm2 is then capable of running on its own as if it were an extension of the Arm1 coordinate system. You also have the option of running them together in multiple arm mode. Each time you enter multiple arm mode (by selecting **Operation | Enter Multiple Arm Mode**), PC-DMIS synchronizes the following between the two computers:

- Probe file changes
- Wrist calibration changes
- Error map data changes
- Probe changer data
- Calibration tool data changes

Creating a Measurement Routine Using Multiple Arm Mode

Once you have defined and calibrated the probes, you can create the measurement routine. You create a multiple arm measurement routine like any other measurement routine with some differences. Mainly, in a multiple arm measurement routine, you need to assign specific arms to execute different commands and define exclusion zones so that the arms don't collide. The topics below describe how to do this:

Assigning a Command to an Arm

By default, PC-DMIS assigns new commands to the current active arm. You can use the **Active Arms** toolbar (**View | Toolbars | Active Arms**) to switch the current active arm or to execute only those commands tied to a specific arm.

Active Arms Toolbar

The **Active Arms** toolbar contains multiple arm icons with corresponding color coded check mark icons. Each arm icon corresponds to an arm on the machine.

	Arm1
	Arm2
	Execute on Arm1
	Execute on Arm2
	Move to Arm1 Start Point
	Move to Arm2 Start Point
	Execute From Start Point

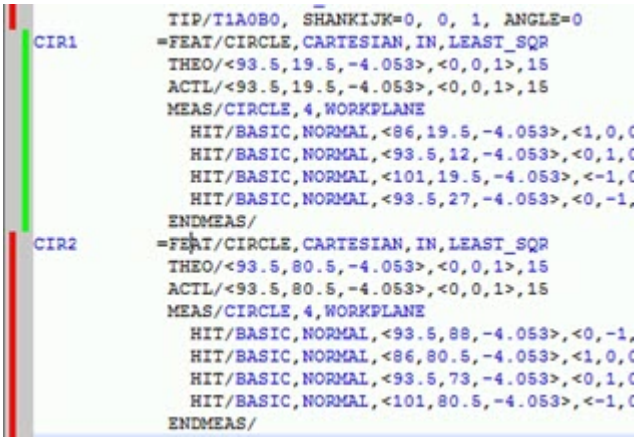
The *Arm(n)* icons (those with numbers) switches the current active arm.

The *Execute on Arm(n)* icons (those with check marks) executes only those commands associated with a specific arm.

The *Move to Arm(n) Start Point* icons move the cursor in the routine to the top of that arm's starting position.

The *Execute From Start Point* icon executes the routine for both arms at the currently defined start point.

Once Multiple Arm Mode is initiated, PC-DMIS inserts colored *vertical* lines into the left margin of the Edit window's Command Mode. This is used to identify the commands that are used for each arm. (In Summary Mode, PC-DMIS indicates commands assigned to the Arm2 with bold text.)



Edit window with green (feature CIR1) and red (feature LIN1) lines designating Arm2 and Arm1 portions respectively

- Features assigned to Arm1 are indicated using the red line.
- Features assigned to Arm2 are indicated using the green line.
- Features assigned to multiple arms are indicated using multiple colored lines.

Colored *horizontal* bars (instead of vertical) signify that the commands affect both arms and neither arm is allowed to execute this command until both arms have executed all prior commands. This type of command (usually a branching or alignment command) is executed by both arms at the same time.

Assigning Existing Commands to a New Arm

The **Toggle Multiple Arm Markings** menu item is only available on systems that have entered into multiple arm mode.

If you have commands assigned to a particular arm and you want to assign them to a different arm, do the following.

1. Place the Edit window into Command Mode.
2. Select the Edit window command you want to add.
3. Select **Operation | Toggle Multiple Arm Markings**.

When you select this option, PC-DMIS connects all of the highlighted commands to the other arm.

- If you don't highlight any commands, PC-DMIS connects the command where the cursor resides in the Edit window.
- You can assign most commands to be executed by Arm1, Arm2, or both arms. For example, you could create a `PREHIT` or `ALIGNMENT` command that applies to both Arm1 and Arm2, or to just one of the arms.

- You cannot assign certain commands to multiple arms. These include *features*, *hits*, *dimensions*, and *probe commands*.

Multiple Arm Measurement Routine Execution

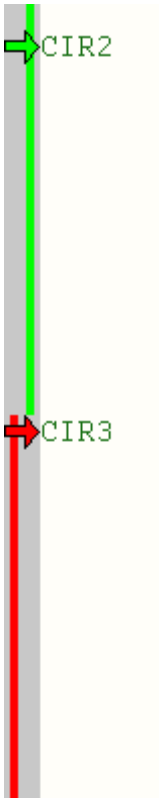
Unless you choose to execute just the commands assigned to a particular arm, when you execute the measurement routine, the routine flow continues as it normally does from the top of the Edit window to the bottom. Each arm executes the commands assigned to it.

When you execute in multiple arm mode, Arm2 runs a little behind Arm1. This type of delay is expected.

Setting Start Points for Multiple Arms

You can assign start points for the current arm by selecting the **Set Start Point** icon ( ) on the **Edit Window** toolbar. This is also accomplished by right-clicking in the Edit window in Command Mode and selecting it from the shortcut menu.

To set multiple start points, change the current learn arm before you click the **Set Start Point** toolbar icon.



A special start point arrow will appear in the left margin of the Edit window in a color that corresponds to the active arm color on the **Active Arms** toolbar (**View | Toolbars | Active Arms**).

The screen capture to the left shows that Arm1, in red, will begin execution at CIR3 while Arm2, in green, will begin execution at CIR2.

If you cancel execution, PC-DMIS automatically moves the start points for each arm to the command where the execution was cancelled for each arm.

Start points tell PC-DMIS to begin the routine's execution at that point when you select the **File | Partial Execution | Execute From Start Points** menu option. For information on using start points, see the "Setting Start Points" topic in the "Editing a Measurement Routine" chapter.

Be aware that if the current tip for that location in the routine does not match the current orientation of the probe head, PC-DMIS does not go back to execute the tip command above it in order to change the tip orientation.

Causing an Arm to Wait to Prevent Collision

Sometimes you may want one arm to wait until the other arm has finished measuring in an area that is overlapping. This is usually to prevent a collision between the arms. There are two different commands to prevent this.

You cannot use the `MOVE/SYNC` command within a Move Exclusion zone.

Using a Move Sync Command

You can put `MOVE/SYNC` commands at the beginning and end of a measurement sequence where you want to make sure that only one arm is moving. See "Inserting a Move Sync Command" in the "Inserting Move Commands" chapter for additional information.

Using a Move Exclusive Command

You cannot use the `MOVE/SYNC` command within a Move Exclusion zone.

You can use the `MOVE/EXCLUSIVE_ZONE` command.

- The advantage of using this method is that PC-DMIS only makes one of the arms wait if the other arm is in its zone.
- The disadvantage is that you need to put `MOVE/EXCLUSIVE_ZONE` commands around all the blocks of commands that command an arm into the overlapped area in the middle of the dual arm volume.

To use this command:

1. Find the sequence of commands that causes one of the arms to enter into an overlapped portion of the CMM volume.
2. Place a `MOVE/EXCLUSIVE_ZONE=ON` command at the start of the sequence.
3. Place a `MOVE/EXCLUSIVE_ZONE=OFF` at the end of the sequence.

Use the `MOVE/EXCLUSIVE_ZONE=ON` command to specify two corner points that form a 3D zone. PC-DMIS reserves this zone for the arm that is assigned the command. If the other arm is already in the requested zone, then PC-DMIS waits until the first arm is out of the way and exits the contested space with a `MOVE/EXCLUSIVE_ZONE=OFF` command. See the "Inserting a Move Exclusion Zone Command" topic in the "Inserting Move Commands" chapter in the PC-DMIS documentation for additional information.

Using Temperature Compensation with Multiple Arm Calibration

A thermocouple is a thermoelectric couple that measures temperature differences.

If you're compensating for temperature on the CMMs, you need to insert two temperature compensation commands into the routine - one command for Arm1 and another command for Arm2. Also, only the thermocouple for the part which is attached to the Arm1 controller is used for recording the part temperature.

In addition to having the STP file (Serv1.stp) in the appropriate directory on the Arm2 computer, you must also ensure that another STP file for Arm2 (named Serv1s.stp) resides in the corresponding directory on the Arm1 computer. To do this, copy Serv1.stp from the Arm2 computer, rename it to Serv1s.stp and place it on the Arm1 computer.

Synchronizing Arms for TEMPCOMP Commands

Once you have the TEMPCOMP commands inserted, you need to ensure that the arms are synchronized. To do this, insert two MOVE/SYNC commands, one before and one after the TEMPCOMP command assigned to Arm2, as shown here:

```
MOVESPEED/ 100
FLY/ON,3
TEMPCOMP/ORIGIN=0,0,0,Material Coeff=0.0000115,Reference Temp=20
  ,Hi Threshold=40,Lo Threshold=10,Sensor num=3
  ,X Axis Temp=21.141,Y Axis Temp=22.7843,Z Axis Temp=23.3941,Part Temp=21.6783
LOADPROBE/TEST_MASTER
MOVE/SYNC
TEMPCOMP/ORIGIN=0,0,0,Material Coeff=0.0000115,Reference Temp=20
  ,Hi Threshold=40,Lo Threshold=10,Sensor num=3
  ,X Axis Temp=,Y Axis Temp=,Z Axis Temp=,Part Temp=21.3603
MOVE/SYNC
LOADPROBE/TEST_MASTER
TIP/T1A0B0, SHANKIJK=0, -1, 0, ANGLE=0
LOADPROBE/TEST_SLAVE
TIP/T1A0B0, SHANKIJK=0, 1, 0, ANGLE=180
```

The highlighted items in this graphic show the two MOVE/SYNC commands around the TEMPCOMP command for Arm2

- The first MOVE/SYNC command provides the correct sequence for writing temperature values to the report. This ensures that temperature values are displayed for Arm1 first.
- The second MOVE/SYNC command prevents Arm2 from measuring until Arm1 finishes obtaining the part's temperature.

For more information on MOVE/SYNC, see "Inserting a Move Sync Command" in the "Inserting Move Commands" chapter.

For more information on temperature compensation, see the "Compensating for Temperature" topic in the "Setting Your Preferences" chapter.

Running the Arm1 Measurement Routine on Arm2

If you need to run your Arm1 measurement routine on Arm2, you can run PC-DMIS in Reverse Axes mode. This mode internally reverses the sign of the X and Y axes, making the Y axis positive towards the center of the machine and the X axis opposite the Arm1 X axis.

All existing probe files, wrist maps, tool changers, and other calibration and error comp files are usable in this mode without any change.

Calibrations done in either normal or Reverse Axes mode are correct and usable in either mode.

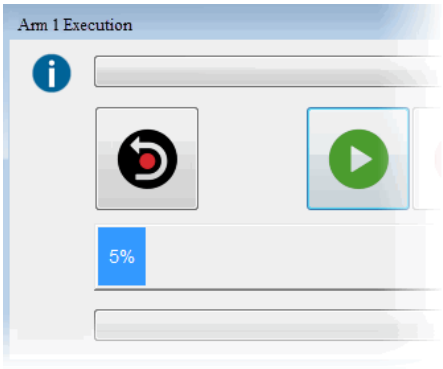
To add an icon that runs PC-DMIS in Reverse Axes mode

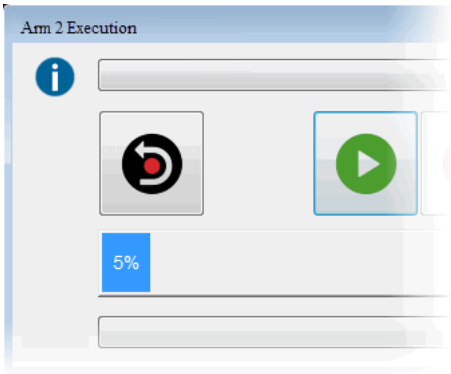
1. Use Windows Explorer and navigate to the directory where you want to add the icon.
2. From Windows Explorer's **File** menu, select **New** | **Shortcut**. A **Create Shortcut** wizard appears and prompts you to enter the path to the routine.
3. In the **Command Line** box, either type the complete path to the PC-DMIS executable file, or use the **Browse** button to navigate to and select the file. The default path is "C:\Pcdmisw\Pcdlrn.exe".
4. Once you have the path in the **Command Line** box, place your cursor at the end of the path, type a space, and then type a **/r** or **-r** switch. This switch notifies PC-DMIS to run in Reverse Axes mode. You can also combine this with Operator mode by adding a **–o** or **/o** switches to the command line.
5. Click **Next**.

6. In the **Select a Name for the Shortcut** box, type something like "PC-DMIS Reverse Axes Mode".
7. Click **Finish**. The new icon appears.

Dialog and Message Boxes in Multiple Arm Mode

Once you enable Multiple Arm Mode for your measurement routine, any dialog boxes or message boxes that are related to a specific arm contain an "Arm 1" or "Arm 2" identifier in their captions, like these **Execution** dialog boxes:





Dialog and message boxes that are affected include:

- **Execution** dialog box (**File | Execute**)
- **Probe Changer** dialog box (**Edit | Preferences | Probe Changer**)
- Info messages
- Warning messages
- Error messages

Dual Arms with Wrists Calibration Example

This advanced topic assumes you have a solid working understanding of PC-DMIS, including such concepts as probe angles, calibrations, the autocalibrate command, move point commands, multiple arm mode, and other concepts.

This topic provides a typical calibration example to show the steps needed to calibrate a dual-arm horizontal machine with different extensions on the wrist and with a probe changer. After a full calibration, additional steps explain how to create two measurement routines that will automate the calibration steps so you can more quickly perform the calibration when needed. Some steps are described generally and assume that as an advanced user, you are already familiar with the underlying principles needed.

This example uses CW43L_Multiwire wrists with short, medium, and long probe extensions containing TP2/TP6 probes. Your actual procedure will differ.

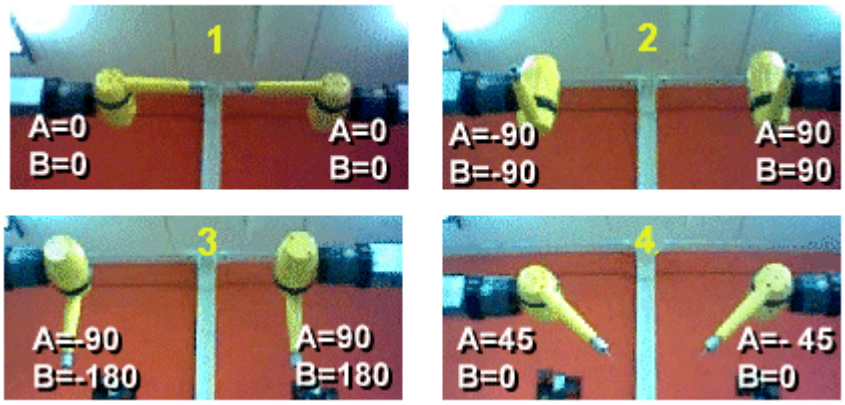
Before you begin, make sure the following has taken place:

- The machine has been properly compensated. This compensation data must reside in the Leitz controller for both the arms.
- You understand the importance of temperature compensation.

  Be sure to take into account how temperature will affect your calibration, and compensate as needed.

  The coefficient of expansion of the part (or in this case the calibration sphere) is zero. For more information, see "Compensating for Temperature" in the "Setting Your Preferences" chapter.

- Both arms have been properly assembled and aligned.
- A technician should have corrected any possible skew that may have occurred during the assembly and mechanical levelling of the wrists. Data from this correction should have been stored in the controller in the COSDAT.
- You should understand the wrist orientations on each arm. Each wrist when positioned at specific angles determines the orientation of that wrist's head compared with the machine's axes. The following example demonstrates some angles used on a PRIMA machine:



Example showing angular positions and the corresponding orientation values for each arm.

Long extensions are often useful on vertical machines and wrists. If you use a very long extension, you'll need to create a map for that extension. These machine types are highly accurate, and it takes several hours to finish a mapping on this type of machine.

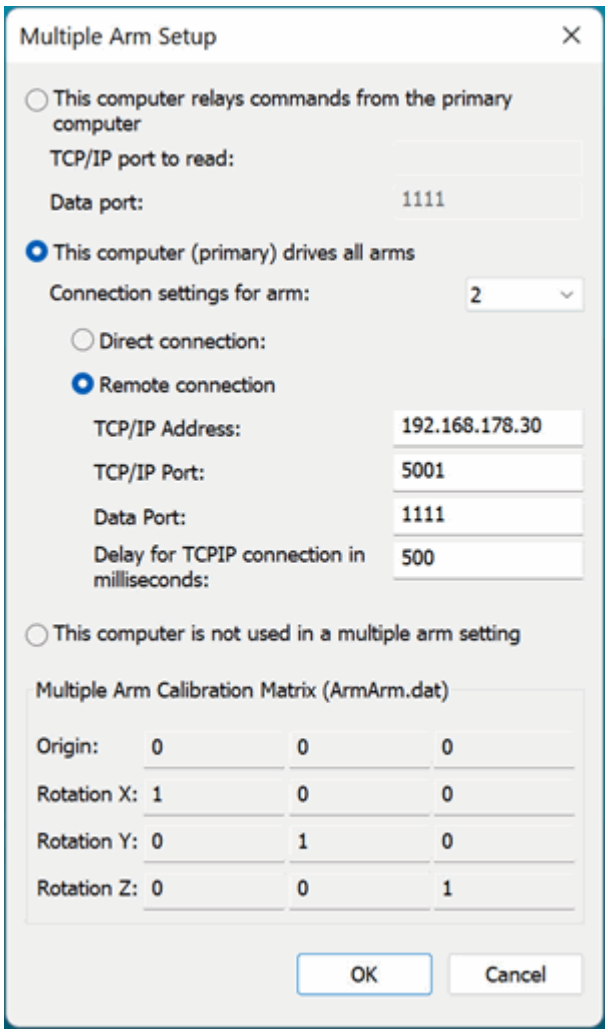
Step 1: Define Connection Info and Configure Angles

Since the two arms use different orientations, you need to properly configure the setup parameters to reflect this.

Set Up the Primary Computer

On your primary computer, modify the **Multiple Arm Setup** dialog box with these options:

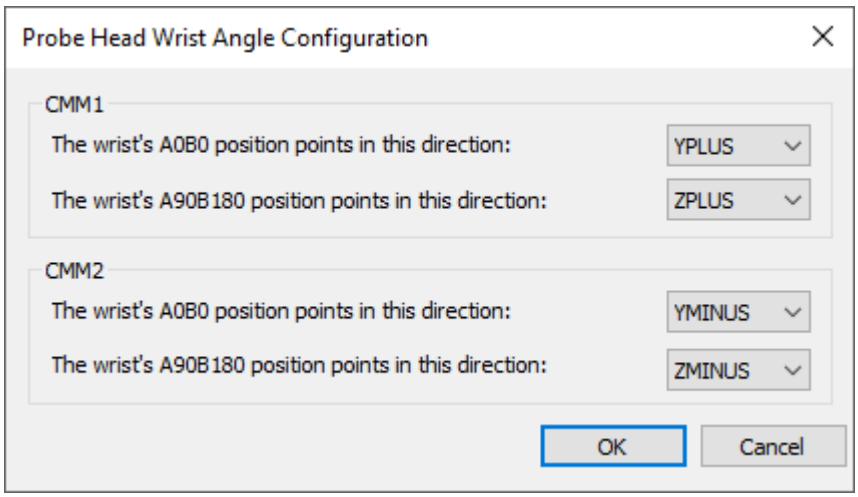
1. To access this dialog box, select **Edit | Preferences | Multiple Arm Setup**.
2. Mark the **This computer (primary) drives all arms** option.
3. Choose **2** from the **Connection settings for arm** list.
4. Choose **Remote connection**. Set the connection values to the secondary machine: **TCP/IP Port**, **Data Port**, and **Delay for TCPIP connection in milliseconds**.
5. Click **OK** to close the **Multiple Arm Setup** dialog box.



Multiple Arm Setup Dialog Box

Also, modify the **Probe Head Wrist Angle Configuration** dialog box with these options:

1. To access this dialog box, select **Edit | Preferences | Setup**, the **Part/Machine** tab, and then the **Probe Head Orientation** button.
2. In the **CMM1** area, set the wrist's A0B0 position to point in YPLUS direction. Set the A90B180 position to point in ZPLUS.
3. In the **CMM2** area, set the wrist's A0B0 position to point in the YMINUS direction. Set the A90B180 position to point in ZMINUS.
4. Click **OK** to close the **Probe Head Wrist Angle Configuration** dialog box.
5. Click **OK** to close the **Setup Options** dialog box.



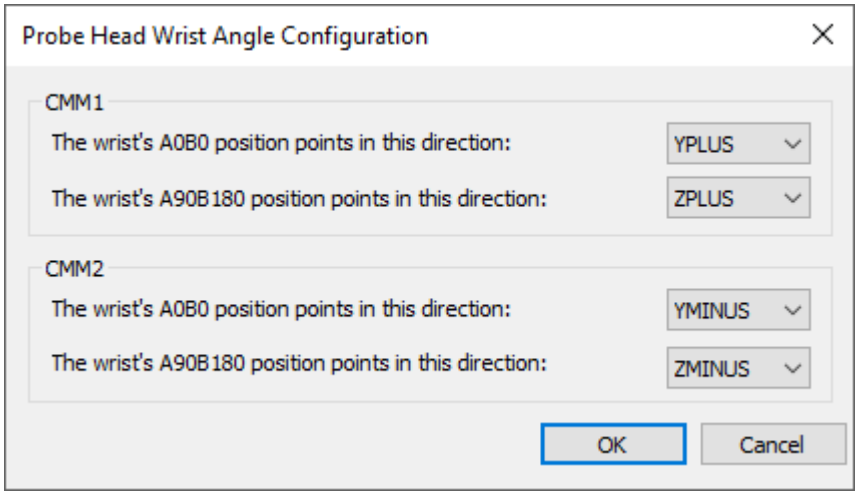
Probe Head Wrist Angle Configuration Dialog Box

Set Up the Secondary Computer

On the secondary computer, modify the **Multiple Arm Setup** dialog box, and click the **This computer relays commands from the primary computer** option.

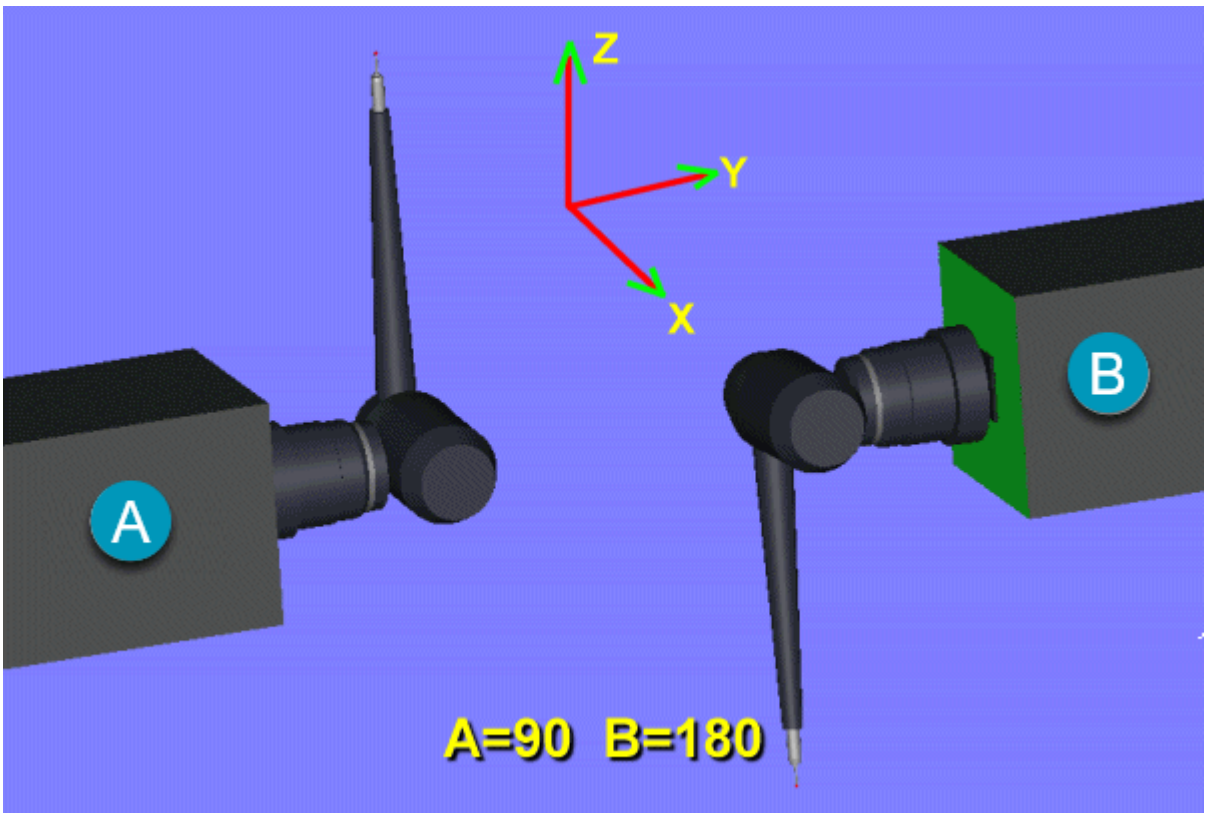
Modify the **Axis** tab of the **Machine Options** dialog box (**Edit | Preferences | Machine Interface Setup**) with these options:

- Set the **X** list to **-X**
- Set the **Y** list to **-Y**
- Set the **Z** list to **Z**



Probe Head Wrist Angle Configuration Dialog Box

The angles are now set up as shown here:



A. *Arm1*

B. *Arm2*

Angles with DEA Wrists

If you're using a machine with DEA wrists, use the PC-DMIS Settings Editor to modify these entries *on both computers* to the following values:

- `DEAWrist = 1`
- `RotateWristFromController = TRUE`
- `FlipBAxis = TRUE`
- `AaxisMax = 181`
- `AaxisMin = -181`

> The `AaxisMax` and `AaxisMin` entries define the allowed amount of angular rotation for the wrists. For very long extensions, you should not exceed +/- 124 degrees.

On the primary LEITZ computer (arm 1), set these entries:

- `AxisX = 0`
- `AxisY = 2`

- `AxisZ = 4`

On the secondary LEITZ computer (arm 2), set these entries:

- `AxisX = 1`
- `AxisY = 3`
- `AxisZ = 4`

Note on Different Configurations

If you need to configure your machine differently from a conventional DEA machine; for example, if you want to use the wrist convention used in the PHS Renishaw, you would need to set the following parameters. They are then stored in the JSON file:

1. In the **Machine Options** dialog box for arm 1 (computer 1), set the **B offset** value to -180.
2. In the **Probe Head Wrist Angle Configuration** dialog box, specify the proper axes associated with the wrist rotations.

The figure below shows the values you should change along with some sample wrist rotations at the noted angles:

The next step provides information on backing up your files.

Step 2: Backup and Delete Files

Before you continue with the calibration, be sure to back up the following items on both computers:

- The entry settings
- The *.PRB, and the *.Results files
- These data files: abcalib.dat, abcalib_CMM2.dat, abcomp.dat, abcomps_CMM2.dat, aboutput.dat, aboutput_CMM2.dat, wristm.dat, wrists_CMM2.dat, armarm.dat, tool.dat, toolc.dat

The PC-DMIS Settings Editor provides the ability to back up entry settings and user data files. See the "Working with Backup Files and User Data" topic in the PC-DMIS Settings Editor documentation for information on backing up important files.

You may have to back up some files manually. See the "Understanding File Locations" topic for information on where the above file types are stored in case you need to do this.

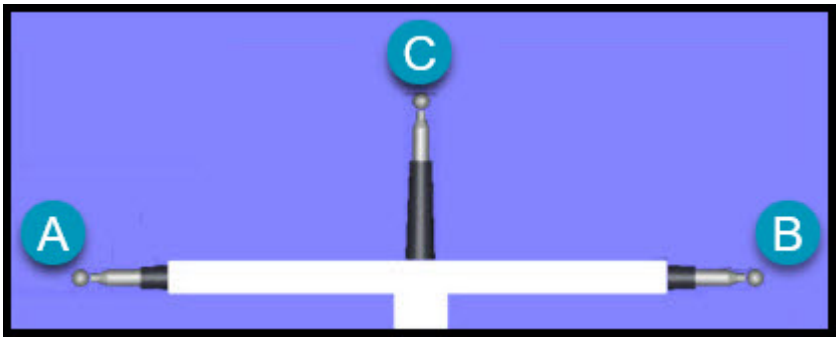
Now that you have backed up the above files, you should delete the original ones (keep the backed-up ones as needed). This ensures that you're starting with a fresh setup and that previous calibration data won't affect the current calibration.

The next step provides information on creating the basic arm-to-arm relationship.

Step 3: Create a Basic Arm-to-Arm Relationship

In this step, you'll create a measurement routine, define your probes, define your calibration spheres, and then perform an initial calibration to establish a basic relationship between the two arms. This calibration consists of machines containing wrists with probe extensions at a length of 332 mm. This will result in a calibration of medium accuracy. The probe changer will not be used at this point.

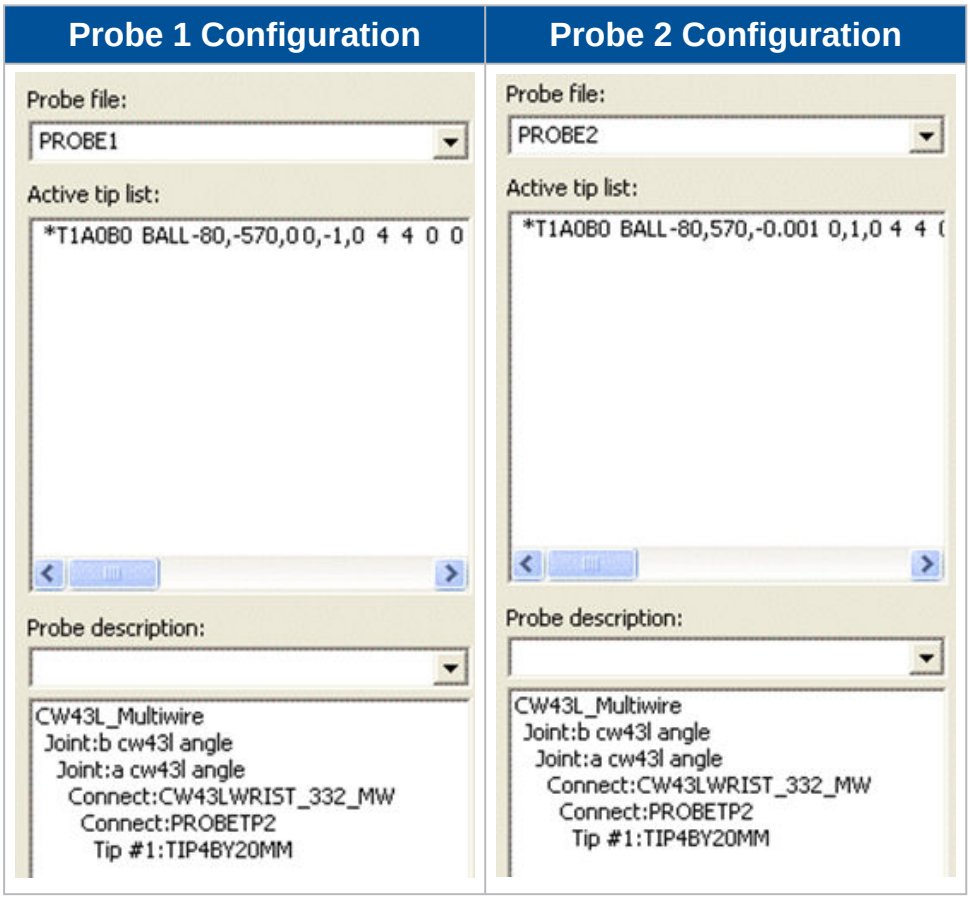
Three calibration spheres mounted on a cross-like fixture with 15 mm calibration spheres will be used for this calibration, like this:



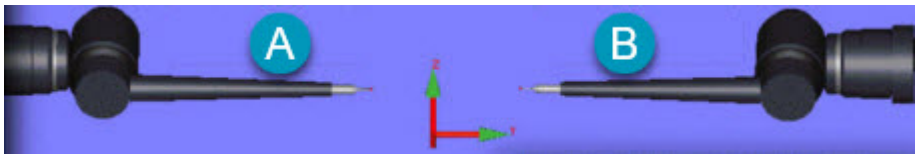
(A) - Sphere 1
(B) - Sphere 2
(C) - Sphere 3

Create a New Measurement Routine

Create a new measurement routine and then load or create the two probe files, PROBE1 and PROBE2, into your measurement routine.

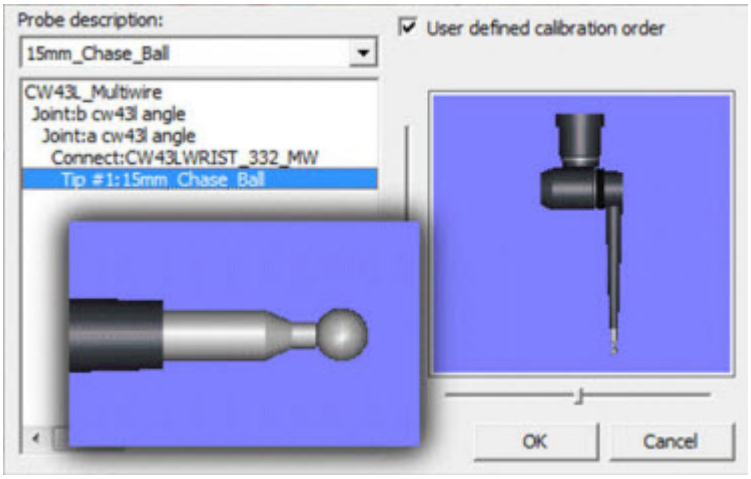


They should look like this:



(A) - Probe 1
(B) - Probe 2

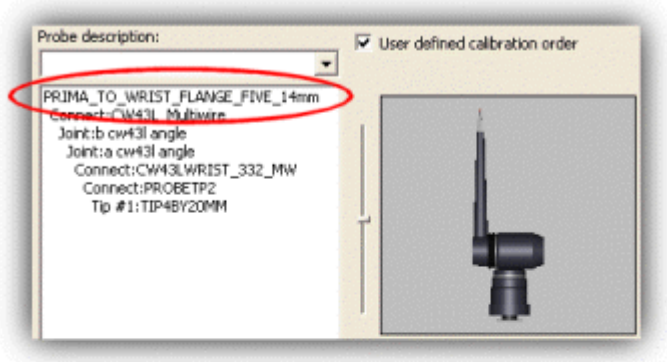
Load in a third probe file, name it PROBALL, and give it a 15 mm fixed rigid ball, like this:



Your measurement routine should look like this:

```
STARTUP    =ALIGNMENT/START,RECALL:,LIST=YES
            ALIGNMENT/END
            MODE/MANUAL
            PREHIT/6
            RETRACT/6
            CHECK/6,1
            MOVESPEED/ 80
            TOUCHSPEED/ 5
            SCANSPEED/80
            FORMAT/TEXT,OPTIONS, ,HEADINGS,SYMBOLS, ;NOM,TOL,MEAS,DEV,OUTTOL, ,
            LOADPROBE/PROBE1
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
            LOADPROBE/PROBE2
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
            LOADPROBE/PROBALL
            TIP/T1A0B0, SHANKIJK=0, 0, 1, ANGLE=0
                              END OF MEASUREMENT FOR
      PN=2010MR1_Test                 DWG=                SN=
   TOTAL # OF MEAS =0      # OUT OF TOL =0      # OF HOURS =00:00:00
```

If your physical structure of the probe and arm are not displayed in the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), be sure to edit the USRPROBE.DAT so that the dialog box displays the necessary hardware. For information on data files, see "Understanding Data Files" topic in the "Setting Your Preferences" chapter.



USRPROBE.DAT

```
Comment ------------------------------- FLANGE PRIMA FOR FIVE
ITEM:PRIMA_TO_WRIST_FLANGE_FIVE_14mm ARM
color 30 30 30
ribcount 10
solid 5
face 4 36 32.5 0 -36 32.5 0 -36 32.5 -4 36 32.5 -4
face 4 36 -32.5 0 -36 -32.5 0 -36 -32.5 -4 36 -32.5 -4
face 4 36 32.5 0 36 -32.5 0 36 -32.5 -4 36 32.5 -4
face 4 -36 32.5 0 -36 -32.5 0 -36 -32.5 -4 -36 32.5 -4
face 4 36 32.5 -4 -36 32.5 -4 -36 -32.5 -4 36 -32.5 -4
cylinder 0 0 -4 0 0 -14 41.5
connect 0 0 -14 0 0 1 ARM
Comment ------------------------------------------------------
```

Example usrprobe.dat file edited to contain a flange between the arm and wrist

Define the Calibration Spheres

1. Access the **Add Tool** dialog box:

- Select **Insert | Hardware Definition | Probe**.
- Click the **Measure** button.
- Click the **Add Tool** button.

2. In **Tool ID**, type SPHERE3.
3. In **Tool type**, choose SPHERE.
4. In the **Shank vector IJK** boxes, type 0,0,1.
5. In the **Diameter/Length** box, type 15.875.
6. Click **OK** to close the **Add Tool** dialog box.
7. Repeat steps 2 through 6 to define the SPHERE1 calibration sphere. Use 0,-1,0 for the vector.
8. Repeat steps 2 through 6 to define the SPHERE2 calibration sphere. Use 0,1,0 for the vector.
9. The information is stored and written to your Tools.dat file.
10. Click **Cancel** to close the **Measure Probe** dialog box.
11. Click **Cancel** to close the **Probe Utilities** dialog box.

Do a Preliminary Calibration

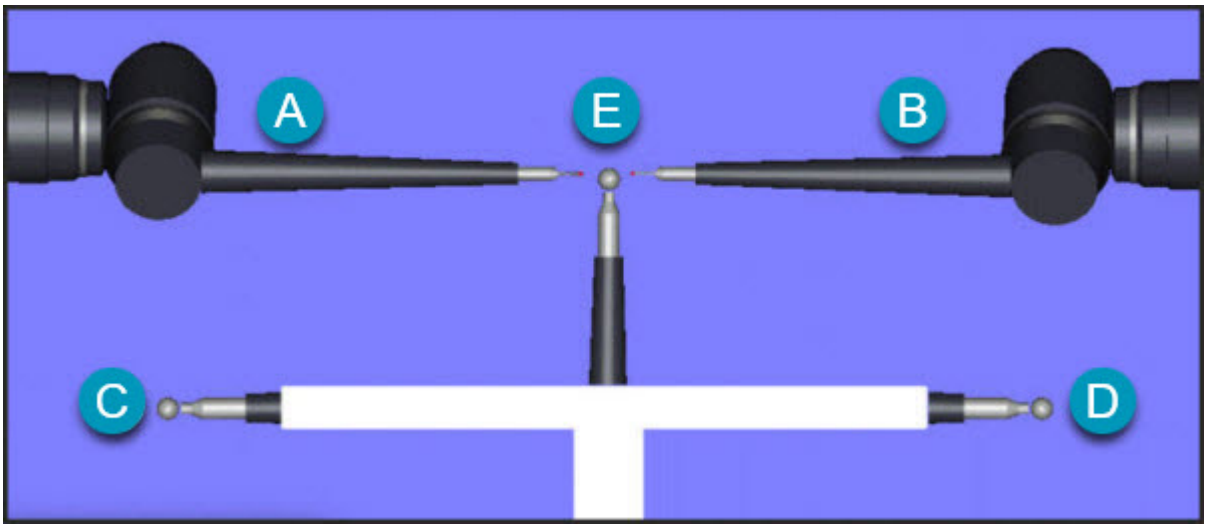
You need to perform a preliminary calibration on SPHERE3 in order to temporarily map the two arms. You can do this calibration without needing to calibrate the actual probe tips for PROBE1 and PROBE2.

1. Access the **Multiple Arm Calibration** dialog box by selecting **Operation | Calibrate/ Edit | Multiple Arm Mode**.
2. Set the **First arm** list to CMM1.
3. Set the **Second arm** list to CMM2.
4. Choose the **Both arms measure tool** option.
5. Type a value of 1 in the **Number of spheres to measure** box.
6. Choose the **Manual calibration** option.
7. Choose the **Origin only** option.
8. Set **First arm probe** to PROBE1.
9. Set **First arm tip** to T1A0B0.

10. Set **Second arm probe** to PROBE2.
11. Set **Second arm tip** to T1A0B0.
12. Choose **SPHERE3** from the list of available tools.
13. Click **Calibrate**. Follow any on-screen prompts.

When the calibration finishes, your Probe Readouts window for arm 1 and arm 2 should contain nearly identical X and Z values.

Before Calibration	After Calibration



Following a Preliminary Calibration on SPHERE3
(A) - PROBE 1
(B) - PROBE 2
(C) - Sphere 1
(D) - Sphere 2
(E) - Sphere 3

You've successfully calibrated SPHERE3 in order to temporarily map arms 1 and 2. The calibration information is stored within an ArmArm.dat file.

The next step provides information on initiating a more accurate calibration.

Step 4: Initiate a More Accurate Calibration

In this step, you will perform a more accurate calibration to define the alignment of the axes between arm 1 and arm 2.

1. Swap out the TP2 tip on the arm 1 extension with a 15 mm tip.

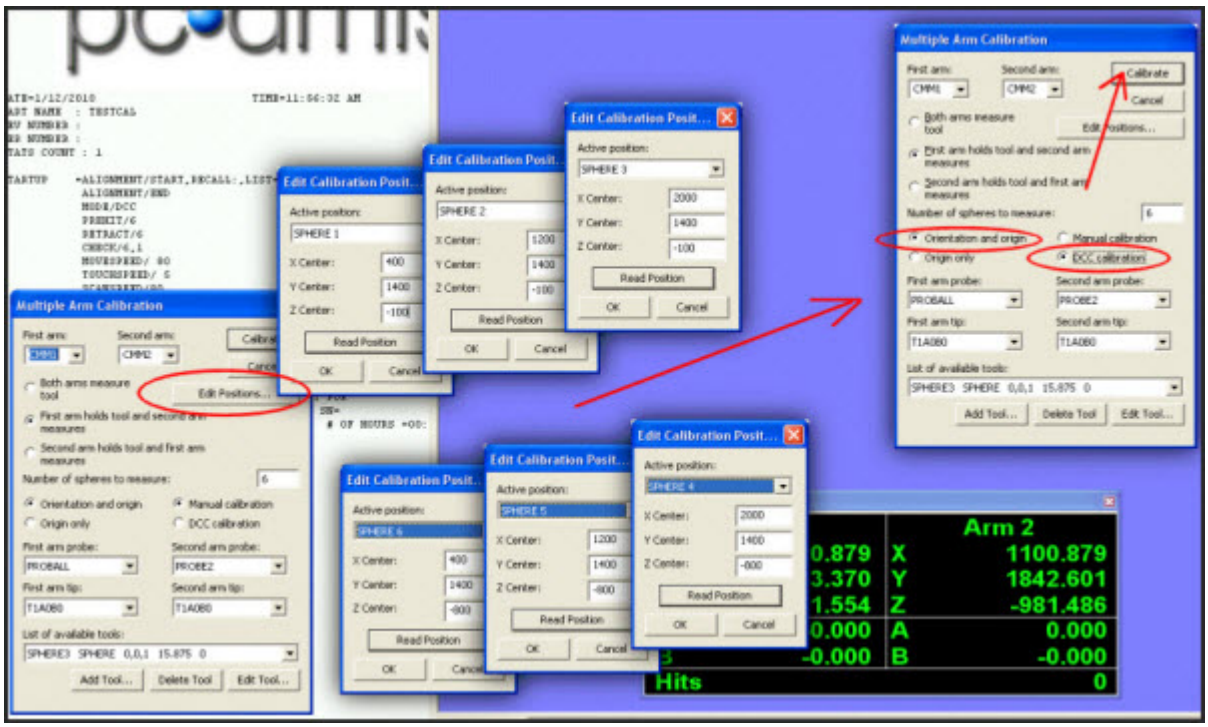
2. In the **Multiple Arm Calibration** dialog box (**Operation | Calibrate/Edit | Multiple Arm Mode**), make sure the following are set:

    - For **First arm probe**, select the PROBALL probe.
    - For **Second arm probe**, select the PROBE2 probe.
    - For **Number of spheres to measure**, define how many spheres PC-DMIS will measure with each arm. For this example we'll use a value of 6. But for your machine, you may want to choose a different number based on the dimensions of your machine. A value of 9 is good. 12 is the maximum. The higher the number, the greater the accuracy, but you'll also experience an increase in calibration time with more spheres during the DCC calibration.

3. Calculate the orientation plane so that the sphere of arm 1 will move on a plane lying on the Y center line of the machine. This will allow arm 2 to later reach the ball in all positions. To do this:

    - Manually move arm 1 to the six sphere positions specified in the **Multiple Arm Calibration** dialog box.
    - From the **Multiple Arm Calibration** dialog box, click **Edit Positions** and correct each sphere position as needed to improve the distribution. This may be needed since the positions are not perfectly aligned nor evenly distributed on the vertical plane.
    - You may find it helpful to read the position of the active arm to fill in the X, Y, Z coordinates. The **Read Position** button automatically uses the current X, Y, Z position of the arm.

4. Choose the **Orientation and Origin** option.
5. Choose the **DCC calibration** option.
6. Click **Calibrate**. The armarm.dat file is regenerated with a more accurate mapping from this calibration.



Different dialog boxes showing the settings used.

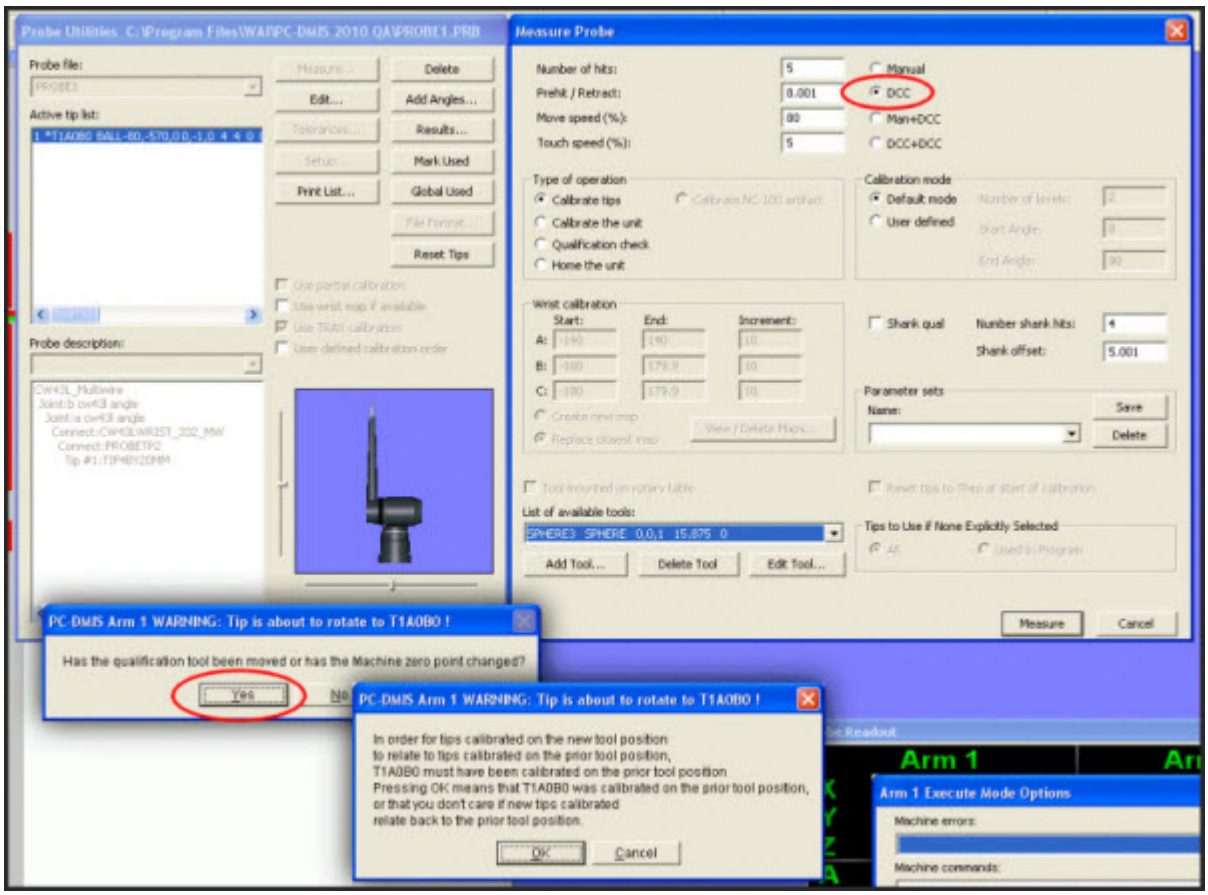
The next step provides information on performing the DCC calibrations.

Step 5: Performing a DCC Calibration

DCC Calibration of SPHERE3 Using Arm 1

Continue the calibration in DCC Mode for arm 1 measuring SPHERE3. This is the center sphere on the calibration fixture.

1. Swap out the 15 mm tip on the Arm 1 extension with the TP2 tip.
2. In the **Measure Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button) for the PROBEALL probe, choose **DCC**.
3. In the **List of available tools**, choose SPHERE3.
4. Click **Measure**.
5. When PC-DMIS asks if the qualification tool has moved or the machine zero point has changed, click **Yes**.



Different dialog boxes showing the settings used.

DCC Calibration of SPHERE1 Using Arm 1

1. In the **Measure Probe** dialog box, in the **List of available tools**, choose SPHERE1.
2. Ensure that the IJK orientation of this calibration sphere is correctly set to 0, -1, 0.
3. Click **Measure**.
4. When PC-DMIS asks if the qualification tool has moved or the machine zero point has changed, click **Yes**.

DCC Calibration of SPHERE2 using Arm 2

1. Access the **Measure Probe** dialog box for PROBE2, the probe on Arm 2.
2. In the **Measure Probe** dialog box, choose **DCC**.
3. In the **List of available tools**, choose SPHERE2.
4. Ensure that the IJK orientation of this calibration sphere is correctly set to 0, 1, 0.
5. Click **Measure**.
6. When PC-DMIS asks if the qualification tool has moved or the machine zero point has changed, click **Yes**.

The next step provides information on mapping the wrists of Arm 1 using SPHERE1.

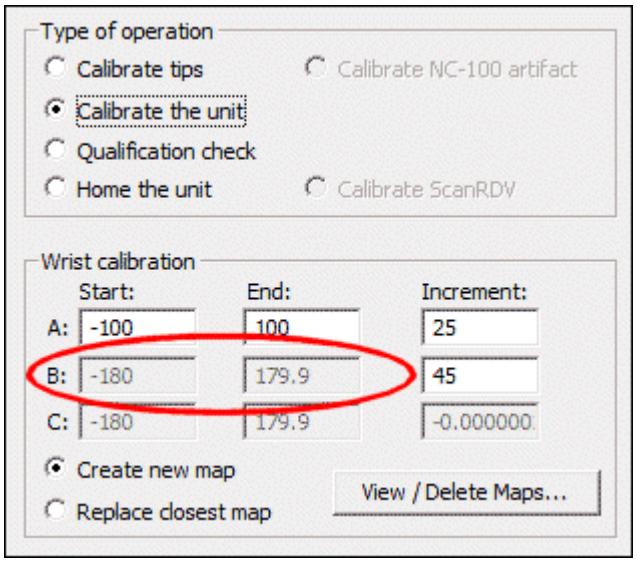
Step 6: Map the Wrists of Arm 1 using SPHERE1

At this point, you have a probes for both arms defined. You've also defined the orientation between the arms. Now, you need to map available wrist angles you'll be using.

1. Check to see the ability to use wrist maps is already enabled. You can do this by accessing the **Probe Utilities** dialog box for a probe that has a wrist. If you see the **Use wrist map if available** check box, this functionality is enabled. If you don't see it, check to ensure that the `DeaWrist` entry is set to 1.

2. Access the **Measure Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button) for PROBE1 and set these options and values:

    - In the **Type of operation** area, choose **Calibrate the unit**.
    - In **Calibration Mode**, choose **User defined**.
    - The boxes in the **Wrist calibration** area should now be editable.
    - Choose **DCC + DCC**. This is needed for long extensions.
    - For the A angle values (the probe's pitch), define the following: For **Start A**, type -90; for **End A**, type 90; and for **Increment**, type 30.
    - In the B angle values (the probe's roll), define the following: For **Start B**, type -180; for **End B**, type 180; and for **Increment**, type 45. Note that you cannot adjust the A and B angles. You can only adjust the increment.



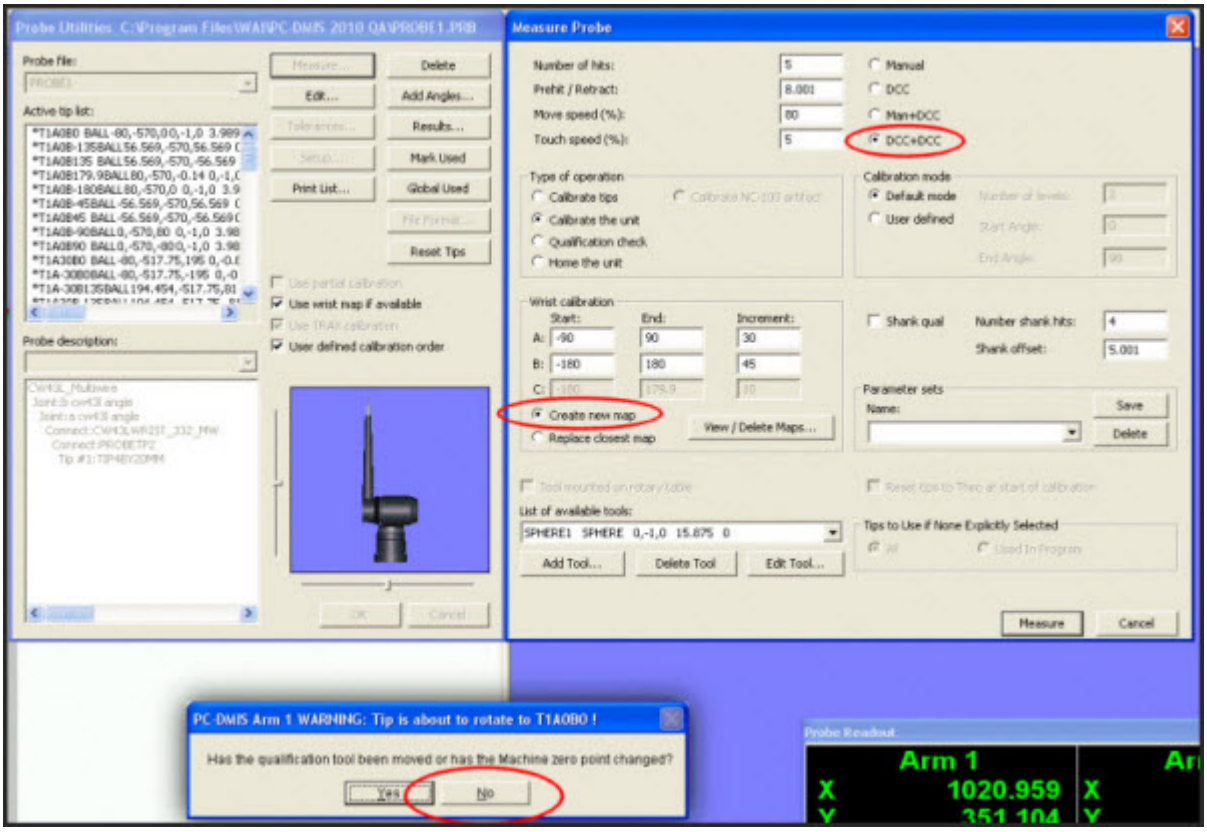
Wrist Calibration area with B angles disabled in version 2012 and later.

    - Choose **Create new map**.

> For increased precision in your calibration, you can decrease the increment value. This increases the time. For shorter extensions, the value of 30 and 45 is a medium value.

3. For **List of available tools**, choose SPHERE1.
4. Click **Measure**.
5. When PC-DMIS asks if the qualification tool has been moved or the machine zero point changed, click **No**.

Arm 1 starts to measure all the positions defined in the dialog box. Generally, this lasts approximately an hour, but it depends on the speed of the machine and the size of your probe's extension. A shorter extension is faster.



Different dialog boxes showing the settings used.

The next step provides information on mapping the wrists of Arm 2 using SPHERE2.

Step 7: Map the Wrists of Arm 2 Using SPHERE2

Repeat Step 6, but instead use Arm 2 with SPHERE 2. After both arms have calibrated the wrist angles, the following files are created or modified with the latest calibration data:

- *. PRB

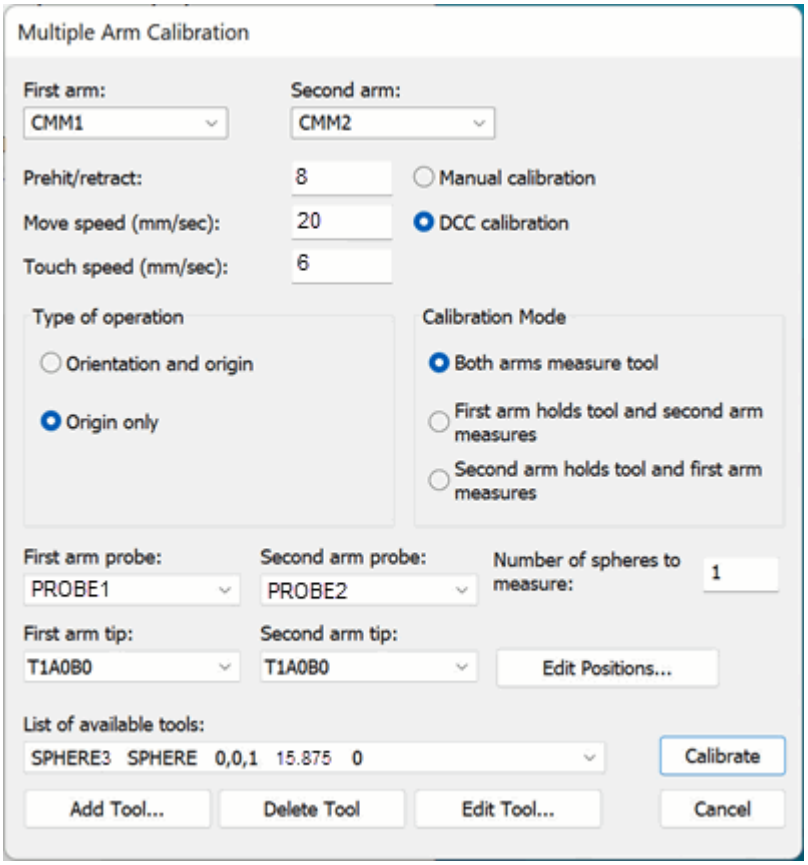
- *. Results
- abcalib.dat and abcalib_CMM2.dat
- abcomps.dat and abcomps_CMM2.dat
- aboutput.dat and aboutput_CMM2.dat
- wrists.dat and wrists_CMM2.dat
- toolc.dat and toolc_CMM2.dat
- armarm.dat
- tool.dat

The next step provides information on mapping the origin between arms.

Step 8: Map the Origin Between Arms

In the previous steps, you created a basic relationship between arm 1 and arm 2. They were mainly used to establish the orientation. The origin between arms was not established because the mapping didn't exist then. Now that the wrist mapping is done for both arms, you can now map the origin. This will generate two maps, one for each arm.

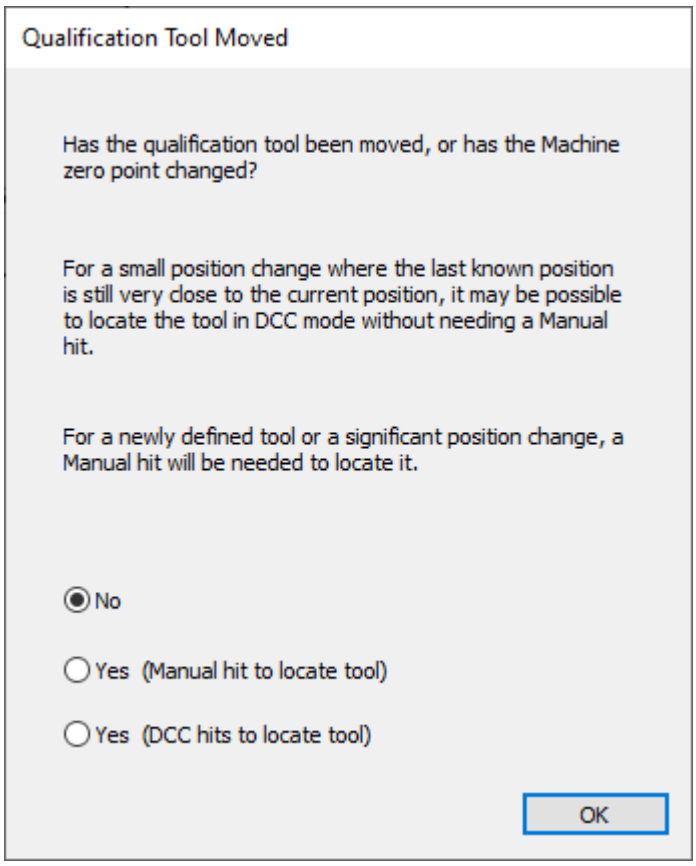
1. From the **Multiple Arm Calibration** dialog box (**Operation | Calibrate/Edit | Multiple Arm Mode**), set the following:
    - Choose **Both arms measure tool**.
    - Set **Number of spheres to measure** to 1.
    - Choose **Origin only**.
    - In **First arm probe**, choose PROBE1 and set the **First arm tip** to T1A0B0.
    - In **Second arm probe**, choose PROBE2 and set the **Second arm tip** to T1A0B0.
    - In **List of available tools**, choose SPHERE3.



Multiple Arm Calibration dialog box showing the settings used.

2. Click **Calibrate**.

3. When PC-DMIS asks if the qualification tool has been moved or the machine zero point changed, click **No**. Since the calibration spheres are almost always located in the same position of the machine (it's screwed in place), you can choose **No**. PC-DMIS doesn't care if the spheres are off by some tenth of a millimeter.



The next step provides information on performing a test measurement.

Step 9: Perform a Test Measurement

At this point, do a test measurement to ensure that everything is set up correctly and to see if the accuracy is good.

1. Use a test part, something like a 700 mm sized test block.
2. Tilt the part in 3D space.
3. Measure the part using an alignment.
4. Reposition it in different locations inside the machine volume.
5. Measure with a single arm.
6. Measure with both arms.
7. Compare the results against measurements done on separate, single-arm machines.

The next step provides information on mapping the other extensions.

Step 10: Map the Other Extensions

Some machines come with three extensions: short, medium, and long. For your first calibration, it's best to use a short extension. If you have other extensions to use on each arm, you would need to perform a mapping for each additional extension. Since you have already created a mapping using the shortest extensions, you need to now do mappings of the other two extensions (long and medium extensions for both arms).

This step is needed before we can calibrate and use probe changers that will hold the different extensions.

Your probes should be named or renamed using a format that corresponds to the different extension lengths and the different arms.

For example, this procedure uses the following:

- ARM1_1 is the short extension on arm 1
- ARM1_2 is the medium extension on arm 1
- ARM1_3 is the long extension on arm 1
- ARM2_1 is the probe with the short extension on arm 2
- ARM2_2 is the medium extension on arm 2
- ARM2_3 is the long extension on arm 2

PC-DMIS supports multiple maps. You can potentially even have a separate map for each probe. But having so many maps is very impractical, given the work and attention needed for a single mapping procedure. So, when possible, try to group similar probe lengths together so that they use a single map. However, you should create a new map if the range between probe extensions is very large.

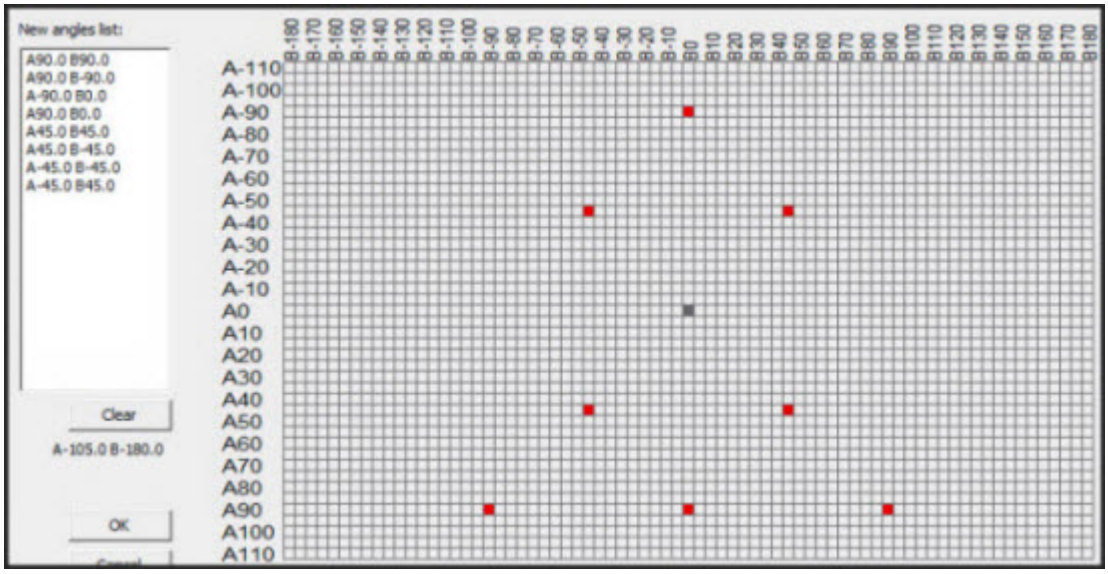
Map the Long Extensions

Two new maps are needed for the long extension. To do this, create a new map by repeating Step 6, Step 7, and Step 8; this time however, use the probes with the long probe extensions.

Map the Medium Extensions

For the medium extensions, you can update the extensions to an existing map.

1. Access the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**) for the probes with the medium extensions, and using the **Add New Angle** dialog box add at least 9 angular positions. This defines the calculation of the new offset, due to variation in the extension. These are some good recommended angles to use: [0,0] [90,90] [90,-90] [-90,-90] [-90,90] [45,45] [45,-45] [-45,-45] [-45,45]



Add New Angle dialog box showing the suggested angles.

2. Access the **Measure Probe** dialog box.
3. Choose **Calibrate tips**. Since you are updating an existing map, not creating a new one, you'll see that **Calibrate the unit** and **Create new map** are both grayed out.
4. Click **Calibrate**.

The next step provides information on calibrating the probe changers.

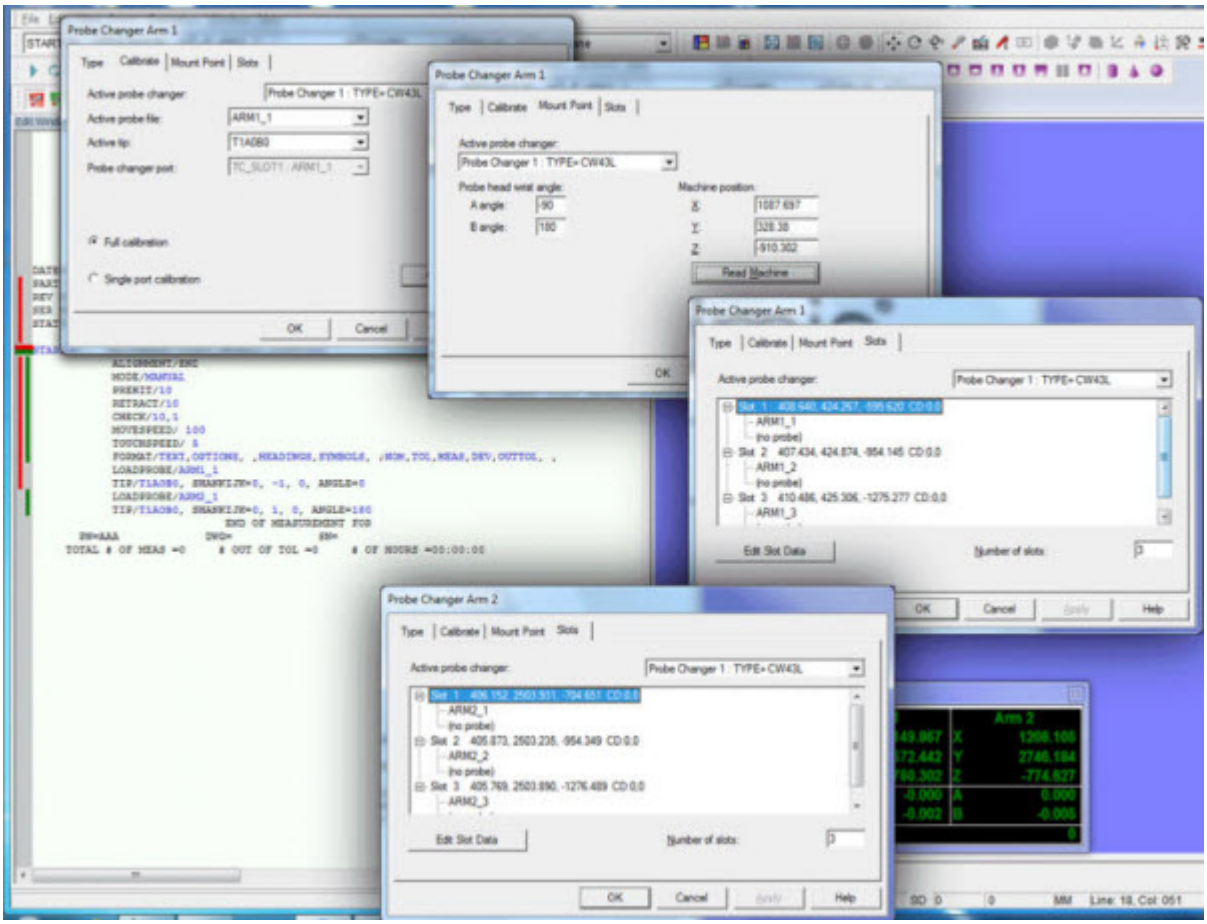
Step 11: Calibrate the Probe Changers

> **!** Before you calibrate the probe changer, make sure that all tips have been calibrated and the relationships between the arms is also established.

Since you'll have multiple extensions (three for each arm is the minimum), you'll need to do a probe changer calibration to support the loading and unloading of probe extensions with the probe changer. It is assumed that you already know how to do the probe changer calibration.

Follow your existing procedure to calibrate the probe changers. If you want to see how some probe changers are calibrated, see the "Defining Probe Changers" topic here.

If you're planning to use a laser probe on your probe changer, you can devote a slot on the probe changer rack to that particular component. To do this, use the `CW43LThirdAxisTCSlot` entry in the PC-DMIS Settings Editor.



The next step provides information on setting up routine files for automatic use.

Step 12: Setup Routine Files for Automatic Use

With the previous steps completed, the dual arm setup is now ready to use. The tips are calibrated and related for both arms as well as with the probe changers.

However, there are times when you'll need to adjust the calibration or correct the offset (for example if you need to replace a damaged tip or if you want to substitute a different tip). Repeating the previous manual steps would be very time consuming. For this reason, it makes sense to use an automated approach. All you'd have to do is run a routine and PC-DMIS would automatically perform the procedures you did previously.

Before doing this, you should be very familiar with PC-DMIS operations.

Create two empty measurement routines:

1. AUTO_MAPS.PRG - This measurement routine is used to create a complete automatic mapping with all probes. It'll measure many angles around the calibration sphere to create all the needed mappings. If you need to run this measurement routine, it'll still take a long time to run (approximately one hour per arm), but at least the process will be automated.

2. AUTO_UPDATE.PRG - This measurement routine is used to update probes when, for example, a tip crashes and you need to update the calibration of a single tip. In this case you just need to have one update measurement routine that uses AUTOCALIBRATE commands. This measurement routine is much shorter to run as it involves only measuring a few orientations on the calibration sphere and is used to adjust the mapping for a new tip.

For now, just create the measurement routines. You'll add content to them in later steps.

The next step provides information on adding commands to AUTO_MAPS.PRG.

Step 13: Add Commands to AUTO_MAPS.PRG

You do not need to create an arm-to-arm map for every extension. While a single map is insufficient for good accuracy, you can obtain good accuracy with two maps. It's better to do one mapping of the short extension followed by one with the long extension. This step will help you build the AUTO_MAPS.PRG routine for the recommended two maps.

Define Parameter Sets for Probe Extensions in AUTO_MAPS.PRG

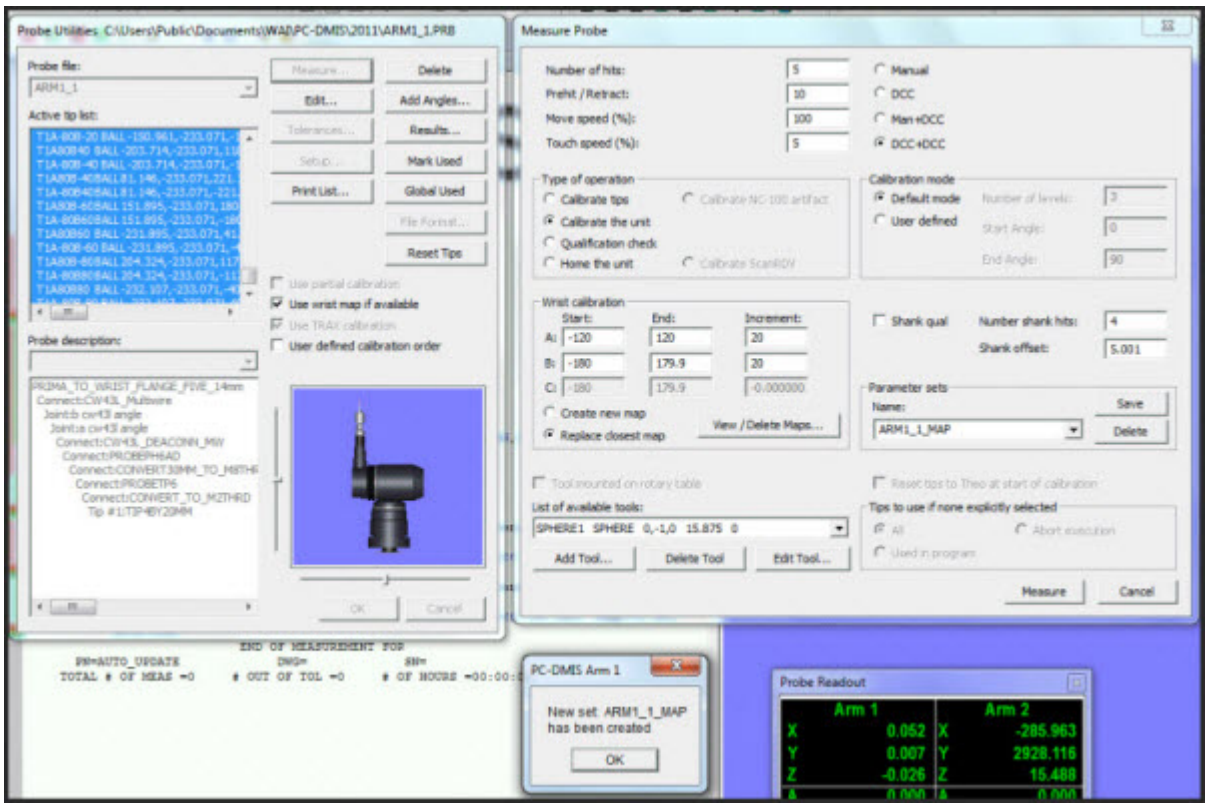
1. Access AUTO_MAPS.PRG and place the Edit window in Command Mode.

2. Start with the ARM1_1 probe. Access the **Measure Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button) for that probe.

3. Define all the parameters you need in this dialog box to create a new arm-to-arm mapping of that probe.

4. Choose **Replace closest map** in the **Wrist Calibration** area.

5. In the **Parameter Sets** area, give the set a name by typing in the **Name** box the name of the probe with a "_MAP" suffix appended (for example, "ARM1_1_MAP").



Example of Parameter Set creation

6. Click **Save**. A parameter set is created for ARM1_1 probe. Click **OK** to close the message box.
7. Repeat 2-6 for each probe, creating a parameter set for each.
8. When finished, you should have these six parameter sets:

   ARM1_1_MAP - For the probe named ARM1_1 (the short extension on arm 1)

   ARM1_2_MAP - For the probe named ARM1_2 (the medium extension on arm 1)

   ARM1_3_MAP - For the probe named ARM1_3 (the long extension on arm 1)

   ARM2_1_MAP - For the probe named ARM2_1 (the short extension on arm 2)

   ARM2_2_MAP - For the probe named ARM2_2 (the medium extension on arm 2)

   ARM2_3_MAP - For the probe named ARM2_3 (the long extension on arm 2)

You only need to create parameter sets for the number of extensions you have on each arm. For example, if you only had two extensions on each arm, you would end up with a total of four parameter sets.

This means, you can now use an AUTOCALIBRATE command to call one of these parameter sets. When the AUTOCALIBRATE statement executes, PC-DMIS will generate a new arm-to-arm mapping with all settings stored in the called parameter set.

Add Preliminary Commands into AUTO_MAPS.PRG

1. Access AUTO_MAPS.PRG.
2. Add in a TEMPCOMP command (temperature compensation) for arm 1. If you're calibrating inside of a climate-controlled environment, the temperature compensation commands aren't needed. See "Compensating for Temperature" and "Using Temperature Compensation with Multiple Arm Calibration".
3. Add a MOVE/SYNC command.
4. Add in a TEMPCOMP command for arm 2.
5. Type a MOVE/SYNC command after the TEMPCOMP command blocks.
6. Type a LOADPROBE/ARM1_1 command to load the probe with the short extension on arm 1. Assign the command to arm 1.
7. Type a LOADPROBE/ARM2_1 command to load the probe with the short extension on arm 2. Assign the command to arm 2.
8. Type a LOADPROBE/ARM1_3 command to load the probe with the long extension on arm 1. Assign the command to arm 1.
9. Type a LOADPROBE/ARM2_3 command to load the probe with the long extension on arm 2. Assign the command to arm 2.

Insert AUTOCALIBRATE Commands for Short and Long Probe Maps in AUTO_MAPS.PRG

1. Access AUTO_MAPS.PRG.
2. Place the cursor after the LOADPROBE commands for the short probes.
3. Insert an AUTOCALIBRATE command (choose **Insert | Calibrate | AutoCalibrate Probe**).
4. Press F9 on the command. The **Calibrate Probe** dialog box appears.
5. In the **Parameter set** list, choose the parameter set for the short probe on arm 1. This is ARM1_1_MAP.
6. Click **OK**. The command updates to use the selected parameter set.
7. Assign the command to arm 1.
8. Repeat 3-6 for the short probe on arm 2. This is ARM2_1_MAP. Assign the command to arm 2.
9. Place the cursor after the LOADPROBE commands for the long probes.
10. Repeat 3-6 for the long probe on arm 1. This is ARM1_3_MAP. Assign the command to arm 1.

11. Repeat 3-6 for the long probe on arm 2. This is ARM2_3_MAP. Assign the command to arm 2.

12. Type a MOVE/SYNC command at the end of the routine.

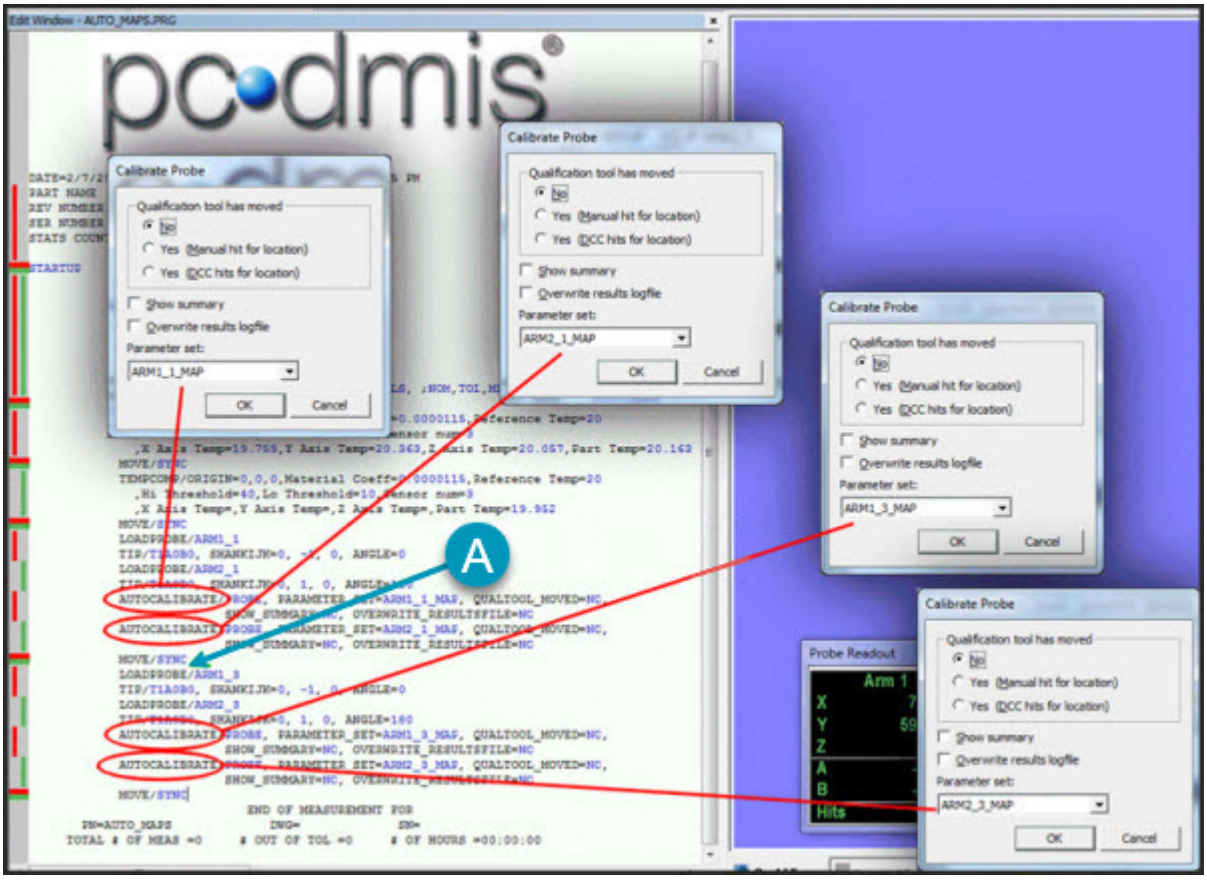
Insert Moves Between Short and Long Probe Maps in AUTO_MAPS.PRG

1. After the AUTOCALIBRATE commands for the short probes and before the LOADPROBE command for the long probes, type a MOVE/SYNC command.

2. After the MOVE/SYNC command, ensure that the tips' angles are adjusted appropriately for drop off at their respective probe changers. You can do this by inserting moves appropriate to your machine's dimensions and arm and probe changer setup.

This completes the information needed for the AUTO_MAPS.PRG.

Your routine should now be organized like this:



Sample AUTO_MAPS.PRG Routine. Safety moves are inserted after the MOVE/SYNC, indicated by green arrow (A).

Notes on using AUTO_MAPS.PRG

You would run this routine to recreate the arm-to-arm mapping in these instances:

Whenever your normal maintenance schedule requires that your mappings should be updated.

Whenever you need to use an entirely new probe. In that case, you would need to add in the LOADPROBE statements for the new probe.

- Whenever your wrist is remounted (for example, after a technician adjusts its electronic compensation).
- Whenever data is lost or corrupted or you're not sure if the mapping is correctly generated.
- Whenever a room situation changes that alters the thermal compensation (for example, if moving the machine out of a climate controlled room).
- Whenever the physical structure of the machine changes.

Before you run this routine, delete old maps. Use the **View | Delete maps** button located in the **Wrist calibration** area of the **Measure Probe** dialog box to do this.

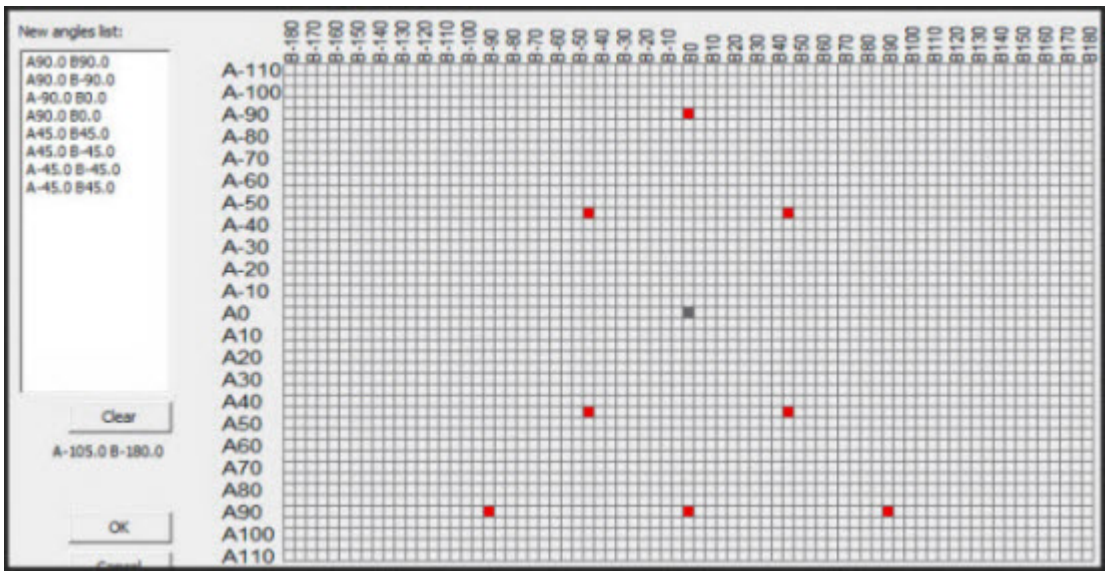
If you're using long extensions, and you aren't running at maximum speeds, expect a minimum of 3-4 hours to run the entire routine. This time is just an estimate, as it depends on your individual machine dimensions, the length of the extensions, and speed percentage used.

The next step provides information on adding commands to AUTO_UPDATE.PRG.

Step 14: Add Commands to AUTO_UPDATE.PRG

Using Step 13 as a guide, do the following:

1. Add in the preliminary commands (`TEMPCOMP` , `MOVE/SYNC` , and `LOADPROBE` commands).
2. In this case, you'll need a `LOADPROBE` commands for all probes (short, medium, and long extensions).
3. Press F9 and ensure **Use wrist map if available** is marked for each. Click **Add Angles**.
4. Use the **Add New Angles** box for each probe, and add in at least 9 angles for the desired probe. These angles are recommended as they result in a sufficiently precise distribution: [0,0] [90,90] [90,-90] [-90,0] [-90,0] [45,45] [45,-45] [-45,-45] [-45,45]

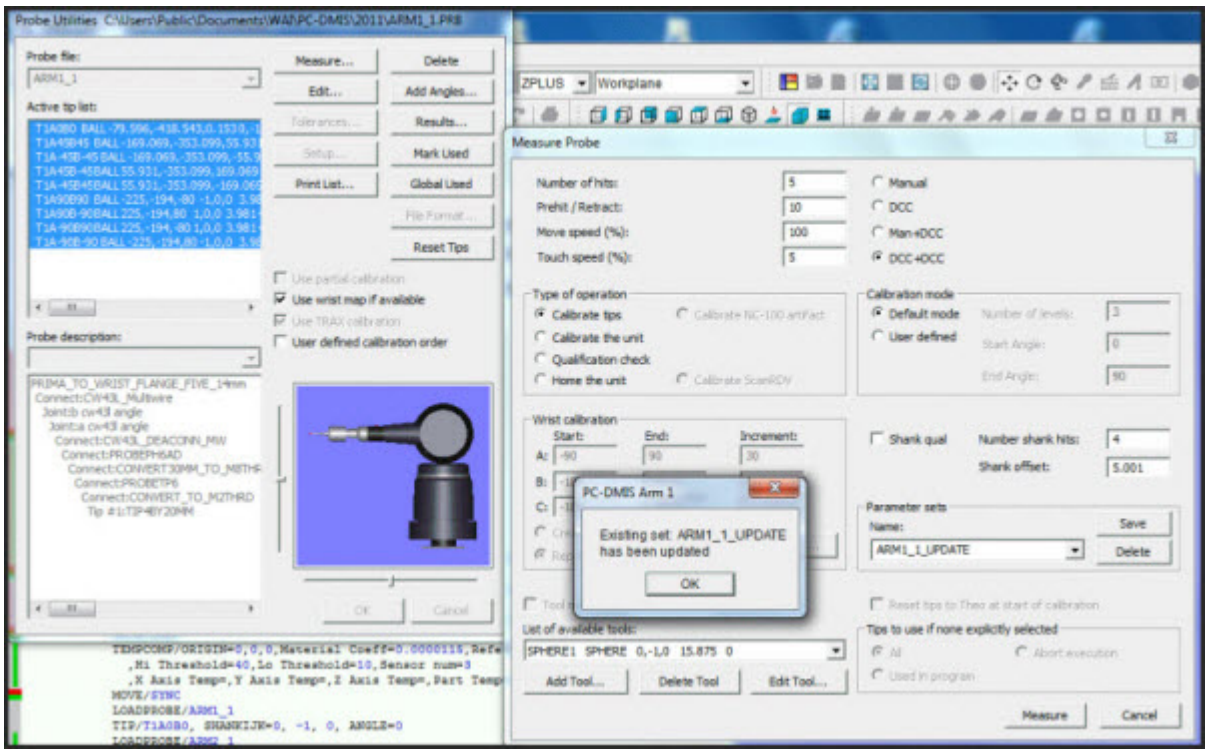


Add New Angle dialog box showing the suggested angles.

For increased precision, you can increase the angles to 17, but this lengthens the execution time of the update.
The recommended 17 angles for this wrist are:
([0,0] [90,0] [90,-45] [90,-90] [90,-135] [90,45] [90,90] [90,135] [-90,0] [45,-20] [45,-65] [45,-110] [45,-155] [45,25] [45,70] [45,115] [45,160])

5. Access the **Measure Probe** dialog box and define parameter sets for each probe. Ensure that **DCC + DCC** and **Calibrate tips** are selected.

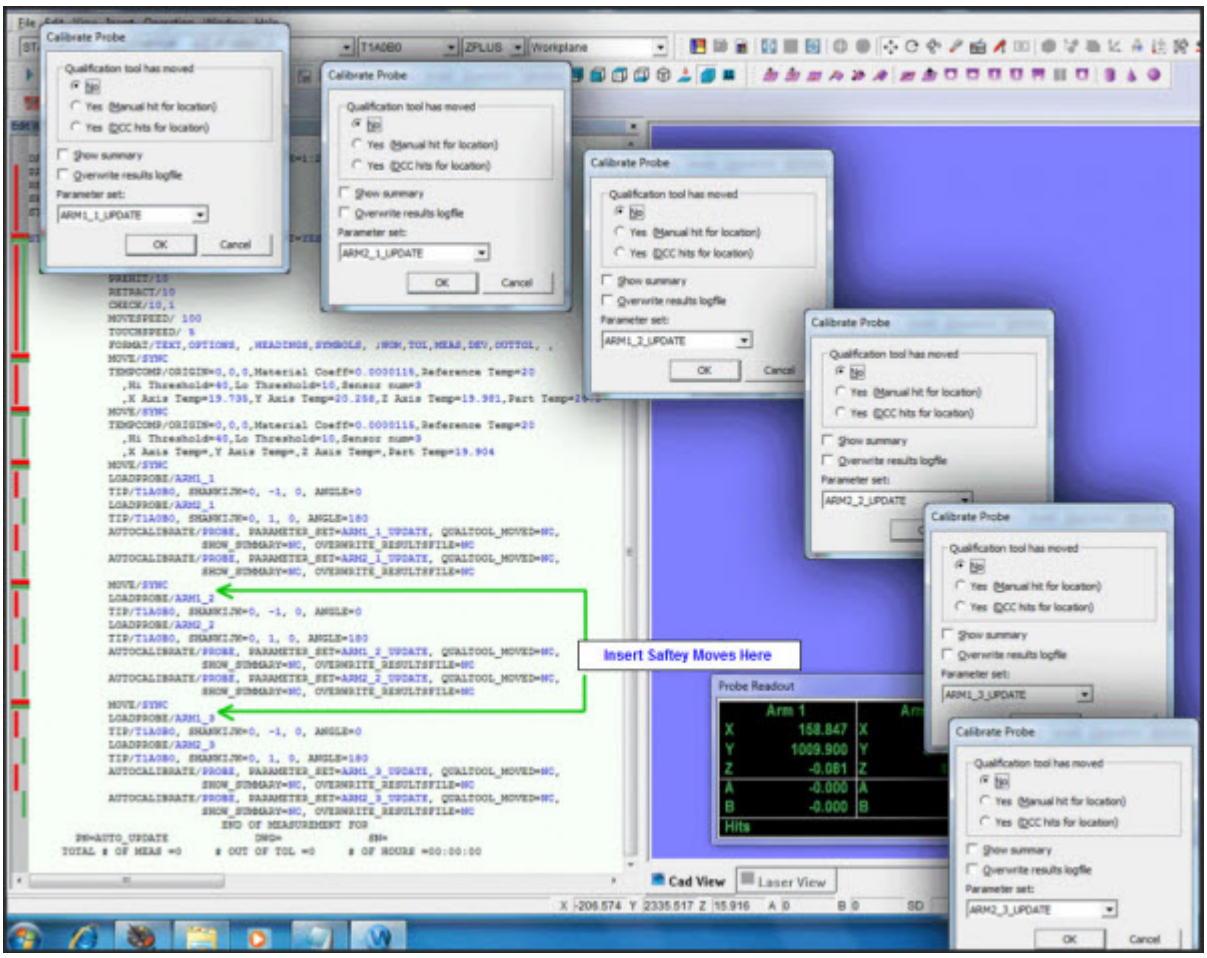
6. Name each parameter set by basing it off the probe name and adding on an "_UPDATE" suffix. For example, ARM1_1 would have a parameter set named ARM1_1_UPDATE.



Example of Parameter Set creation

7. After each set of `LOADPROBE` commands, insert two `AUTOCALIBRATE` commands that correspond to the parameter sets for those loaded probes.
8. Insert a `MOVE/SYNC` command after each `AUTOCALIBRATE` pair.
9. Between each `AUTOCALIBRATE` pair, after the `MOVE/SYNC` command, insert some safety move commands to prevent against possible crashes when the arms move to swap probes in and out of the probe changer.

At this point your measurement routine should be organized like this:

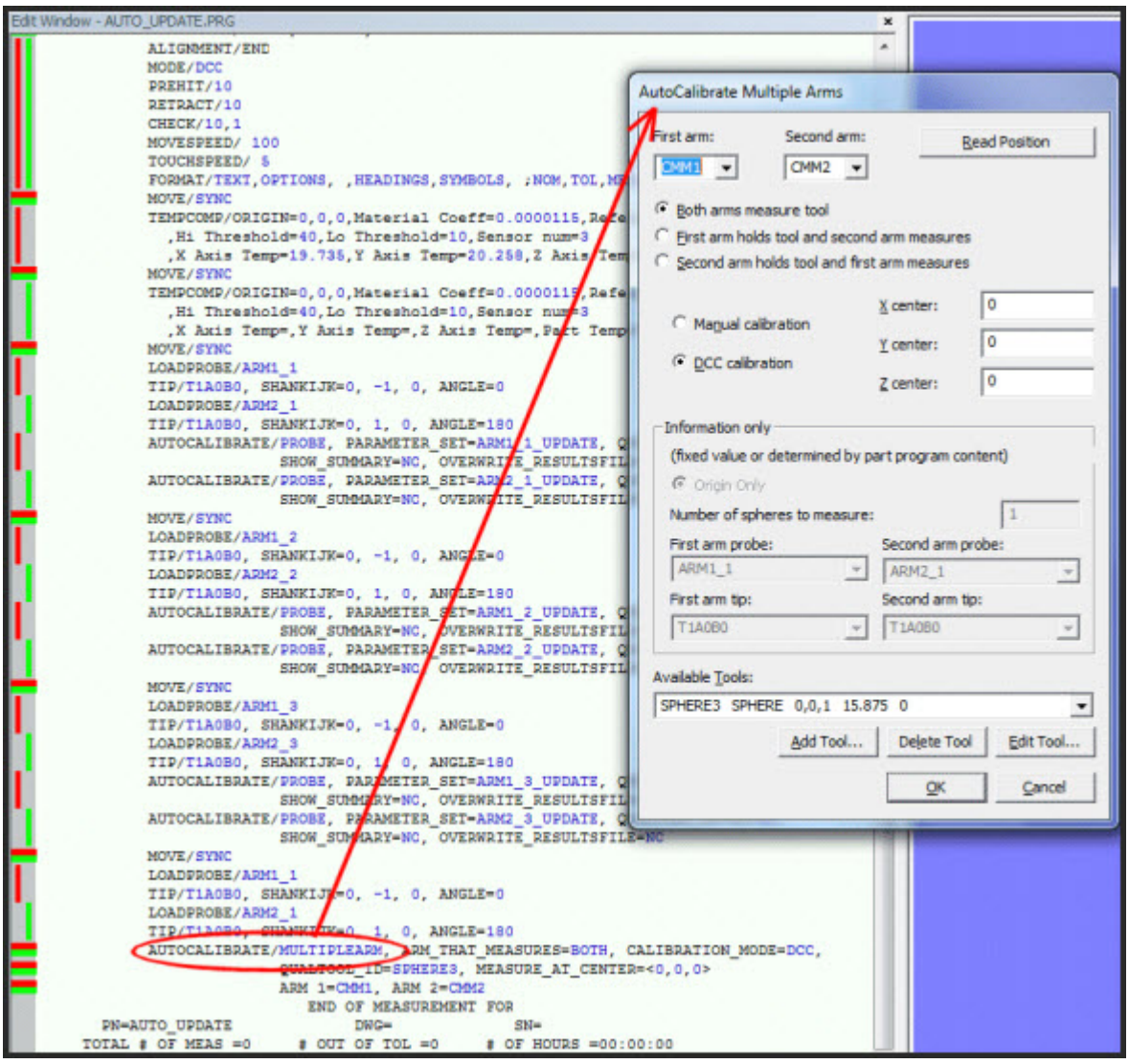


A sample AUTO_UPDATE.PRG measurement routine.

Continue by adding these commands:

1. Go to the end of the measurement routine, add in a `LOADPROBE` command for ARM1_1 and for ARM2_1.
2. Insert an `AUTOCALIBRATE/MULTIPLEARM` command and press F9. The **AutoCalibrate Multiple Arms** dialog box appears.
3. For **First arm**, choose CMM1. For **Second arm**, choose CMM2.
4. Choose **Both arms measure tool**.
5. Choose **DCC calibration**.
6. Choose SPHERE3 from the **Available Tools** list.

This last piece causes the short extension on both arms to go and measure SPHERE3 in order to adjust the relationship one last time between arm 1 and arm 2. Generally, you only need to use this final command after perhaps a month machine usage (or after many measurements) to occasionally adjust the offset and improve accuracy.



AutoCalibrate Multiple Arms Dialog Box

Before running this final portion of the measurement routine, be sure to place calibration spheres (SPHERE1, SPHERE2, and SPHERE3) back on the table.

Notes on using AUTO_UPDATE.PRG

You would run this measurement routine to update the mapping for a specific tip in these instances:

- Whenever you want to adjust the accuracy of the machine.
- Whenever you want to adjust a tip or need to add a new tip.

This measurement routine is used more frequently than AUTO_MAPS.PRG.

A typical `AUTOCALIBRATE` command with update parameters and 1 extension can be done using the Execute Block command.

Running the entire measurement routine for all probe extensions and using the recommended nine angles takes about 30 minutes.

Navigating and Displaying Multiple Windows

Navigating and Displaying Multiple Windows: Introduction

This chapter describes how to use the Window menu to navigate between and view multiple open measurement routines.

The main topics in this chapter include:

- Switching Between Open Measurement Routines
- Arranging Open Windows
- Giving Open Windows the Focus

Switching Between Windows of Open Measurement Routines

Here are some ways you can switch between windows of the active measurement routine:

Click the Next or Previous options: Select the **Next** (**Window | Next**) or **Previous** (**Window | Previous**) menu options to switch to the next or previous window of the currently active measurement routine. When you've reached the end of all the windows, selecting **Next** again does nothing.

Click on a list of open measurement routines: At the very bottom of the **Window** menu, PC-DMIS displays a list of all open measurement routines. You can select which measurement routine you want to have the focus by selecting the file's name from the list.

Click on a title bar: If a file's Edit window or Graphic Display window title bar is showing, click on the title bar to switch to that measurement routine. Cascade or tile the windows for the active measurement routine by selecting the appropriate option from the **Window** menu.

Arranging Open Windows

The following menu options allow you to arrange all open windows for the active measurement routine. These menu options do not affect the Edit window until you remove the Edit window from its docked position by right-clicking on the Edit window and deselecting the **Docking View** menu option.

Cascade

Stacks the windows one on top of the other with just the title bars showing and places the window with the focus on the top of the stack.

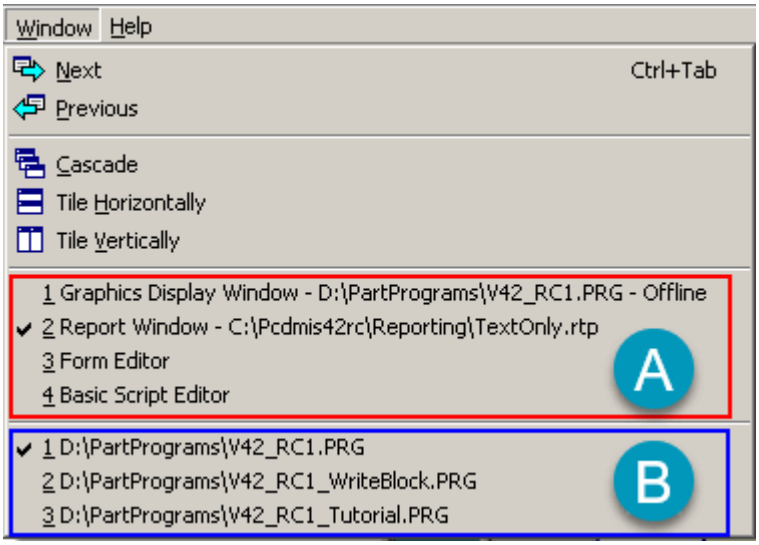
Tile Horizontally

Tiles the windows horizontally and places window with the focus on the top.

Tile Vertically

Tiles the windows vertically and places the window with the focus on the left.

Giving Open Windows the Focus



Sample Window Menu

A. This section shows the list of open windows for the current measurement routine.

B. This section shows the list of open measurement routines.

Above the list of measurement routines, PC-DMIS also lists all open windows in the **Window** menu. Select the window from this menu to give that window the focus.

Working in Offline Mode

Working in Offline Mode: Introduction

PC-DMIS in Offline mode enables you to prepare and debug measurement routines without using a CMM. The ability to program offline has become increasingly important over the last several years. Users of CMMs have become more aware that, to fully realize their investments in CMMs, their equipment must be used to measure parts and not write routines to measure parts.

CMM manufacturers' first attempts at adding offline programming capabilities involved cumbersome and specialized text editors. These products, although of limited use, spurred user interest in offline programming. Driven by this interest, several CAD vendors developed products that allowed users to use CAD models to generate measurement routines.

Although these products were vastly superior to text editors, they had a major cost disadvantage. Since each CMM vendor has their own specific measurement language or languages that were constantly changing or being replaced, the expense of developing and maintaining these products put them out of the reach of all but a few well-financed users.

This situation brought about the development of the DMIS specification, which is a generic CMM language. DMIS allowed CAD vendors to develop measurement routine programming packages that were targeted toward a single language instead of many, which greatly reduced their costs. These savings were passed on to their customers, and offline measurement routine programming became a viable option for a large group of CMM users. However, there was still one problem: What about CMM users whose CAD vendors did not support, and did not plan to support, offline measurement routine programming?

Although many mainframe CAD vendors, driven by major customers, have introduced DMIS extensions to their products, PC-based CAD vendors with their diverse customer bases have shown little interest in this area. Many CMM users, particularly small shops, use PC-based CAD systems exclusively. PC-DMIS brings offline programming capabilities to this group.

With PC-DMIS, programmers who use standard IGES models, which virtually every CAD vendor supports, can generate measurement routines on an inexpensive PC or PC clone without going near a CMM. These measurement routines can then drive any CMM that either runs PC-DMIS or supports the DMIS specification.

The techniques for programming offline are much the same as those used for programming in Online mode. However, as might be expected, the methods used to

qualify probes, take measurements, and debug routines differ from those in Online mode. This appendix describes PC-DMIS programming techniques in Offline mode.

The main topics in this chapter include:

- PC-DMIS Offline Prerequisites
- Offline Probes
- Setting Probe Depth
- Measuring Features Offline
- Executing and Debugging Measurement Routines Offline

PC-DMIS Offline Prerequisites

To use PC-DMIS in Offline mode, CAD data in the form of an IGES model, a DES file, a DXF file, or X,Y,Z,I,J,K data must be available. The "Importing CAD Data or Feature Data" topic in the "Using Advanced File Options" chapter has information on how to import these files into the PC-DMIS system.

IGES Entities Supported

IGES Entity	Description
100	CIRCLE/ARC
102	COMPOSITE CURVE
104	CONIC ARC
106	COPIOUS DATA (multiple point line)
108	PLANE
110	LINE
112	PARAMETRIC SPLINE CURVE (with curves and surfaces option)
114	PARAMETRIC SPLINE SURFACE (with curves and surfaces option)
116	POINT
118	RULED SURFACE (with curves and surfaces option)
120	SURFACE OF REVOLUTION (with curves and surfaces option)
122	TABULATED CYLINDER (with curves and surfaces option)
124	TRANSFORMATION MATRIX
126	RATIONAL B-SPLINE CURVE (with curves and surfaces option)

128	RATIONAL B-SPLINE SURFACE (with curves and surfaces option)
140	OFFSET SURFACE
144/142	TRIMMED SURFACE (with curves and surfaces option)
402	ASSOCIATIVITY INSTANCE
408/308	SUBFIGURE
410	VIEW

PC-DMIS is compatible with IGES 3.0, 4.0, and 5.1.

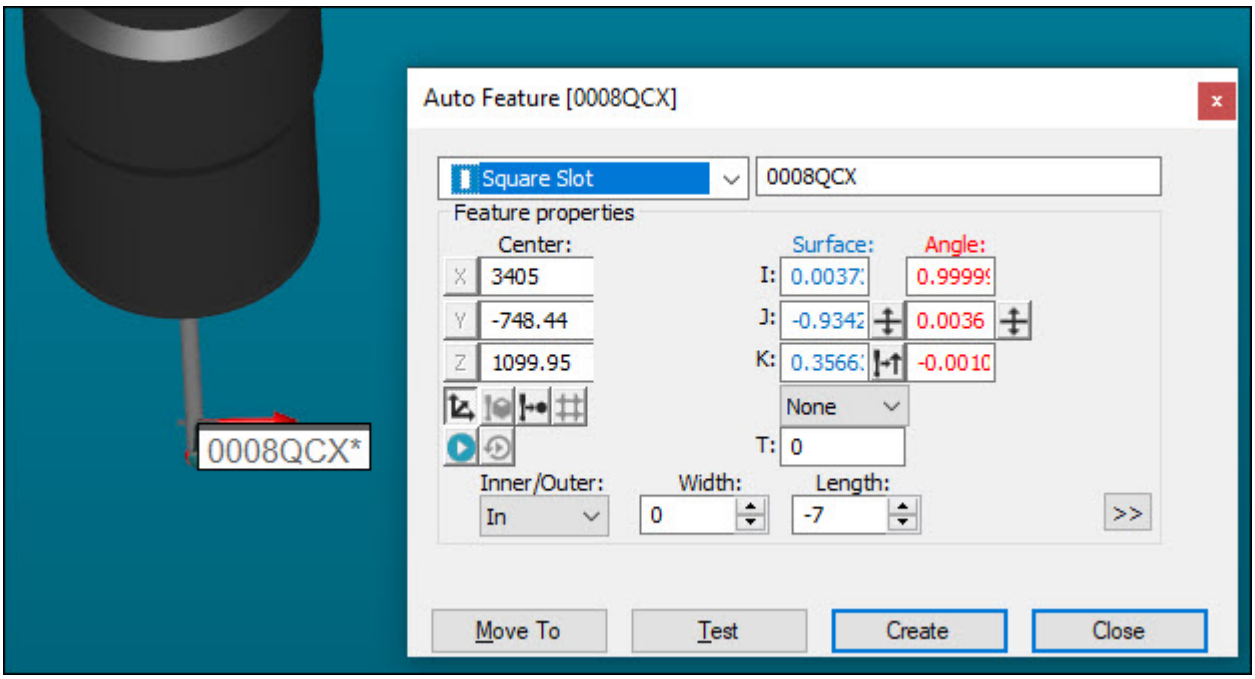
DXF Input

PC-DMIS reads in a DXF (Drawing Interchange File) file as CAD data to be used to create measurement routines. This file format does not support text. It only supports feature data.

This option is not part of standard module of PC-DMIS. Please contact your PC-DMIS software representative if you are interested in purchasing this add-on package.

DES Input

PC-DMIS can read in a DES (Data Exchange Standard) file as CAD data to be used to create measurement routines. The data can come in as feature or fixture data. If it is feature data, after you import, you can click on the feature label in the Graphic Display window to define the type of feature. PC-DMIS displays the **Auto Feature** dialog box for the feature type with the values filled in from the DES point:



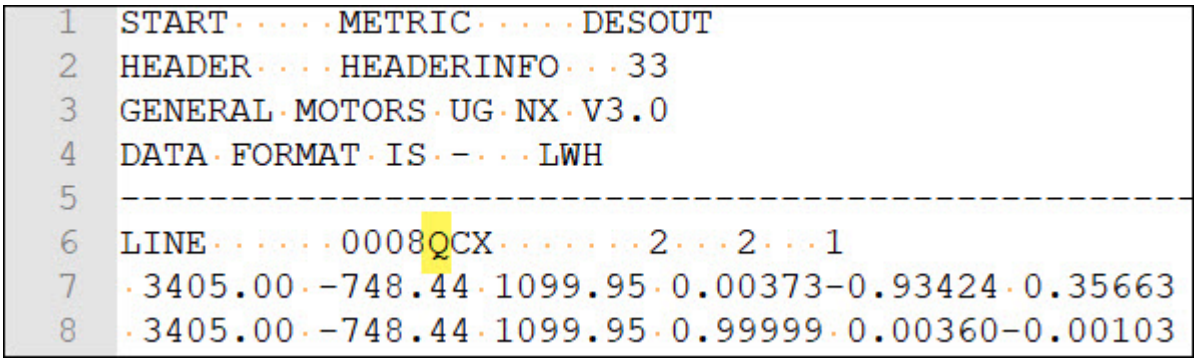
Example showing DES feature interpreted as a square slot feature.

PC-DMIS gets the feature type from the fifth character position of the feature label. So, if the DES feature label is 0008QCX, the character in the fifth position is a Q. These are the available characters and how PC-DMIS interprets them:

CHARACTER	DES Type	PC-DMIS Feature Type
S	Surface	Vector Point
T	Trim	Edge Point
H	Hem Edge	Edge Point
P	Profile	Vector Point
I	Intersection	Vector Point
X	Hole	Circle (in)
Y	Stud	Round Slot
Z	Slot	Circle (out)
V	Square	Square Slot
W	Hexagon	Circle (in)
Q	Square	Square Slot
R	Reference	Vector Point
G	Gap	Edge Point
F	Flush	Vector Point
L	Parallelism	Vector Point

D	Differential	Vector Point

Note that if you open the DES file in a text editor, the fifth character on the feature label is identical to the fifteenth (15) column of the data type LINE:



Example of a DES file with the 15th character highlighted.

For more information in importing a DES file, see "Importing a DES File" in the "Using Advanced File Options" chapter.

XYZ ASCII File

PC-DMIS can read in any ASCII file that contains XYZ (and possibly IJK) data. The file should contain the nominal (theoretical) inspection points that need to be measured.

For detailed information on XYZ ASCII files, see "Importing an XYZIJK File" in the "Using Advanced File Options" chapter.

Offline Probes

When PC-DMIS is in Offline mode, you can use all of the probe definition and calibration capabilities as you can in Online mode. However, you can only enter values. You cannot make measurements. (For example, you cannot actually measure a calibration artifact to find the diameter of a probe.)

For information on how to define a probe, see "Defining Probes" in the "Defining Hardware" chapter.

We recommend that you use the same probe configuration in the offline measurement routine as the online measurement routine.

Setting Probe Depth

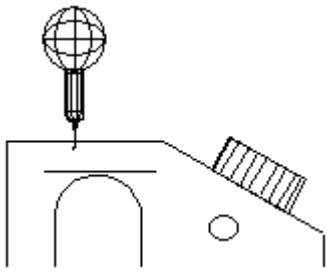
To program measurements in Offline mode, it is important to set the depth of the probe at a specified distance (in relation to the surface of the current workplane). PC-DMIS offers several methods to set the probe's depth.

PC-DMIS must be in Program Mode to use any of these techniques.

Setting Approximate Probe Depth

In most circumstances, it is only necessary to set an approximate probe depth to properly measure a feature. To use PC-DMIS in Offline mode to do this:

1. Position the mouse cursor on the drawing at the desired depth for hit taking.
2. Right-click at the current location. PC-DMIS then redraws the probe in its new position.



Setting Probe Depth

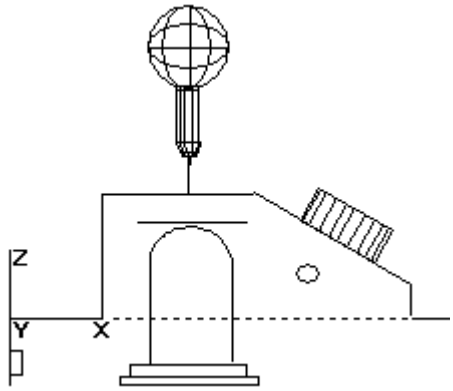
Setting Probe Depth on a Feature

To position the probe on a specific feature (i.e., a plane):

1. Move the cursor close to the feature.
2. Click the right mouse button.

PC-DMIS "snaps" the probe to the closest CAD element and displays the message: "Precise Depth Set on".

The Status bar displays the current number of hits and the probe's location.



Example showing the setting of the precise probe depth on a feature.

Setting Probe Depth on a Sphere

PC-DMIS offers two procedures that allow you to set the probe depth on a sphere. Where the probe is located in relation to the centerline of the drawing determines where the hit is taken on the sphere. If the probe is below the centerline, PC-DMIS takes the hit on the bottom of the sphere. To take a hit on the top of the sphere, the probe depth must be set above the centerline.

For additional information on the following procedures, see "Measuring Features Offline".

Three-Dimensional Procedure

To set the precise depth on a three-dimensional spherical surface:

1. Move the animated probe to the desired circle.
2. Hold down the right mouse button.
3. Release the mouse button. The precise depth is set on the sphere.

PC-DMIS positions the probe on the side of the feature where the mouse button was first held. This determines the type of feature that is measured. If the probe snaps to the outside of a circular CAD element, the hits are placed outside the circle. If the probe snaps to the inside of the same element, the hits are placed inside the circle. The circle’s three-dimensional origin must be the same location as the sphere’s center.

Once the precise depth has been set, all the points that are generated snap on to the spherical surface.

Two-Dimensional Procedure

When you use a two-dimensional drawing, PC-DMIS requires at least two views of the sphere. The sphere should be visible as a circle (or arc) in both views.

1. Set the precise depth for two of the axes using one of the views. PC-DMIS displays this message: "Precise Depth Set on". See "Setting Probe Depth on a Feature" for setting the precise depth.

2. Use the second view to set the precise depth for the third axis. PC-DMIS displays this message: "Precise Depth Set on Sphere". This procedure finds the true three-dimensional center point of the sphere.

Once you have set the precise depth, all the points that are generated snap on to the spherical surface.

Setting Probe Depth on a Cone

PC-DMIS offers two procedures that allow you to set the probe depth on a cone. For additional information on the following procedures, see "Measuring Features Offline".

Three-Dimensional Procedure

To set the precise depth on a conical surface, PC-DMIS requires that two CAD circles (or arcs) on the cone be displayed. It is suggested that you use two views of the surface for this procedure, but it is not required. (Isometric views are also a viable way to set the probe depth on a cone.)

1. Set the precise depth for one end of the cone using one of the circles. PC-DMIS displays this message: "Precise Depth Set on."

2. Set the precise depth for the other end of the cone using the second circle. PC-DMIS displays this message: "Precise Depth set on Cone."

Once the precise depth is set, all the points that are generated snap on to the conical surface. Single hits can be taken on cones by holding down the left mouse button. Click the left mouse button to prompt PC-DMIS to take equally-spaced hits around the cone.

Two-Dimensional Procedure

To set the precise depth on a conical surface in two-dimensional drawings, PC-DMIS requires that you define the length between the two circles (as described above). Because these circles are at the same depth, it is necessary to also define the precise depth of a line. This can be either a straight line or a line on the edge of the cone. After you set the precise depth on the circles, hold down the right mouse button next to the line that is to be used for the length.

Once you have set the precise depth, all the points that are generated snap onto the conical surface. To take single hits on cones, hold down the left mouse button. Click the left mouse button to prompt PC-DMIS to take equally-spaced hits around the cone.

Keying in Probe Depth

It is sometimes necessary to set the probe at some specific location in space. To do this:

1. Click on the X, Y, Z portion of the Status bar while in Program Mode (or select the **Operation | Move To** option described in the "Inserting Move Commands" chapter)

to open the **Auto Move Point** dialog box. The default setting indicates the probe's current location.

2. Change the X, Y, Z to the desired values. If you mark the **Store Move** check box, you can add the `MOVE POINT` command to the routine. You can also select the **Increment moves** check box and the **OK to Move** check box.

3. Once the new X, Y, or Z values are entered, click on **Done** and PC-DMIS moves the animated probe to the new position.

Measuring Features Offline

PC-DMIS offers several methods for you to program measurement routines in Offline mode:

- You can hold down the left mouse button to take a hit.
- You can press the Alt + "-" (minus) keys to remove the last hit as long as the measurement process isn't complete.
- You can press the End key to complete the measurement process. PC-DMIS continues to accumulate hits in the hits buffer until you press the End key.

Automatic Measurements

Based upon the IGES definition of circular and linear feature types, PC-DMIS can make some assumptions about the way they should be measured. Take advantage of these assumptions to speed up the measurement routine programming process.

Circular Features

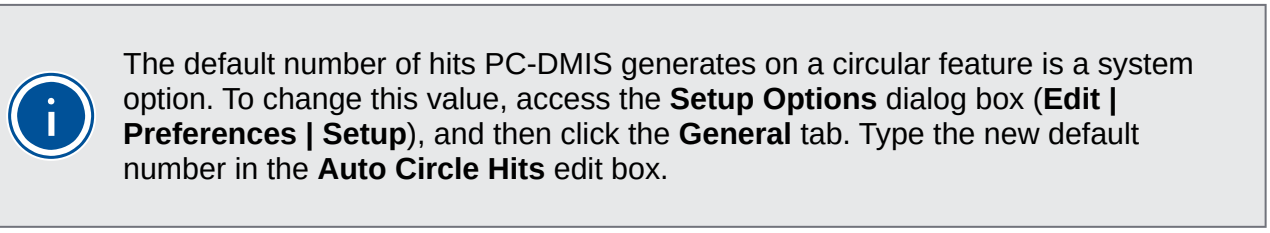
PC-DMIS can automatically generate hits for circles, cylinders and arcs. To do this:

1. PC-DMIS must be in Program Mode.
2. Move the pointer close to the circumference of the feature.
3. Click the left mouse button. PC-DMIS then generates equally spaced hits on the feature at the current probe depth. (See "Setup Options: General Tab" in the "Setting Your Preferences" chapter for setting the number of hits for circles.)

Follow these measurement rules for circular features.

- For an inside diameter (ID), place the cursor just inside the feature.

- For an outside diameter (OD), place the cursor just outside the feature.
- To automatically program cylinders, take at least two sets of hits at different probe depths.
- When programming an arc, PC-DMIS spaces the hits along the length of the arc.
- To measure a sphere or a cone, first set precise depth on the sphere or cone before generating any hits. See "Setting Probe Depth on a Sphere" and "Setting Probe Depth on a Cone".



The default number of hits PC-DMIS generates on a circular feature is a system option. To change this value, access the **Setup Options** dialog box (**Edit | Preferences | Setup**), and then click the **General** tab. Type the new default number in the **Auto Circle Hits** edit box.

Linear Features

The default number of hits PC-DMIS generates on a linear feature is a system option. To change this value, access the **Setup Options** dialog box (**Edit | Preferences | Setup**), and then click the **General** tab. Type the new default value in the **Auto Line Hits** edit box.

PC-DMIS can automatically generate hits for lines and planes. To do this:

1. PC-DMIS must be in Program Mode.
2. Move the pointer close to the line.
3. Click the left mouse button.

Follow these rules for measuring linear features:

- PC-DMIS generates equally spaced hits along the length of the line at the current probe depth. (See "Setup Options: General Tab" in the "Setting Your Preferences" chapter for setting the number of hits for lines.)
- The cursor must be placed on the side of the line where the hits are to be taken.
- To automatically program planes, at least *two sets* of hits at different probe depths must be taken.

Surface Features

You can use the UV scan to automatically place points on a surface along its UV direction. In Program Mode and Surface Mode, press the left mouse button on the surface you want to select. A dialog box opens where you can enter the start and end UV values and specify the number of points along each UV direction.

Discrete Measurements

Automatic measurements speed up the programming process, but it is sometimes necessary due to either part geometry or feature type to precisely place hits on a feature. There are two techniques for placing hits.

Placing Hits on a Surface

In many instances you need to precisely position the hits on a surface (such as a plane, sphere, or cone measurement). To do this:

1. Move the cursor to the position where you wish to take the hit.
2. Hold down the left mouse button (do not move the mouse).
3. Release the button. PC-DMIS programs the hit at that point.

You must hold the pointer in one position while holding down the button. Otherwise, PC-DMIS may misinterpret your intention. If the probe tip snaps onto a feature after the button is released, instead of the surface, then the mouse was moved during the process. In that case, press the Alt + "-" (minus) key combination to remove the hit and begin again.

You must set the precise depth before you place discreet hits on a cone, sphere, or plane.

Placing Hits on a Feature

In many instances you need to precisely position hits on a feature other than a plane. To do this:

1. Move the cursor close to the position where you wish to take the hits.
2. Hold down the left mouse button.
3. Move the probe towards the position where you want to take the hit. (the probe *must* be moved at least 1/8" on the screen).
4. Release the button.

PC-DMIS "snaps" the hit onto the feature. PC-DMIS locates the hit on the side of the feature where you positioned the cursor when you first held down the mouse button.

Ending a Measurement

To end a measurement in Offline mode, press the END key.

Executing and Debugging Measurement Routines Offline

The execution of measurement routines in PC-DMIS in Offline mode behaves the same as in Online mode. The Edit window provides immediate access to all of the commands in a measurement routine, which makes it as easy to fine-tune an offline measurement routine as a routine done on a CMM. For an overview of the many edit options in PC-DMIS, see the "Editing a Measurement Routine" chapter.

Pay careful attention to the probe animation to detect collisions and misplaced hits. The best way to do this is to use the probe path and collision detection functionality to determine collision points between the probe and the part.

The **Operations | Graphic Display Window | Collision Detection** menu option provides an animated graphical representation of the probe's path along the part. This tool for probe path editing can help you when programming measurement routines in Offline mode.

To access the **Collision Detection** option:

1. In the Edit window, mark the features you want to use in the probe path. (See "Marking Commands for Execution" in the "Editing a Measurement Routine" chapter.)

2. Select the **View | Path Lines** option. PC-DMIS displays the path lines of the probe created during the learning portion of the measurement routine.

3. Select **Operation | Graphic Display Window | Collision Detection**. PC-DMIS moves an animated probe along the path lines. It shows any collisions in red on the part and in the **Collision Detection** dialog box. Once execution finishes, a **Collision List** dialog box shows where any collisions occurred. The "Showing, Animating, and Moving Path Lines" topic in the "Editing the CAD Display" chapter has additional information on how to use path lines and detect collisions.

Manipulating Animation and Execution Speeds

You may also find it helpful to manipulate animation and execution speeds while you debug measurement routines. Several settings exist so that you can fine-tune these speeds to meet your needs. For information on these settings, see the "Setup Options: Animation tab" topic in the "Setting Your Preferences" chapter.

Using Inspection Plans in PC-DMIS

Using Inspection Plans in PC-DMIS: Introduction

The following topics describe how to create a measurement routine from your Inspection Plan inside of PC-DMIS:

- Importing an Inspection Plan from Planner
- Mapping Features to Sensors
- Importing an Inspection Plan from Legacy PC-DMIS Planner

These other topics explain how to update your measurement routine later by using the Change Management, how PC-DMIS uses path optimization, void detection, and clearance moves inside the measurement routine:

- Updating Inspection Plans with the Change Manager
- Optimizing the Path
- Void Detection
- Inserting Clearance Moves Automatically

Importing from Planner (.plancad, .planxml, .xml)

You can import the Inspection Plan from Planner into PC-DMIS by importing a .planxml or .xml file exported from Planner. The .planxml or .xml file contains XML elements with the inspection plan information.

To import a .plancad, .planxml, or .xml file:

1. Select **File | Import | Inspection Plan** to open the **Import** dialog box.
2. From the file type list, select the file type that filters the current folder. If you select **All Plan Files**, you can import a .plancad, .planxml, or .xml file.

- If you import a .PlanCAD file, the software imports both the plan features and the embedded CAD model.
- If PC-DMIS detects changes in the original .PlanCAD file that you're referencing, it asks if you want to use the Change Manager to handle those changes. For information on the Change Manager, see "Updating Inspection Plans with the Change Manager".

Your file to import should have the same measurement units as the measurement routine that takes the import. If the files don't have the same measurement units, you can still import the file, but the measurement routine ends up with incorrect information. For example, if you have a circle with an X value of 4 inches in the file to import, the software imports it as 4 millimeters if the measurement routine that takes the import expects millimeters.

3. Click **Import** to import the inspection plan and create a measurement routine from it. For .plancad and .planxml files, once you click **Import**, the software opens the **Sensor Mapping** dialog box so that you can map features to sensors. For more information, see the "Mapping Features to Sensors" topic.

PC-DMIS uses entries to determine the settings for the imported features. You can use the Measurement Strategy Editor or the **Auto Feature** dialog box to modify these settings.

Mapping Features to Sensors

You can use the **Feature - Sensor Mapping** dialog box to assign or map features to probes (sensors). A sensor with a mapped feature measures that feature during execution.

To open the **Feature - Sensor Mapping** dialog box, do one of the following:

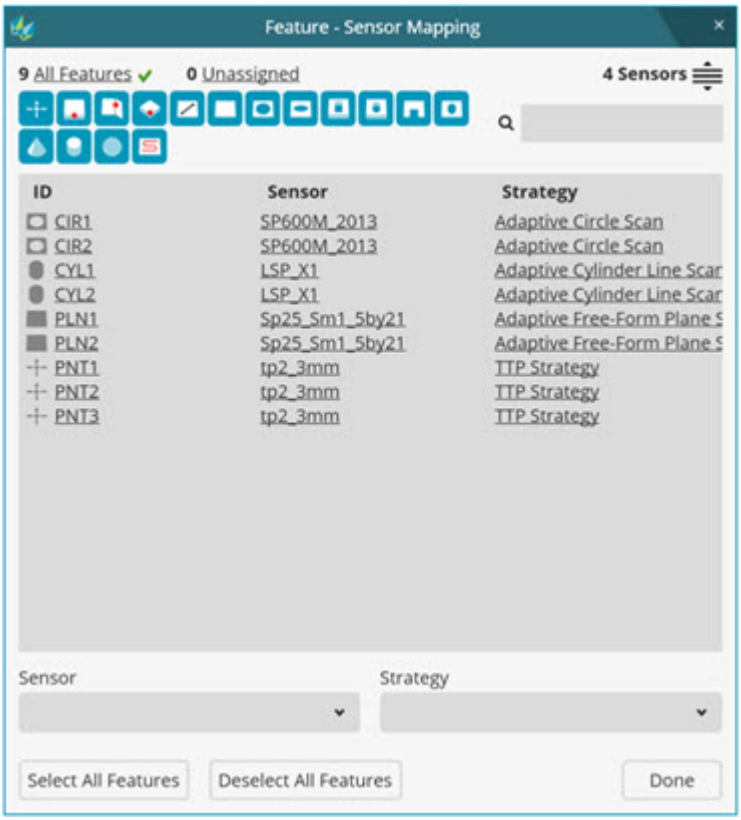
- Select **Edit | Preferences | Feature - Sensor Mapping**.

  The software opens an empty dialog box. You can then use QuickFeatures to populate the dialog box. For more information on QuickFeatures, see "Creating QuickFeatures" in the "Creating Auto Features" chapter.

  If the menu item is disabled, be sure to close any instances of the Measurement Strategy Editor.

- Select **File | Import | Inspection Plan**. Then choose a supported inspection plan file type (.plancad or .planxml) and click **Import**.

The software populates the dialog box with the features in the inspection plan. You can then use QuickFeatures to populate the dialog box with additional features.



Feature - Sensor Mapping Dialog Box

To Assign a Feature to a Sensor

1. In the Edit window, position the cursor after the `LOADPROBE` command. If the cursor is not positioned after the command, QuickFeature selection is possible, but the **Edit** window will not be populated with the selected features.

2. From the **Feature - Sensor Mapping** dialog box, from the feature list, select one or more features. You can then use the feature icons to filter by feature type. The software highlights the features that you select from this list in the Graphic Display window. Press the Shift key and position your mouse pointer over a feature in the Graphic Display window. Then click the feature to add it to the list. The features are not added to the Edit window until you click **Done**.

   > You can use the Shift key to select multiple sequential features and the Ctrl key to select multiple non-sequential features. Each selected feature is highlighted in the Graphic Display window.

3. From the **Sensor** list, select the probe to which you want to assign the selected feature or features. Any new features added to the list are also assigned this sensor.

4. If you want to use a specific strategy for those features, from the **Strategy** list, choose a strategy.

5. Click **Done** to complete the mapping and to continue the import process.

> Be patient while the software does post processing and finishes the import. This process can take a long time. On a standard test computer, 18 features mapped to two probes took about a minute.

6. Once the software finishes the import, the Edit window shows `LOADPROBE` commands for each probe you selected. After each `LOADPROBE` command, the Edit window shows the features that you mapped to that probe.

To Organize Features by Sensor Order

You can also define a probe order and organize your imported features according to that order.

1. From the **Feature - Sensor Mapping** dialog box, start to assign features as usual according to the above procedure.

2. Before you click **Done**, click the **Sensors Ordering** button (top right of dialog box) to open the **Sensors Ordering** dialog box. This dialog box shows a list of the current mapped probes.

3. For one or more probes, select the probe, and then drag and drop (or click the up and down arrows) to change the probe order.

4. From the **Sensor Ordering** dialog box, click **Done** to use the order when PC-DMIS finishes the import.

5. From the **Feature - Sensor Mapping** dialog box, click **Done** to finish the import.

6. Once the software finishes the import, the order of `LOADPROBE` commands matches the probe order you defined in the **Sensors Ordering** dialog box.

Description of the Feature - Sensor Mapping Dialog Box

All features - This text selects all the features in the feature list. It also shows the number of features.

Unassigned - This text selects those features that you have not yet assigned to a sensor.

Sensors - This text shows the current number of sensors in the sensor list. The **Sensors Ordering** button next to **Sensors** opens the **Sensors Ordering** dialog box. You can use the **Sensor Ordering** dialog box to define a sensor order and organize your imported features according to that order.

- The search box filters the feature list by the text you type here.

- The feature icons filter the feature list. Features that match the selected icons appear in the feature list.

Feature list - This list shows all the imported features in the inspection plan, along with the sensor and strategy assigned to them. Click any of the column headers to rearrange the list in alphanumeric order based on that column. When you click **Done**, the features are organized in the same order in the Edit window with the `LOADPROBE` command for the selected sensor added.

The features list contains these columns:

> **ID** - This column shows a symbol and name for each imported feature.
>
> **Sensor** - This column shows the sensor name assigned to that feature.
>
> **Strategy** - This column shows the current strategy for the feature. If nothing appears here, then the software uses the default strategy for that feature. For information on how to define strategies, see "Using the Measurement Strategy Editor" topic in the "Setting Your Preferences: Introduction" chapter.

Sensor - This list shows all the sensors available to the measurement routine.

Strategy - This list shows all the available strategies.

Select All Features - This text selects all the features in the feature list.

Deselect All Features - This text deselects all the features in the feature list.

Done - This button applies the feature to sensor mapping as shown in the **Feature - Sensor Mapping** dialog box.

Importing a Legacy Planner Inspection Plan (.ip)

To import a Planner Inspection Plan as an .ip file, perform the following tasks first:

- Define Parameters and Rules
- Perform an optimization step on the imported plan
- Ensure that the probe won’t collide with the part by inserting proper move commands

Once you have completed these tasks, you can then execute your imported inspection plan as a newly-created PC-DMIS measurement routine.

If you have problems using the PC-DMIS Planner commands inside PC-DMIS, ensure that there is an *InsPlan.dll* file located in the directory where you installed PC-DMIS.

Inspection Plan Default File (.ipd)

You must have an Inspection Plan Default file or .ipd to import a .ip file into PC-DMIS. The "Legacy Planner Parameters" topic provides more information on this file.

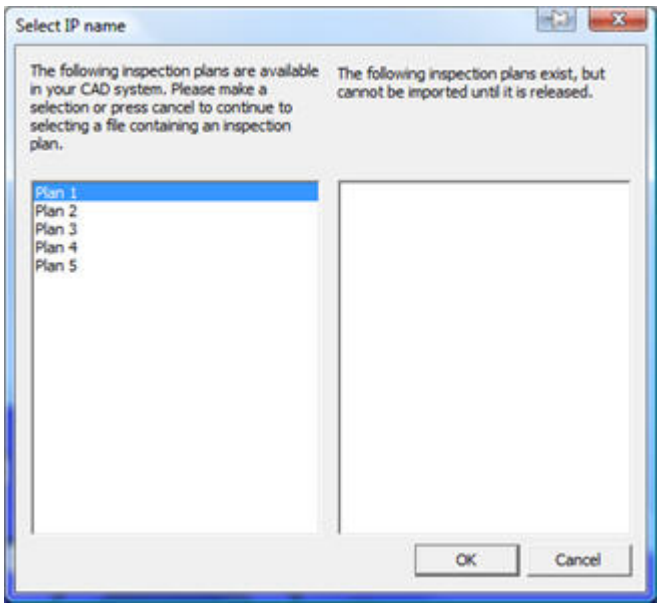
Once you've performed the above preparatory steps, you can import the legacy inspection plan and any rules by importing an inspection plan's .cad file that has the inspection plan embedded in it or by importing a previously-exported inspection plan.

To import an embedded Inspection Plan from a CAD file:

1. Select **File | Import | CAD**. The **Open** dialog box appears.

   When importing a CAD file using the **CAD by Reference** menu option, PC-DMIS updates the file accordingly if any changes were made to the IP CAD file by the legacy PC-DMIS Planner application.

2. Browse to and select the needed IP CAD file, and click **Import**. PC-DMIS imports the IP CAD file.
3. Select **File | Import | Inspection Plan**. The **Select IP name** dialog box appears.



Select IP name Dialog Box

4. Select the needed Inspection Plan, and click **OK** or click **Cancel** to browse to and select an exported Inspection Plan via the **Open** dialog box.

5. Follow any on-screen instructions.

    - Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Legacy PC-DMIS Planner Parameters and Rules" topic for more information.

    - If the insertion point is not at the end of the measurement routine, you are prompted to move the cursor to the end of the measurement routine before continuing.

    - If the CAD file comes from a CATIA v5 model, any comments associated with a feature in the eTool data of the CATIA model are imported as Feature Locator Text for the feature. Note that eTool is a proprietary format used by Chrysler. For information on Feature Locator Text, see the "Providing and Using Feature Locator Instructions" topic in the PC-DMIS CMM documentation.

    - Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See "Optimizing the Path" for more information.

    - Specify the options for "Inserting Clearance Moves Automatically" and click **OK** to complete this process or click **Cancel** to skip this step.

To import an Inspection Plan from an exported .ip text file:

1. Select **File | Import | Inspection Plan**. An **Import** dialog box appears.

2. Browse to the directory containing your file with an .ip filename extension.

3. Select the inspection plan, and click **Import**.

4. Follow any on-screen instructions.

- Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Legacy PC-DMIS Planner Parameters and Rules" topic for more information.
- If the insertion point is not at the end of the measurement routine, you are prompted to move the cursor to the end of the measurement routine before continuing.
- Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See "Optimizing the Path" for more information.
- Specify the options for "Inserting Clearance Moves Automatically" and click **OK** to complete this process or click **Cancel** to skip this step.

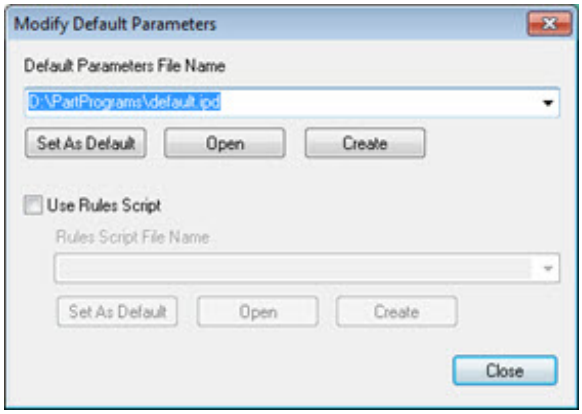
Once you've imported your inspection plan, the imported commands are preceded by a start comment that provides the following basic information for the imported inspection plan:

- Date and time
- The .ip file name with path
- The .ipd file name and path

An end comment indicates the end of the imported inspection plan.

Legacy Planner Parameters

You can use **Edit | Preferences | Inspection Plan Defaults** menu option to open the **Modify Default Parameters** dialog box. You can use this to edit the inspection plan default parameters file (.ipd).



Modify Default Parameters Dialog Box

The Inspection Plan Defaults File (.ipd)

An inspection plan defaults (.ipd) file determines the following:

- The parameters for an inspection plan imported into the software.

- The parameters for Auto features, Constructed Features, and Dimensions.
- How to take hits from features within the inspection plan.
- How to display comments and features.

During an .ip import, if you modify an .ipd file to contain feature parameters and sets it as the default, the software modifies the settings. It then adds the new feature and uses the settings to determine its properties.

The Inspection Plan Rules File (.ipr)

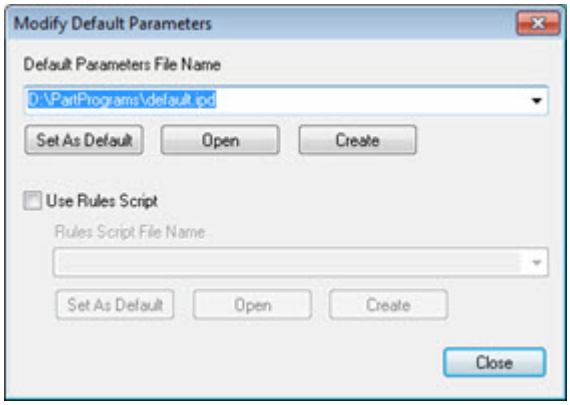
An inspection plan rules (.ipr) file is a Basic script that tells the software how to perform automatic processes on the inspection plan when its imported. Rules can be modified for certain criteria. For example, the number of hits taken on a circle based upon the diameter of the circle and so forth.

Before importing an inspection plan from a CAD file or an .ip or .xml file, you need to set up these files to apply to imported inspection plans.

Creating and Modifying an Inspection Plan Default (.ipd) File

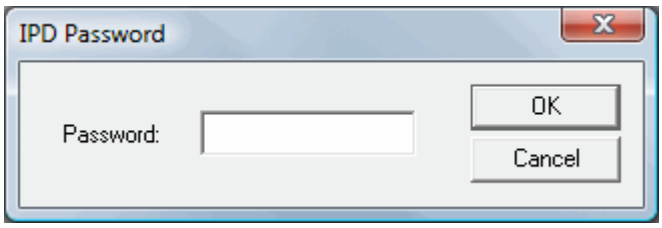
To create a new .ipd file or make changes to an existing .ipd file, follow this procedure:

1. Select the **Edit | Preferences | Inspection Plan Defaults** menu option to display the **Modify Default Parameters** dialog box.

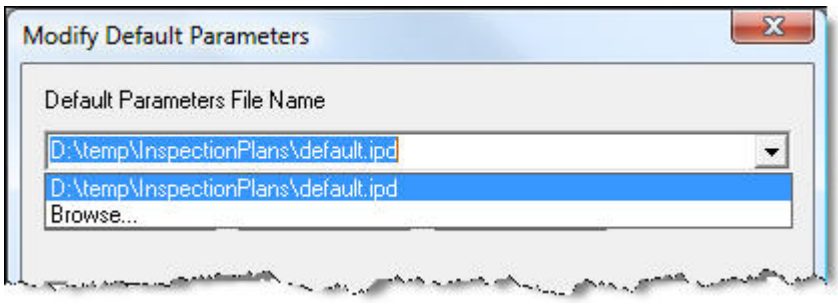


Modify Default Parameters Dialog Box – PC-DMIS Planner Default File

If password protection is enabled for PC-DMIS settings, you need to type the correct password into the **IPD Password** dialog box to proceed. The password is case-sensitive. For more information on password protection, see the "Password" topic discussed under the "Setup Options: General Tab" topic in the "Setting Your Preferences" chapter.



2. From the **Modify Default Parameters** dialog box, in the **Default Parameters File Name** list, select the file that you want to use. If the file isn't listed, click **Browse** to locate and open one from your directory.

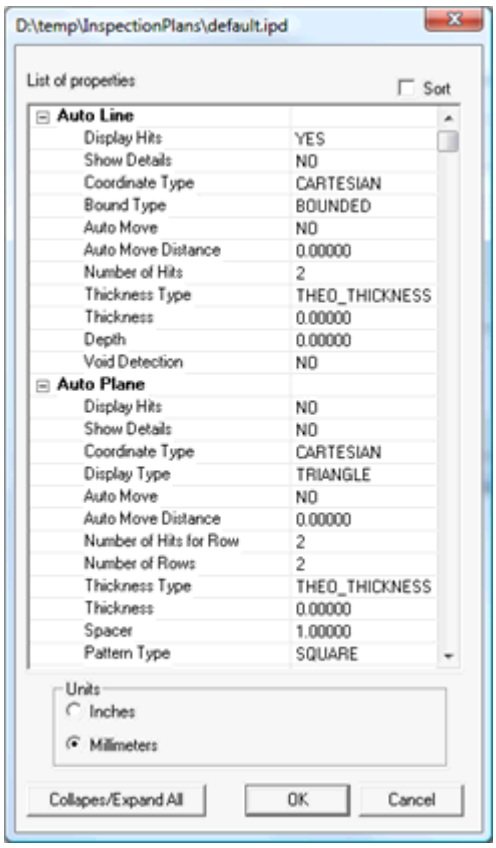


Default Parameters File Name List - Default.ipd

> The default path and file name used for your .ipd file is stored in the `Parameters File` entry in the **Planner** section of the Settings Editor. If no file name is specified, PC-DMIS uses the default value of <PC-DMIS Install Directory>\default.ipd.

3. Decide if you want to edit parameters on an existing .ipd file or edit parameters on a new one.

   - If you want to open and modify an existing .ipd file, click the **Open** button to show the **Default Parameters** dialog box for the selected .ipd file.
   - If you want to create a new .ipd file, click **Create** to show the **Default Parameters** dialog box for the new .ipd file.



Default Parameters

4. To edit a property, from the right column, click on that property's value to show a drop-down list.

5. Select the drop-down list, and choose a different value. These values may represent distance, amount, or some measured value. You can select values from option lists or toggle fields.

    - Option lists provide a list of choices relevant to the associated item.

    - Toggle fields provide only two choices, such as ON/OFF, YES/NO, or TRUE/FALSE.

6. If you want to sort the **List of properties** alphabetically, click the **Sort** check box.

7. Select either **Inches** or **Millimeters** to determine how units are displayed in the **Default Parameters** dialog box. PC-DMIS also converts imported Inspection Plans to the specified units if they are different from the Inspection Plan units.

8. Click the **Collapse/Expand All** button to collapse or expand the **List of properties**.

If you do not achieve the desired results for imported Inspection Plan items, you may need to update the needed parameter and recreate the .ipd file. Also, note that .ipd files are only compatible with the version in which they were created. For example, an .ipd created for PC-DMIS 4.2 is not compatible with PC-DMIS 4.3.

9. Click **OK** to open the **Save As** dialog box, or click **Cancel** to abort any changes that have been made.
10. Click **Save** to save the .ipd file.
11. If you want to make the file listed in the **Default Parameters File Name** box the default file, click **Set as Default**. This copies any inspection planner settings for plan features contained in that file and sets the equivalent default entries to those settings.
12. Click **Close**.

Updating Inspection Plans with the Change Manager

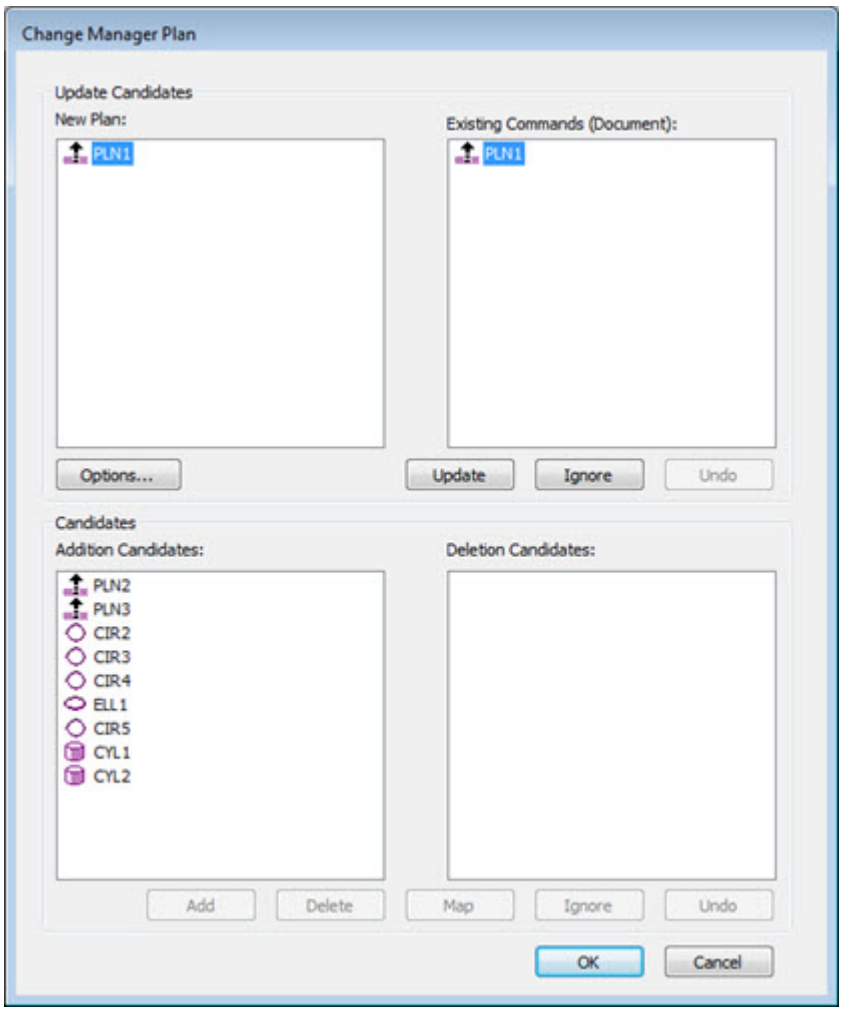
While this documentation specifically addresses Inspection Plans, it also applies to updating measurement routine files created from Datalog files as well.

The Change Manager Plan updates a measurement routine you created from a previously-imported version of your plan with the latest version of your plan. The Change Manager CAD compares the measurement routine's feature values with those in the CAD. You can then update the measurement values based on the CAD values if necessary.

The number of decimal places that Change Manager uses follows the current measurement routine's units. If the routine is in mm, Change Manager uses three decimal places. If it is in inches, Change Manager uses four decimal places.

Updating a Measurement Routine from an Inspection Plan

1. Select the **File | Change Manager | Plan** menu item to open an **Import** dialog box.
2. You can import a .planxml or .xml file created from Planner or an .ip file created from the legacy Planner.
3. Choose the plan to import, and click **Import** to convert the file and show the **Change Manager Plan** dialog box:



Change Manager Plan Dialog Box

4. You can double-click on a feature name from any list to show its parameters for that feature in order to compare values before updating your measurement routine.

5. Use the following controls to process the differences in the measurement routine and the imported plan.

    - **Options** - Opens the **Change Management Options** dialog box. See "Change Management Options".

    - **Update** - This updates the selected feature into the **Existing Commands** from the **New Plan**. If you select a feature from one list, the software highlights the corresponding feature from the other list. You can select more than one feature at a time to update. This button also updates any associated dimensions.

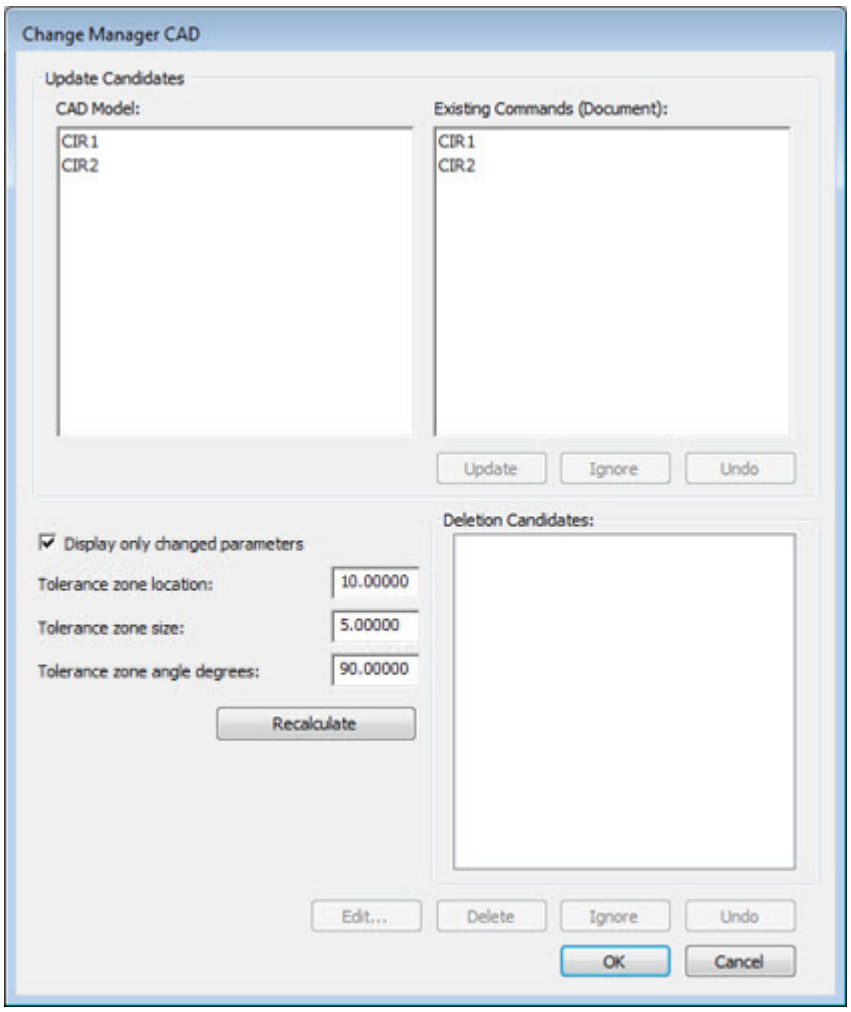
    - **Ignore** - The selected feature is ignored for update into the **Existing Commands** from the **New Plan** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to ignore.

    - **Undo** (under **Existing Commands** area) - This is available only after you use **Update** or **Ignore**. This reverts the last change made.

- **Add** - This adds the selected feature from the **Addition Candidates** list to the bottom of the existing plan. You can select more than one feature at a time to add.
- **Delete** - The selected feature from the **Deletion Candidates** list is deleted from the existing plan when you click this option. You can select more than one feature at a time to delete.
- **Map** - This is available only when you select one feature from both the **Addition** and **Deletion Candidate** lists. This replaces the selected **Deletion Candidate** feature with the selected **Addition Candidate** feature. This does not allow multiple selections. You can map both like and unlike feature types (for example, circle to a point or point to a point).
- **Ignore** - This ignores (removes) the selected feature from the **Deletion Candidates** list, but it does not delete the feature from the existing plan when you click this option. You can select more than one feature at a time to ignore. Any feature that you remove from the list remains in the measurement routine.
- **Undo** (under **Deletion Candidates** area) - This is available only after **Add**, **Delete**, or **Map** has been processed. This button reverts the last change made.

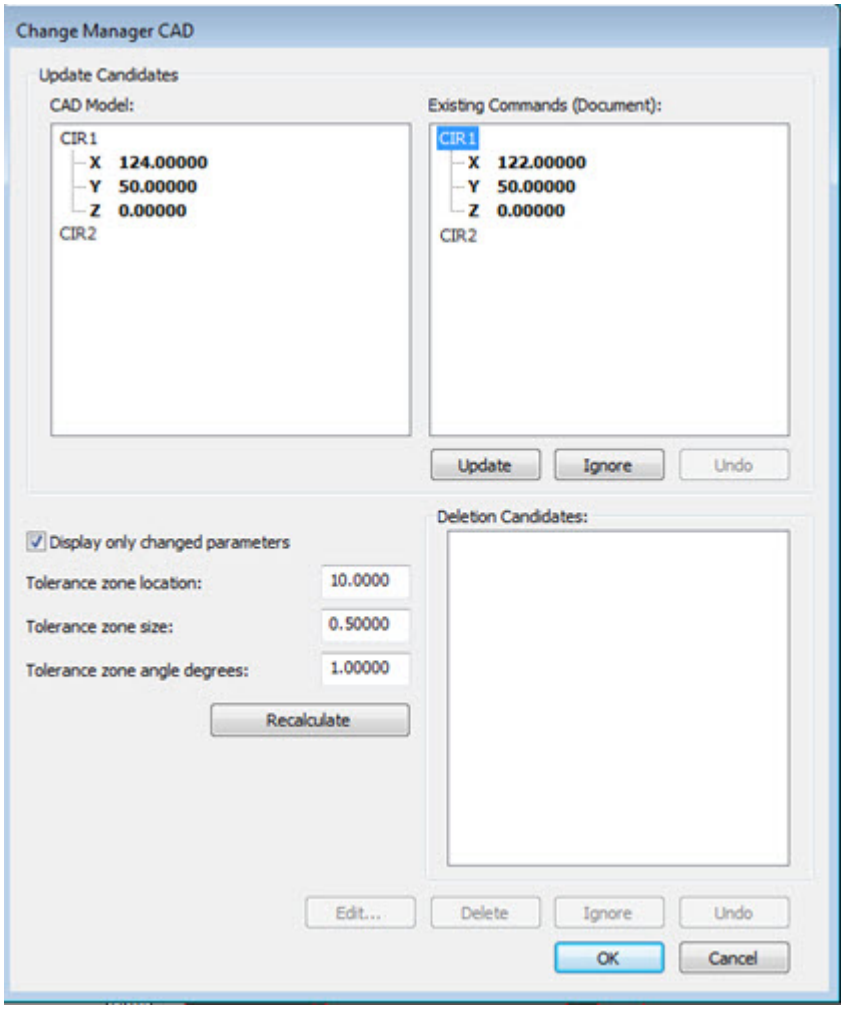
Updating a Measurement Routine from a CAD Model

1. Import a CAD model into your measurement routine (**File | Import | CAD**) or open a measurement routine that already contains a CAD model (**File | Open**).
2. Select the **File | Change Manager | CAD** menu item to open the **Change Manager CAD** dialog box:



Change Manager CAD Dialog Box

3. If the software finds differences between the CAD and command feature components, it lists the features in the dialog box. It displays the CAD features in the **CAD Model** area and the measurement routine features in the **Existing Commands (Document)** area of the dialog box. Double-click any of the listed features to display its components.



Change Manager CAD dialog box showing feature component values

4. Click the **Update** button to change the measurement routine's feature component values to those of the CAD's. This button also updates any associated dimensions.

   The **Change Manager CAD** dialog box functions the same as the **Change Manager Plan** dialog box, except for the following items:

   - The **Add** button and the **Addition Candidates** section are not available.
   - The **Edit** button for the **Deletion Candidates** section displays the edit dialog box for the selected feature in the **Deletion Candidates** list so that you can edit the feature instead of just deleting or ignoring it.
     - Select the **IPD** file to apply to the imported Inspection Plan if you have not previously selected it. See "Legacy PC-DMIS Planner Parameters and Rules" for more information.
     - If the insertion point is not at the end of the measurement routine, the software prompts you to move the cursor to the end of the measurement routine before continuing.

- Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See "Optimizing the Path" for more information. Review the optimization and close the **Optimize Path Workflow** dialog box.
- Specify the options for "Inserting Clearance Moves Automatically" and click **OK** to complete this process, or click **Cancel** to skip this step.

- If the tolerances have changed, click the **Recalculate** button to recalculate using the new tolerances. PC-DMIS then reviews the settings to determine which features are close enough to the CAD model to be considered the same.

The **Change Management Options** functions described for the **Options** button are integrated into the **Change Manager CAD** form.

Centering and Highlighting Features

You can also use the **Change Manager CAD** dialog box to **Center** a feature or **Highlight** and **UnHighlight** features you need to change or delete. This can help you to better see where changes in a CAD model have been made or how you should update your measurement routine to match a new CAD model.

To access this functionality, from the **Change Manager CAD** dialog box (**File | Change Manager | CAD**), right-click on any feature name in any of the lists (CAD Model, Existing Commands or Deletion Candidates) to display the options.

The options perform the following functions:

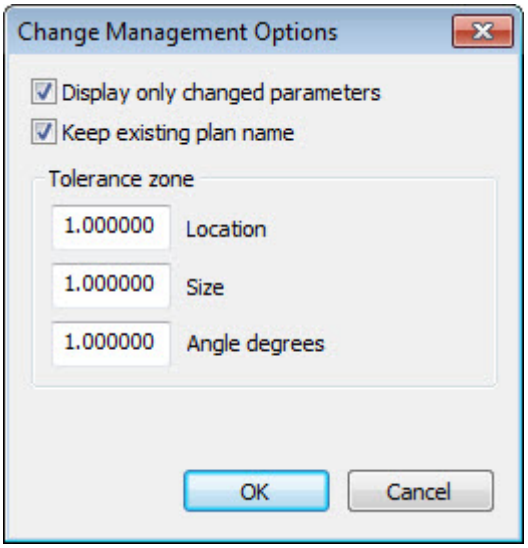
- **Center in CAD** - This centers the selected feature location in the CAD Window.
- **Highlight in CAD** - This highlights the selected feature in the CAD window. You can only highlight one feature at a time, and you cannot highlight any feature in the CAD Model list if the feature does not currently exist in the measurement routine.
- **UnHighlight in CAD** - This clears the highlight on the feature in the CAD window.

Reloading a CAD File

If the software doesn't automatically detect a modified CAD model and reload it in the Change Manager (which can happen if the original native CAD file is moved from its known location), you can use **File | Change Manager | Reload CAD** to manually tell the Change Manager to reload the CAD model.

PC-DMIS uses entries to determine the settings for the imported features. You can use the Measurement Strategy Editor or the **Auto Feature** dialog box to modify these settings.

Change Management Options



Change Management Options Dialog Box

To access this dialog box:

1. Select the **File | Change Manager | Plan** menu item.
2. From the inspection plan **Import** dialog box, navigate to the inspection plan file to import, and click **Import**.
3. Click **Options** to display the **Change Management Options** dialog box.

The following options are available:

Display Only Changed Parameters - This displays only the changed parameters for expanded features in the lists of the **Update Candidate** area.

Keep existing plan name - This retains feature names in the existing plan. Otherwise the software replaces the feature name with the feature name from the imported plan.

Tolerance Zone - You can use this area to define the tolerance values the software should use to evaluate the features. The software shows the features that are within tolerance in the **New Plan** and **Existing Plan (Program)** lists. The software evaluates the following feature dimensions:

- **Location** - Provides the tolerance for *location* evaluation.
- **Size** - Provides the tolerance for *size* evaluation.

- **Angle Degrees** - Provides the tolerance for *angle* evaluation.

Optimizing the Path

Once you have created a measurement routine (or imported one from an Inspection Plan), the Edit window contains commands to measure features on your part. The software originally adds the features in the order that you select them. However, this may not be the most efficient order to use.

With the **Optimize Path Workflow** dialog box (**Edit | Optimize Path**), you can have PC-DMIS optimize the order in which it measures the features in your measurement routine. PC-DMIS can also recommend the ideal tip angles for the different features on the part.

Optimize Path can provide significant savings in execution time.

Optimize Path works only with routines that use touch probes. It does not work with routines that use vision or laser devices.

You can access the **Optimize Path Workflow** dialog box at any time to perform path optimizations on all or part of your measurement routine. This dialog box optimizes the order in which PC-DMIS measures the features and what tip angles it uses and when.

For information on how to use this functionality, see these topics:

- Path Optimization Procedure
- The Optimize Path Workflow Dialog Box

Path Optimization Procedure

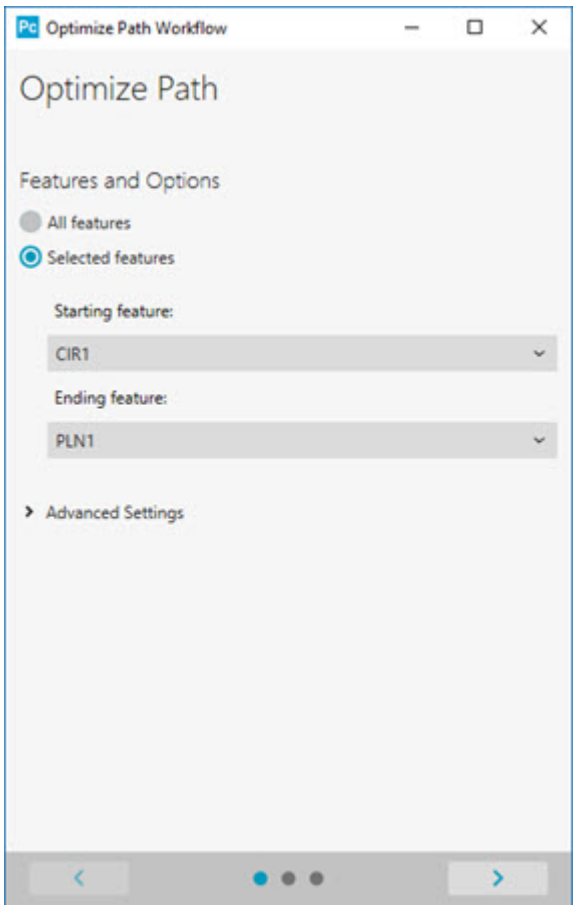
This topic shows the procedure to optimize the path with the **Optimize Path Workflow** dialog box. It uses the default options.

This dialog box uses three panels. The dots at the bottom indicate the panels and which panel you're on. From left to right, these dots indicate panels 1, 2, and 3.

Step 1 - Set Options (Panel 1)

Panel 1 shows options you can select, as well as advanced settings.

1. Select **Edit | Optimize Path**, or click **Path Optimizer** ( ) from the **QuickMeasure** toolbar. This opens the **Optimize Path Workflow** dialog box and changes the Edit window to show Command Mode. For information on the dialog box, see "The Optimize Path Workflow Dialog Box".



Optimize Path Workflow dialog box (Panel 1)

2. On panel 1, from the **Features and Options** area, decide to optimize the path for all the features (**All features**) or a subset range of features (**Selected features**).

   A. If you choose **Selected features**, from the **Starting feature** list, choose a start feature. You can also choose the feature directly from the Graphic Display window.

   B. If you choose **Selected features**, from the **Ending feature** list, choose an end feature. You can also choose the feature directly from the Graphic Display window.

> You can also open the dialog box with the starting and ending features already selected if you choose them first from the Edit window.
>
> 1. From the Edit window, select the first feature in the range.
> 2. Press Shift.
> 3. Select the last feature in the range.
> 4. Select **Edit | Optimize Path**.

3. If you need to do so, modify items under **Advanced Settings**.

4. Click the next button ( ) to move to panel 2 in the dialog box. This initiates the path optimization process and shows a progress bar.

> The path optimization makes permanent changes to your measurement routine. Make sure you're satisfied with the options in the first panel before you start the process on panel 2.
>
> *Once the optimization process finishes, the only way to revert it is to choose* **File | Quit** *and close the modified routine without saving it.*
>
> The previous button ( ) only remains active until the optimization process finishes. Once the process finishes, you cannot go back to the first panel.

Step 2 - View the Results (Panel 2)

Panel 2 shows how much the path improved.

1. On panel 2, review the circle graph and the **Shorter** value.

2. Click the next button ( ) to move to panel 3.

Step 3 - Cleanup Uncalibrated or Missing Tips (Panel 3)

Panel 3 shows the number of features without tips and the number of uncalibrated tips.

If you have features without tips:

1. From the **Features without tips** list, click on each feature to see that feature in the Graphic Display window and to see an optimum tip angle.

2. Use the area above the **Features without tips** list (the area with the **Use only calibrated** check box), and define a tip for the feature. This area behaves the same as the **Select Tip** toolbar in the "Using Toolbars" chapter.

3. Click **Apply** to evaluate the new tip against the remaining features in the list. If this new probe tip can measure any of these features, PC-DMIS assigns it to that feature and then removes the feature from the **Features without tips** list.

4. If you want to re-optimize the path based on the new tips, mark the **Perform routing** check box and click **OK**.

5. Once you click **OK**, the **Optimize Path Summary** dialog box closes, and the **Auto Insert Moves** dialog box opens. Use this dialog box to automatically insert `MOVE/`

`POINT` commands between features. For more information, see "Inserting Clearance Moves Automatically" in the "Inserting Move Commands" chapter.

If you have uncalibrated tips:

1. From the **Uncalibrated Tips** area, click on a tip and click **Calibrate tip** to open up the **Probe Utilities** dialog box.
2. From the **Probe Utilities** dialog box, click **Measure** to begin the calibration. For information on how to use the **Probe Utilities** dialog box to calibrate the tip, see "Measure" in the "Defining Hardware" chapter.

When you're done, click finish ( ) to close the **Optimize Path Workflow** dialog box.

Step 4 - Apply Clearance Moves

If the **Auto Insert Moves** dialog box opens, you can use this dialog box to insert moves between features. For information, see the "Inserting Clearance Moves Automatically" topic in the "Inserting Move Commands" chapter.

Step 5 - Check for Collisions

It's always a good idea to check for collisions whenever your measurement routine changes. For more information, see "Detecting Collisions" in the "Editing the CAD Display" chapter.

During Optimization

During path optimization, PC-DMIS accounts for machine volume as the optimization makes the following default modifications:

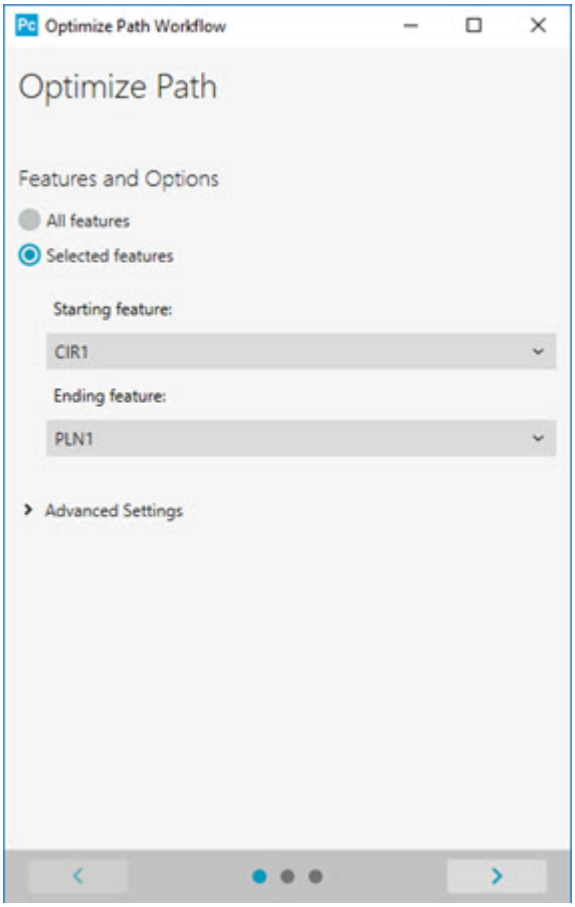
- PC-DMIS sorts the features by the tips used to measure them. However, PC-DMIS does not sort across `ALIGNMENT` or `LOADPROBE` commands.
- PC-DMIS also sorts features inside of groups.
- PC-DMIS removes all `MOVE/POINT` commands, but preserves any motion commands or other parameters that may affect measurement, such as retract distance, prehit distance, and so on.
- PC-DMIS determines the best path for the probe to take based on distances between features and the tip change.
- PC-DMIS adds dimensions at the end of the measurement routine.
- PC-DMIS adds constructed features as soon as possible.
- PC-DMIS ensures that features that have the RMeas option are not measured prior to their related features.

- PC-DMIS then determines the best tip angle for each feature. For example, the optimal tip angle for a cylinder is based on the cylinder vector, which may or may not correspond to an exact tip angle.
- If PC-DMIS cannot find an appropriate tip, it assigns a tip of T?A?B? to that feature in the measurement routine. It then adds this feature to the **Features without tips** in panel 3.

The items under **Advanced Settings** in panel 1 can affect the above path optimization process and procedure. For information on these settings and their effects, see "Advanced Settings" in "The Optimize Path Workflow Dialog Box - Panel 1".

The Optimize Path Workflow Dialog Box

To access the **Optimize Path Workflow** dialog box, choose **Edit | Optimize Path**.



Optimize Path Workflow Dialog Box

With this dialog box, PC-DMIS can reorder the measurement routine's commands to create the most efficient path for the probe to take. At the end of this process, PC-DMIS can also recommend optimal tip angles for the features in your routine. For the procedure that goes with this dialog box, see "Path Optimization Procedure".

This dialog box uses three panels. The dots at the bottom indicate the panels and which panel you're on. From left to right, these dots indicate panels 1, 2, and 3.

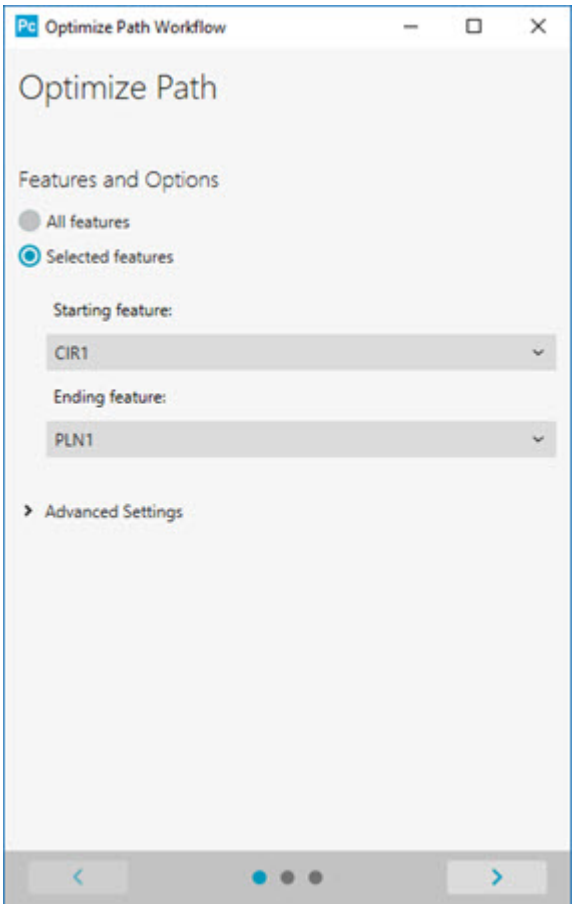
When the dialog box opens, it shows panel 1.

The topics for the panels show all the items on the dialog box.

When you access this dialog box for the first time, PC-DMIS positions it on top of the Edit Window. You can then move or size the dialog box. When you next open the dialog box, the software remembers its latest position and size.

The Optimize Path Workflow Dialog Box - Panel 1

Panel 1 contains the options for the path optimization. In this panel, you can decide to optimize the path for all the features or just a subset of features based on a selected range of features.



Optimize Path Workflow Dialog Box (Panel 1)

Features and Options area - With this area, you can decide what features receive the path optimization:

All features - This option makes the path optimization occur for all the features in the measurement routine.

Selected features - The option makes the path optimization occur for only a range of features in the measurement routine.

Starting feature - This is the first feature in a range of features. You can also choose the feature directly from the Graphic Display window.

Ending feature - This is the last feature in a range of features. You can also choose the feature directly from the Graphic Display window.

Advanced Settings

These values are persistent. This means if you change these values, the next time you open the dialog box, the software pulls in the values for these items from the JSON file.

Tip Changes area - With this area, you can decide if and how PC-DMIS adds tip changes during optimization.

Add tip changes - This switch determines whether PC-DMIS automatically inserts tip changes during the optimization. If you turn this switch off, the **Use only defined tips** and **Ask if tip not found** switches are disabled.

Use only defined tips - This switch determines whether PC-DMIS only uses defined tips to measure features. If PC-DMIS cannot find an appropriate tip then it assigns a tip of T?A?B? to that feature in the measurement routine. It then adds this feature to the **Features without tips** in panel 3 of the dialog box after the optimize path process finishes (for more information, see "The Optimize Path Workflow Dialog Box - Panel 3"). If you turn off this switch, PC-DMIS uses undefined tips. Prior to execution, PC-DMIS requires that you calibrate any undefined tips.

Ask if tip not found - This switch determines whether PC-DMIS prompts the operator if it cannot determine the optimal tip for a given feature. This lets the operator manually specify the most appropriate tip.

Filter tips - This switch filters tips so that PC-DMIS considers only those tips with angles that are within the **Filter angle** value. The **Filter angle** value defines a maximum range for considered tip angles. If you don't filter the tips, PC-DMIS considers all available tips.

Dimensions area - With this area, you can decide when to insert dimensions from your inspection plan into the measurement routine.

Insert at end - This option inserts dimensions at the end of all the measured features.

Insert as soon as possible - This option inserts dimensions as soon as possible after each measured feature.

Sorting features and groups area - With this area, you can decide how PC-DMIS handles sorting features and sorting features in groups.

Reorder features - This switch determines whether PC-DMIS sorts the features during the optimization. This enables the following two options.

If you turned on **All features** under **Features and Options**, then this sorts all features. If you chose **Selected features** then PC-DMIS only sorts that selected range of features. The default value for this switch is on.

Reorder features inside of groups - This switch determines whether PC-DMIS sorts features inside of `GROUP` commands during the optimization. If you don't choose to sort inside groups, the software doesn't sort features in groups. Instead, it only sorts the groups themselves as if they were a single feature. The default value for this switch is off.

Sort Inside Groups - This linked text opens a **Sort Inside Groups** dialog box that contains all the `GROUP` commands in the measurement routine. You need to mark the check box next to each group you want to sort. The software sorts inside groups with the inner-most group first. It handles each group as a single feature to its parent group.

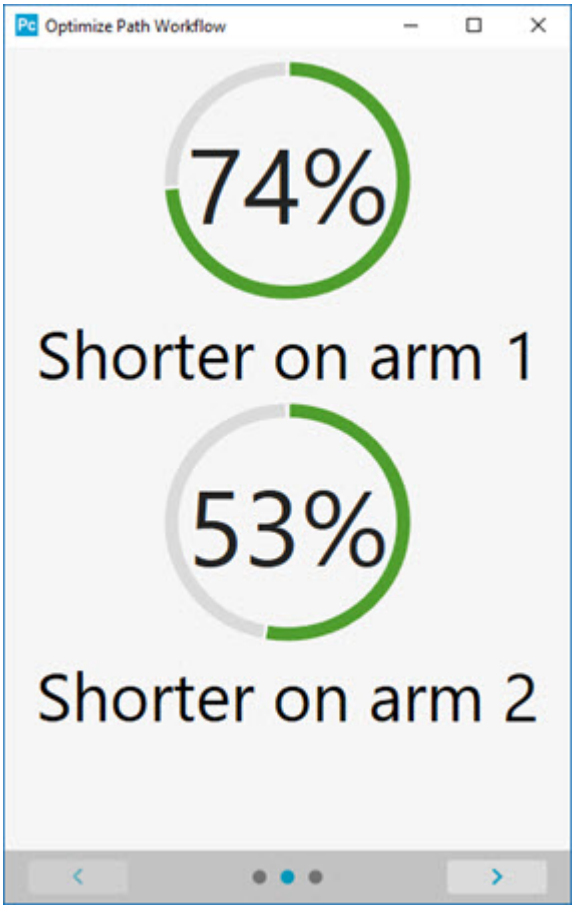
The Optimize Path Workflow Dialog Box - Panel 2

Panel 2 of the **Optimize Path Workflow** dialog box shows the result of the path optimization. It shows a percentage value. The path is that percentage shorter after the optimization.

PC-DMIS calculates the total path length before and after optimization to get this value.

Panel 2 showing the optimization summary for a single-arm machine.

If you use multiple CMM arms to measure the part, it shows the percentage shorter for each arm:



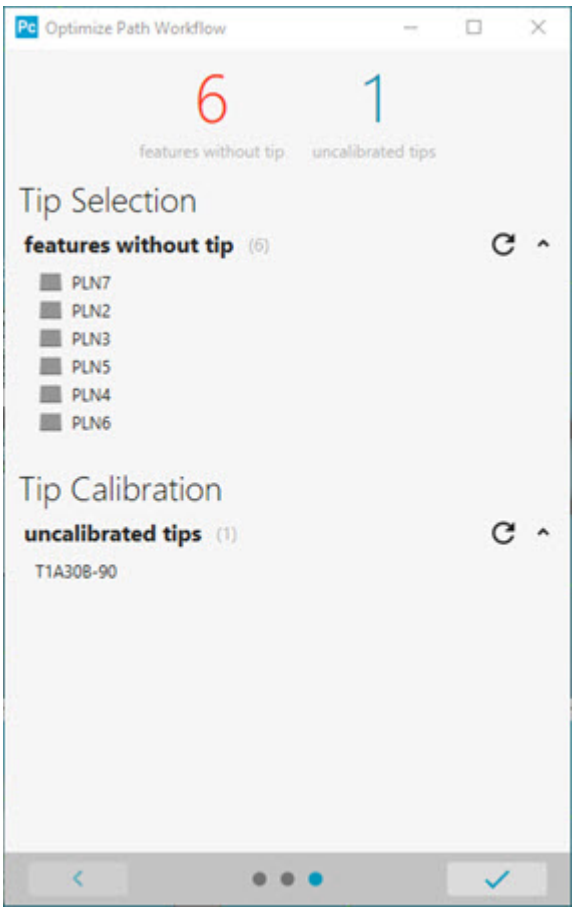
Panel 2 showing the optimization summary for a dual-arm machine.

- The previous button shows panel 1. However it only remains active until the optimization process finishes. Once the process finishes, you cannot go back to the first panel, and you cannot undo the changes the process makes.

- The next button shows panel 3 where you can work with uncalibrated or missing tips.

The Optimize Path Workflow Dialog Box - Panel 3

Panel 3 of the **Optimize Path Workflow** dialog box shows features without tips and uncalibrated tips. You can use this panel to resolve those items.

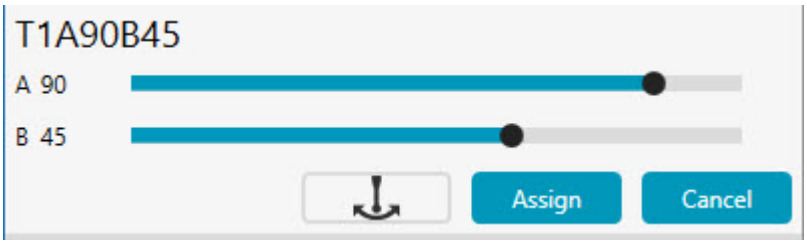


With this panel, you can do the following:

- Define probe tips for features without tips
- Calibrate uncalibrated probe tips

Features without tip list - This option shows the list of features that need a tip command. The number in parentheses shows the number of items in the list. It corresponds to the red number at the top of the panel.

You can click on a feature in this list to get a recommended tip. A mini window slides into view. This window initially suggests the best tip and angles to use to measure that feature:



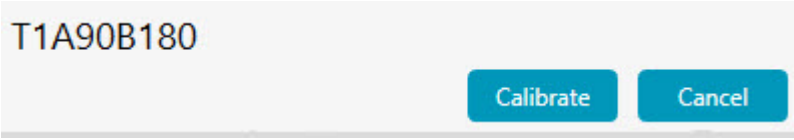
A and **B** sliders show the current angle. You can also use them to modify the A and B angles for the tip. A **C** slider may also appear if your probe supports C angles.) You may want to use these sliders to tweak the angles. The animated probe shows the current orientation in the Graphic Display window.

Find Optimum Tip Angles - This sets the slider bars back to the optimal tip angles for that feature.

Assign - Assigns the final tip angle to the feature and closes the mini window.

Tip Calibration list - This shows the list tips that need calibration. The number in parentheses shows the number of items in the list. It corresponds to the blue number at the top of the panel.

You can click on a tip to calibrate it. A mini window slides into view. This window contains these items:



Calibrate - This closes the mini window and opens the **Probe Utilities** dialog box so that you can calibrate the probe tip. You can use the **Measure** button on the **Probe Utilities** dialog box to calibrate the probe. For more information, see "Calibrating Probe Tips" in the PC-DMIS CMM documentation.

Attempt to find valid tip - This attempts to find a valid tip for the feature.

- The small triangle buttons turn blue when you hover your pointer over them. They collapse or expand the lists.

- Closes the **Optimize Path Workflow** dialog box and opens the **Auto Insert Moves** dialog box to automatically insert clearance moves between features.

Void Detection

Void Detection prevents hits from being taken in empty spaces (such as holes). This functionality has been added to these Auto Features:

- Auto Surface Point
- Auto Edge Point
- Auto Line
- Auto Plane
- Auto Circle
- Auto Round Slot
- Auto Cylinder

- Auto Cone

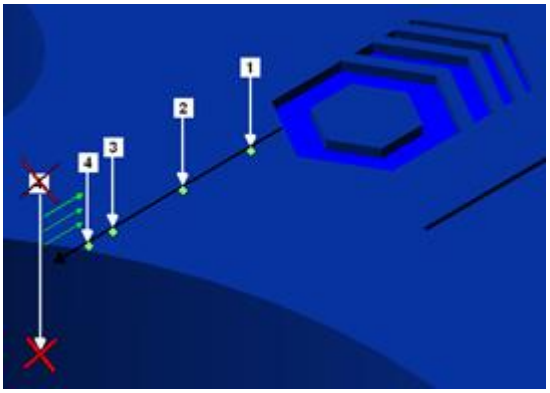
Without Void Detection, an Auto feature may place hits in holes and other empty spaces in the CAD surface and the part.

PC-DMIS automatically applies void detection to the generation of the probe hits when you create a supported Auto feature by using the parameters from the corresponding **Auto Feature** dialog box (**Insert | Feature | Auto**) such as **# hits**, **Start Angle**, and **End Angle** for an auto circle. Any probe hits that would normally fall into a void are moved so that they reside on the surface of the selected feature.

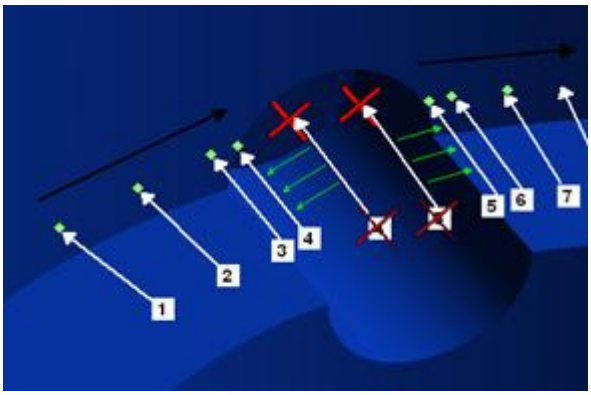
- For an Auto Plane feature, you can determine this distance, termed a "boundary offset", from any edge. This boundary offset is specified inside the **Void detection** area of the **Auto Move Properties** tab of the Probe Toolbox.
- For the other supported Auto features, this distance is the tip's radius value.

Requested sample hits for Auto Circles or Auto Cylinders are also automatically generated with Void Detection.

The following examples show when Void Detection would be useful for accurately and safely measuring an Auto feature. Hits denoted with an "X" are determined to be voids and are corrected to the edge of the Auto feature surface.



Example showing the void detection for an Auto Plane feature.



Example showing the void detection for an Auto Cylinder feature.

Void Detection can be turned On or Off using the **Void Detection** toggle on the dialog box. For details on the toggle, see the "Void Detection Toggle" topic in the "Creating Auto Features" chapter.

Inserting Clearance Moves Automatically

Once you import or create features and perform a path optimization, you need to ensure that your probe won't collide with your part during measurement. PC-DMIS contains menu options in the **Operation | Graphic Display Window | Clearance Moves** submenu that insert clearance moves automatically as `MOVE/POINT` commands to help avoid unintentional collisions with the part. These menu items under the **Clearance Moves** submenu are discussed below.

For more information on how to detect collisions, see "Detecting Collisions" in the "Editing the CAD Display" chapter.

More:

Clearance Moves Menu Items

Using Auto Insert Moves

Related Topics:

Inserting Move Commands

Detecting Collisions

Using a Wrist Device

Using a Wrist Device: Introduction

PC-DMIS provides complete support for calibrating and using infinitely indexable wrist devices such as the Renishaw PHS, the DEA CW43, and the DEA CW43L as well as devices provided by other manufacturers.

The main topics in this chapter include:

- Wrist Installation
- Tip Calibration
- Calibrate the Unit
- Qualification Check
- Home the Unit
- Using the Wrist in a Measurement Routine
- Calibrating and Using a CJoint Device

Wrist Installation

If a wrist is installed on your CMM, PC-DMIS adds A and B axes to the Probe Readouts window where the X, Y, and Z values are normally displayed. The wrist option must be enabled in the PC-DMIS license for wrist support to be available. In addition, you may need to set certain entry values. Because of the variety of infinite wrists and interfaces supported, you should consult with your software vendor for specifics on what PC-DMIS entries need to be changed (see the "Modifying Setting Entries" appendix for information on using the PC-DMIS Settings Editor to modify entries).

PC-DMIS automatically queries the controller and determines that the wrist is present.

Notes for Renishaw PHS with Leitz Interface

When you are using the kinematics mount for the Renishaw wrist, you need to set the `RenishawKinematicMount` entry to 1 in the **Option** section of the PC-DMIS Settings Editor. For information on modifying entries, please view the "Modifying Setting Entries" appendix.

When the system starts, PC-DMIS asks if the PHS Wrist is mounted on the CMM. PC-DMIS only asks this question when the controller has just been powered on. Once you inform PC-DMIS what is on the end of the arm, it does not ask the question again until it senses the controller has been shut down and restarted. When you have the wrist mounted, PC-DMIS adds A and B axes to the Probe Readouts window. This is in addition to the X, Y, Z axis readouts that are normally displayed.

The Probe Readouts window does not reflect the value changes until you run PC-DMIS after click **Yes** to the question regarding the PHS Wrist.

Tip Calibration

Tip calibration assumes that you have already calibrated the wrist. It does not need to be done for the tip that is used to calibrate the wrist. Tip calibration is done automatically for the tip used to calibrate the wrist.

The purpose of tip calibration is to calculate the distance from the last joint's center point (A joint) to the tip center. Theoretically, measuring one tip AB combination is sufficient for PC-DMIS to compute this distance once the wrist is calibrated. However, to get better accuracy, it's a good idea to measure more than one AB combination so that PC-DMIS can average the computed tip offsets.

New Probe File

Once you have calibrated the wrist, you can change the tip attached to the wrist and do a tip calibration.

To calibrate a new tip on the wrist:

1. Access the **Probe Utilities** dialog box by selecting the **Insert | Hardware Definition | Probe** menu option.
2. Make sure the probe description matches the new tip that you have loaded.
3. Select one or more AB combinations from the **Active Tip** list that correspond to this new tip. If a desired AB combination is not in the list, you can add it by selecting the **Add Angles** button. You must select at least one AB combination from the tip list in order to calibrate the tip. If you select more than one, PC-DMIS averages the results to get a more accurate tip offset.
4. Once you select the desired tips, click the **Measure** button. The **Measure Probe** dialog box appears for you to set the number of hits; the prehit and retract values; and the speeds.
5. From the **Options to Calibrate** area, select the **Calibrate Tips** option.

6. Set any other desired parameters. For more information on the **Measure Probe** dialog box, see the "Measure" topic.

7. Click the **Measure** button.

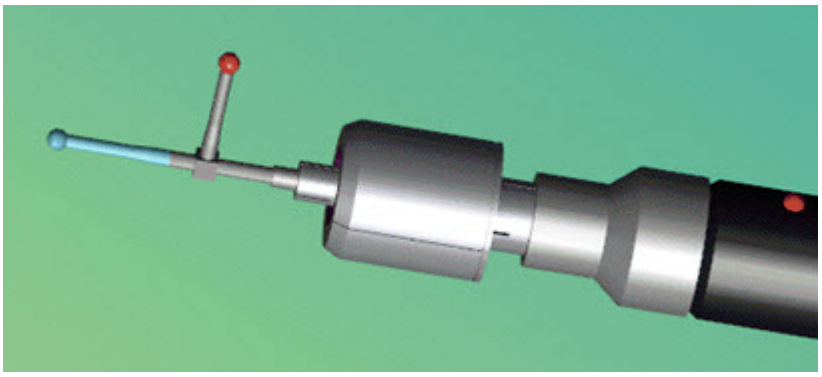
PC-DMIS begins to measure the sphere with the selected AB angle combinations.

Calibrating a "L"-Shape Star Probe with a Wrist Map

If you need to calibrate a star probe that uses a wrist map, you must perform a specific sequence of operations. You must also pay particular attention when you set the wrist rotation angles. Consider this example which shows how to calibrate a typical "L"-shape star probe configuration with a wrist map:

Overview

A star probe is not always composed of all five tips. Star probes often have only two tips. One tip connects along the central axis of the wrist. The second tip connects to an empty connection point to form an "L"-shape configuration like this:

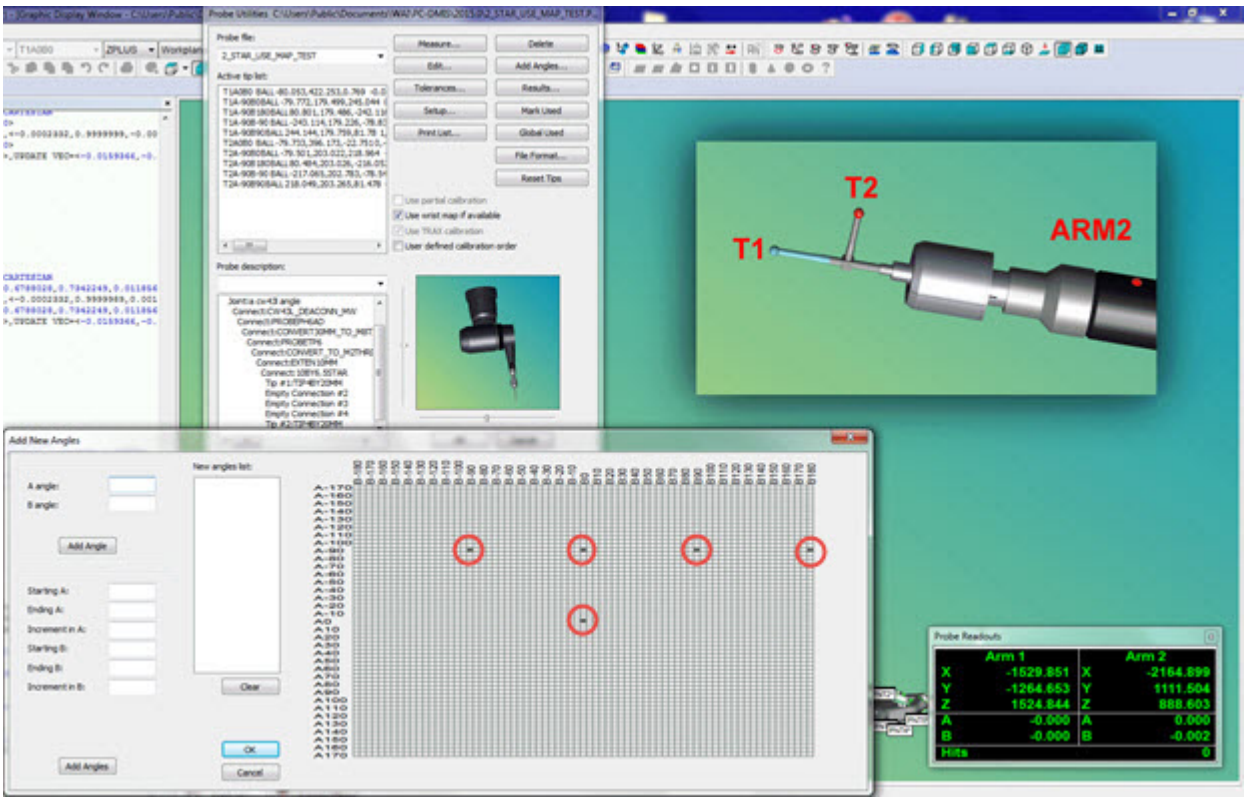


Example of an "L"-shaped star probe formed from two tips

To calibrate the star probe, you must define at least five wrist rotations (AB angles) and then measure the calibration tool with those rotations.

The wrist must already be mapped. Also, from the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), you must mark the **Use wrist map if available** check box. This ensures that PC-DMIS uses the wrist map and takes a sufficient number of measurements on the calibration tool to calibrate the star probe.

Define Wrist Five Rotation Angles



Example of a star probe defined with five wrist rotations (AB angles)

- On the ARM2 arm, the wrist holds a probe that has two tips: T1 (the central tip) and T2 (the side tip). These form an "L" configuration.
- When the wrist rotates to A = 0 and B = 0, T1 has a Y- direction, and T2 has Z+ direction.

You already start with one AB wrist rotation on your current tip. To get the minimum of five wrist rotations, you need to add four more wrist rotations that to later measure the calibration sphere. This provides the best distribution of points during the calibration process.

When you define the four rotations, you can unintentionally define rotation values (AB angles) that can calibrate T1, but cannot calibrate T2. To avoid this, take care to choose the most suitable values for the direction of the T2 tip.

You should choose rotation angles that allow T2 to always touch the calibration tool but won't collide with the tool's stem.

To accomplish this, the above example adds these wrist angles:

- A = -90, B = 0
- A = -90, B = 180
- A = -90, B = -90
- A = -90, B = 90

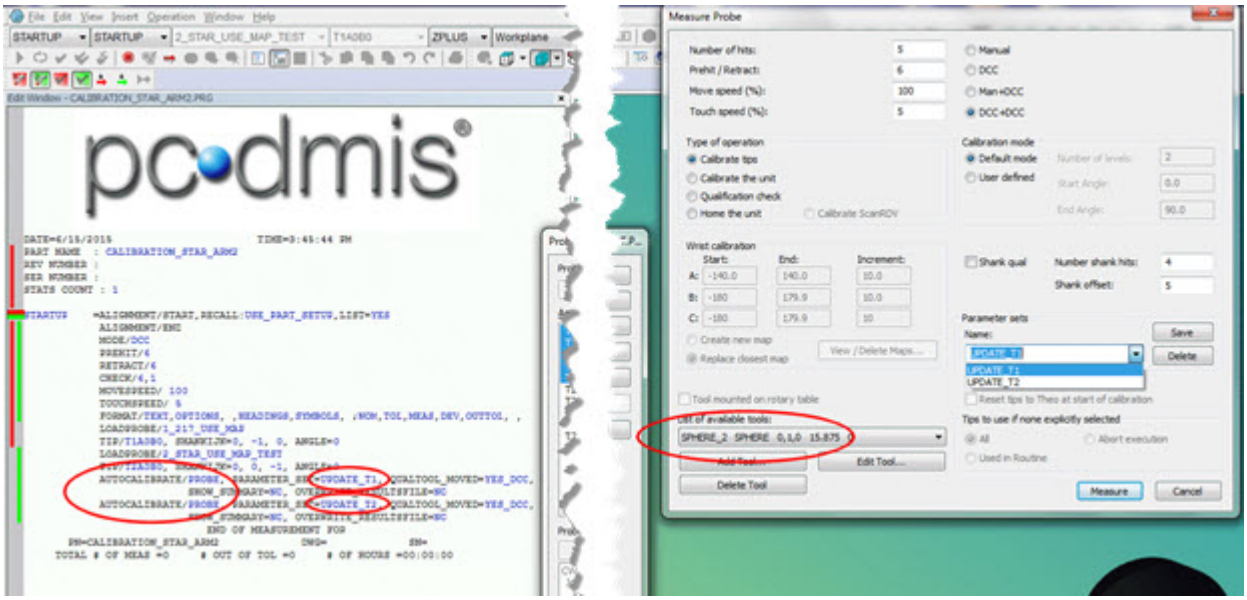
With these additional wrist angles, during calibration T2 always touches the calibration sphere, but it never collides with the calibration sphere's stem.

Once PC-DMIS detects these new tips in the **Active tip list**, it automatically creates an additional five tips. This gives you a total of ten tips: five tips with an identifier of "T1", and five tips with an identifier of "T2".

Calibrate Mapped Wrist

Once the ten wrist rotations appear in the **Active tip list**, you can calibrate them. To calibrate, you must measure the calibration sphere separately for each group of T1 and T2 tips.

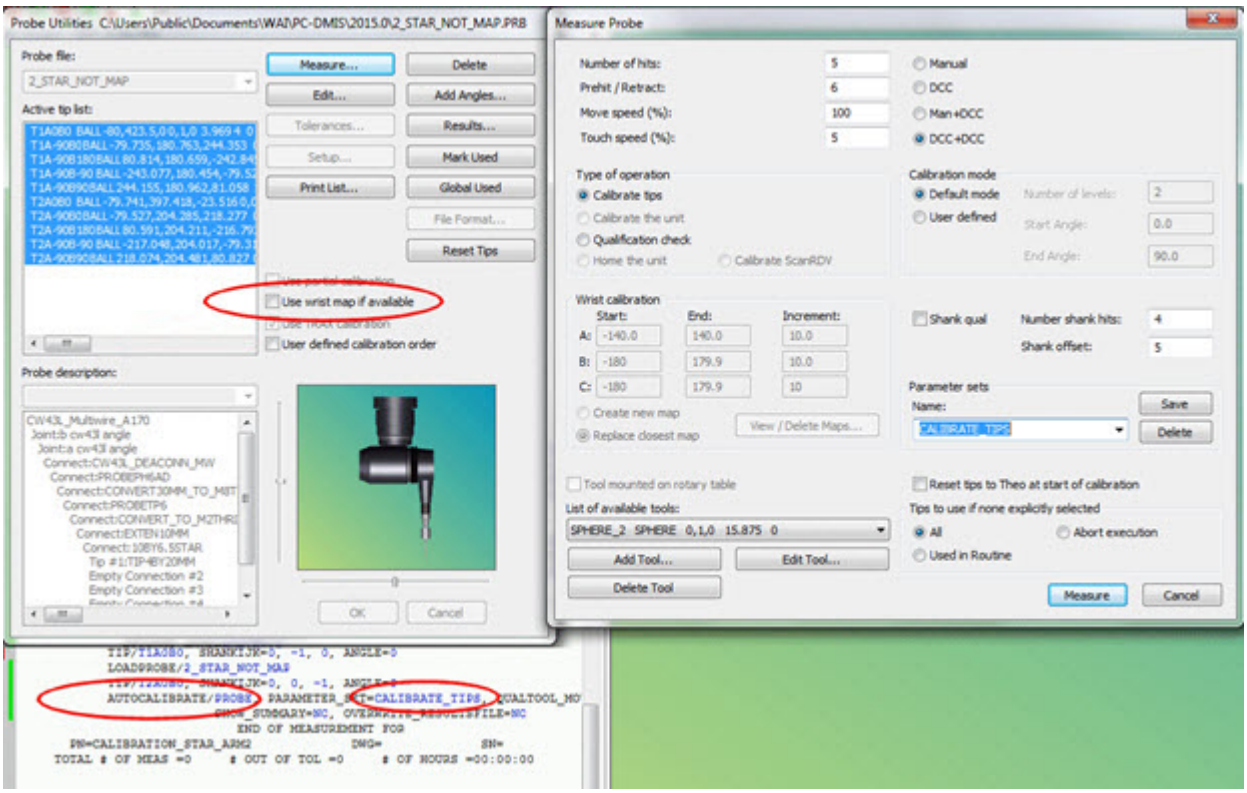
You can use the `AUTOCALIBRATE` command to invoke the parameter sets for each group. The image below shows two parameter sets (UPDATE_T1 and UPDATE_T2) that combine the respective rotations for T1 and T2:



Example of a Star Probe with parameters UPDATE_T1 and UPDATE_T2

Calibrate Non-Mapped Wrist

If the star probe was calibrated without a mapped wrist, you can calibrate the T1 and T2 tip list in the traditional method.



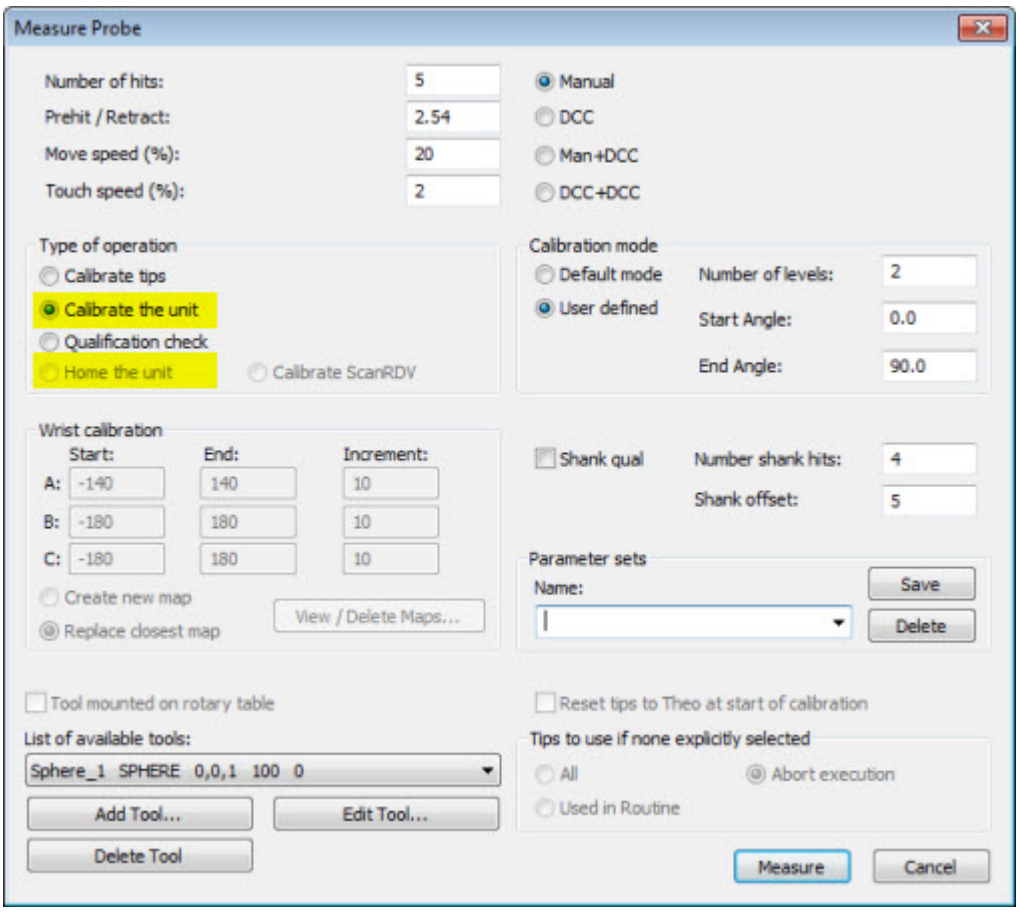
Example of a Star Probe without a wrist map

You can use a custom parameter set with the `AUTOCALIBRATE` command to measure all ten tips in the list on the calibration sphere.

The example above shows a parameter set named CALIBRATE_TIPS for this purpose.

Calibrate the Unit for Infinite Wrist Devices

When you have an indexable wrist on the CMM, PC-DMIS allows the **Calibrate the Unit** and **Home the Unit** options in the **Type of Operation** area of the **Measure Probe** dialog box.



Measure Probe dialog box showing Calibrate the unit and Home the unit option.

This wrist calibration is only done with a single stylus, not star probes. After PC-DMIS completes this calibration, you can use any angular position of the wrist in new probe files by calibrating a minimum of one probe angle. For additional information, see "Calibrate the Unit" under the "Type of Operation area" topic. See the "Calibrating Probe Tips" topic in the PC-DMIS CMM documentation for general information about calibrating probes.

The **Calibrate the Unit** option calibrates the wrist. It only works with single-arm configurations. It measures several angles on a sphere to determine the internal distances within the wrist itself. Once this information is calculated, PC-DMIS uses it to accurately predict the tip position at any A, B angle pair. This allows you to use any A, B angle in your measurement routine without calibrating each individual position.

Be sure you type your desired measurement values in the **Measure Probe** dialog box for the sphere measurement, and select the qualification tool *before* you measure with the **Calibrate the Unit** option selected. From the **Measure Probe** dialog box, you can set the number of hits, the probe prehit and retract distances, and various speeds used in the measurement process. For more information on the **Measure Probe** dialog box's options, see "Measure" in the "Defining Hardware" chapter.

Wrist Calibration

In order to calibrate the wrist, you need to measure at least three A angle positions and at least three B angle positions for a total of nine sphere measurements (each A angle position must be measured at every B position). The **Wrist calibration** area of the **Measure Probe** dialog box gives you the ability to specify the angles for calibrating both the A and the B axes. The first three options are for calibrating the A joint.

For information on using the **Wrist calibration** area to define the AB angle positions, see the "Wrist Calibration area" topic in the "PC-DMIS CMM" documentation.

When using a Renishaw PHS, every time there’s an interruption in electrical power to the PHS controller, you must either perform a wrist calibration; or in the **Measure Probe** dialog box, from the **Type of Operation** area, select **Home the Unit** and click **Measure** again.

Caution When Using SP600 Probes

For most probe types on an infinite wrist, the wrist device calibrates a zone of angles and approximates the rest. However, if you have an SP600 probe on an infinite wrist, the SP600 probe needs its own deflection matrix. You must, therefore, calibrate each AB tip angle that wasn't part of the original mapping process or suffer inaccurate results in your measurements.

Calculate Error Map

Normally, wrist devices are calibrated at relatively small increments (such as 20 degrees). When you calibrate an infinitely indexable wrist, and you select the **Calibrate the Unit** check box on the **Measure Probe** dialog box, PC-DMIS automatically creates a wrist error (wrist map) compensation file named *abcomp.dat*. It uses the file to correct for angular errors in the wrist. The error map increases the accuracy of the wrist which allows PC-DMIS to interpolate the probing offsets for probing positions not previously calibrated.

After PC-DMIS calculates the error map, it stores the results on your computer’s hard drive so that any time you use the wrist, it takes advantage of the improved angular accuracy. You should only calculate the error map periodically (once a week or less), or as necessary. When calculating the error map, PC-DMIS also performs a valid wrist and tip calibration for the currently loaded probe file.

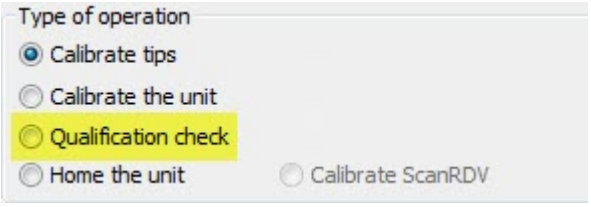
You should do a wrist calibration each time the mounting of the wrist changes. You should also refer to your hardware and vendor information for all appropriate times to map a wrist since this can change based on the construction of your device and the manufacturer's recommendations.

Once you complete the calibration of your infinitely indexable wrist device and a wrist error map file exists, you need to tell PC-DMIS to use the error map file. To do this, from the **Probe Utilities** dialog box, select the **Use Wrist Map If Available** check box. For information on this check box, see the "Use Wrist Map If Available check box" topic in the "Defining Hardware" chapter. You can then create and use any position of new probe files with a minimum number of calibrations.

To do this, create a new probe file and perform a normal tip calibration using at least one probe position on the qualification tool that maintains a link to the probe file used during the **Calibrate the Unit** process. However, you need to use more than one probe position for this calibration to provide a better "fitting" of the probe offset data to the wrist error matrix. This is especially true if you intend to utilize several wrist positions in the new probe file.

Failure to maintain a link to the probe file used during wrist mapping results in measurement errors.

Qualification Check



Type of Operation Area - Qualification Check Option

After you calibrate the unit, a qualification check is recommended, but not required. When you perform a qualification check, it provides you with information on the general accuracy of the wrist calibration and future measurements. You can also use a qualification check to check the errors of new tips added to new probe files.

Qualification Check Explained

When you perform a full calibration of your system, it establishes the relationship between the measurement system and a reference standard. For example, when you calibrate a probe tip on a reference sphere and store the probe tip's exact center.

A qualification check is a subsequent, shorter test where the same artifact (reference sphere) is measured again and the new results are compared to the stored calibration values.

- If the differences (for example, in X, Y, and Z position, or in a combined 3D distance like PR) are within the allowed limits, the probe, stylus, and machine configuration is qualified for use. The previous calibration is considered still valid.
- If the differences exceed those limits, the system is no longer qualified and a new full calibration is required.

During a full calibration of your system, PC-DMIS stores the reference center of the calibration sphere.

The qualification check measures that sphere center again to generate the deviation values in X, Y, and Z. These three deviation values define a vector (ΔX, ΔY, ΔZ). PR is the length or magnitude of that vector and is defined as:

$$PR = \sqrt{(\Delta X)^2 + (\Delta Y)^2 + (\Delta Z)^2}$$

The PR value defines how far in 3D space the current measured physical center of the reference sphere has moved from the calibrated center. It represents the overall 3D shift of the ruby tip since the last calibration. For example, the shift might be due to a bent stylus after a crash.

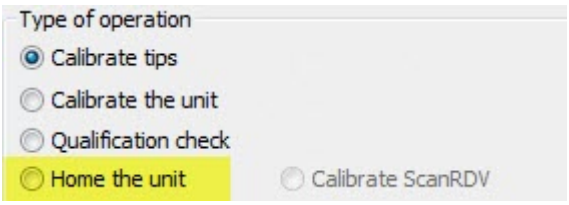
If the PR value is within the allowed limit, the probe tip is still qualified.

If the PR value exceeds the limit, your probe tip is no longer qualified and you must recalibrate it.

To perform a qualification check:

1. Access the **Probe Utilities** dialog box by selecting the **Insert | Hardware Definition | Probe** menu option.
2. On the **Probe Utilities** dialog box, from the **Active Tip** list, select the angles you wish to use for the qualification check. You should select probe angles both used and not used in the wrist calibration.
3. Click the **Measure** button to open the **Measure Probe** dialog box.
4. Provide the parameters you wish to use during the calibration check including selecting the appropriate calibration tool.
5. From the **Type of Operation** area, select the **Qualification Check** option.
6. Click the **Measure** button.
7. Follow any on-screen instructions.

Home the Unit



Type of Operation Area - Home the Unit Option

Some wrist devices, such as the Renishaw PHS, do not have predefined zero positions and use potentiometers instead of scales to position the wrist. These types of wrist devices need to have the zero redefined every time power to the probe head controller is interrupted. To redefine the wrist zero position, you may choose the **Calibrate the Unit** option (For information see the "Tip Calibration" topic discussed in this chapter) or the **Home the Unit** option.

Selecting **Home the Unit** calculates the angular error offset from the previously calculated wrist zero position by qualifying one or more probe angles on a previously calibrated sphere position. The advantage of this is you can use a minimum of one probe tip angle thereby making the process much faster than a wrist calibration.

Since PC-DMIS averages the errors of the homing procedure to provide a more accurate adjustment to the wrist error map, you use more than one probe angle to accomplish the **Home the Unit** operation.

To perform a **Home the Unit** operation, follow these steps:

1. To open the **Probe Utilities** dialog box, select the **Insert | Hardware Definition | Probe** menu option.
2. Select the same probe file that was used to calibrate the unit.
3. From the **Active Tip** list, select the angles that you wish to use for the calibration check.
4. Click the **Measure** button to open the **Measure Probe** dialog box.
5. Provide the parameters that you wish to use during the calibration check. Also select the same calibration tool used during the Calibrate the Unit operation.
6. From the **Type of Operation** area, select the **Home the Unit** option.
7. Click the **Measure** button.

After calibrating the unit and prior to homing the unit, do not move the calibration sphere. If it is moved, you must re-calibrate the unit (for information on calibrating the unit, see the "Tip Calibration" topic in this chapter).

Using the Wrist in a Measurement Routine

On machines that support it, PC-DMIS automatically senses when the wrist has been rotated with the jog box. The tip offsets are dynamically updated based on the current AB angles. That is, the XYZ readout displays the current tip position as it is being rotated. There are four ways (discussed below) to add a new AB tip combination into a measurement routine.

Once you have added a new AB tip combination into your measurement routine by using one of the options below, PC-DMIS displays the active tip as programmed in the measurement routine in **Probe Tips** list on the **Settings** toolbar (**View | Toolbars | Settings**). The selected item is the same as the `TIP/` command above where the cursor is in the Edit window.

To move the wrist to this AB angle combination, from the **Settings** toolbar, pull down and choose the probe from the **Probe Tips** list. PC-DMIS asks if you want to rotate the probe. A **Yes** response rotates the wrist to the desired position. This is useful when you want to go back and insert measurements and moves into the measurement routine.

Option 1

In the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), select the **Add Angles** option. You can use this option to key in new AB angle pairs to the probe file. If the tip is already calibrated, then the new AB combinations are also calibrated and made ready to use. Once you leave the **Probe Utilities** dialog box, you can use these new AB combinations by selecting them from the **Probe Tips** list located on the **Settings** toolbar (**View | Toolbars | Settings**).

Option 2

Manually jog the wrist to the desired AB angles using the jog box. When you reach the desired position, press the **Store Move** button on the jog box, or take a manual probing. The current AB angles are read automatically. If the A or B angles have changed by more than the PH9 **Wrist Warning Delta** value available on the **Part/Machine** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**) then PC-DMIS automatically adds the current AB angles to the **Probe Tips** list and inserts a `TIP/` command in the measurement routine. The new `TIP/` command is added to the measurement routine before the stored move or manual probing. When the measurement routine is then executed, PC-DMIS first changes the tip before moving to the programmed location.

For information on the **Wrist Warning Delta** option, see the "Automatically Adjust Probe Head Wrist" topic in the "Setting your Preferences" chapter.

Option 3

Type a `TIP/` command in the Edit window. Once you finish editing the vector, PC-DMIS calculates the best AB combinations that put the tip shank parallel to that vector.

The **Auto Wrist** icon in the **Measurement Properties** area in the **Auto Feature** dialog box is available if your measurement routine has a wrist probe. You can use this icon to enable PC-DMIS to determine the best wrist position to use when an Auto feature is measured. For more information, see "Auto Wrist".

Calibrating and Using a CJoint Device

PC-DMIS provides complete support for calibrating and using a three continuous axes wrist obtained by mounting a CJoint device on the DEA CW43L wrist.

If a CJOINT is installed on your DEA CW43L wrist, PC-DMIS adds A, B, and C axes to the X, Y, and Z values that are normally displayed in the Probe Readouts window.

The following sub topics are available to help you calibrate your CJoint device.

- Tip Calibration Information
- CJoint Calibration Procedure for Infinite Third Axis Wrist Devices
- Calculate Error Map for CJoint

CJoint Tip Calibration Information

The purpose of tip calibration is to calculate the distance from the second joint's center point (the A joint’s center) to the star probe’s tip center. You may want to measure more than one AB combination so that PC-DMIS can average the star tip offsets computed, resulting in better accuracy in reaching the sphere when the CJoint calibration cycle is performed.

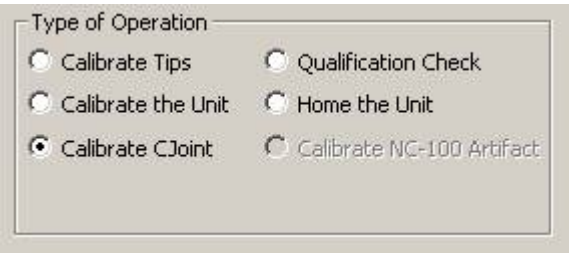
Before proceeding, verify the following:

- You should already have calibrated the AB wrist device; and the files named *acomp(s).dat* and *wrist(m)(s).dat* should exist on your computer’s hard drive.
- You need to mount a star probe on the CJoint device with the tip pointing in the X+ direction. PC-DMIS uses this tip to calibrate the CJoint. Tip calibration in this case does need to be done.

You can also obtain the AB calibration map with the CJoint mounted on the wrist. This increases the accuracy of angle position computations for any A, B, and C angle combinations.

CJoint Calibration Procedure for Infinite Third Axis Wrist Devices

The **Calibrate CJoint** option allows you to calibrate the third axis of the wrist. Once this information is calculated, PC-DMIS uses it, together with the AB calibration data, to predict the tip position at any A, B, and C angle combinations for a probe that needs to be installed on a CJoint device.



Type of Operation Area - Calibrate CJoint Option

To Calibrate using the CJoint Option

1. Ensure that you have already performed the preliminary steps discussed in the "CJoint Tip Calibration Information" topic of this chapter.
2. Select **Insert | Hardware Definition | Probe** to display the **Probe Utilities** dialog box.
3. Click the **Measure** button to open the **Measure Probe** dialog box.
4. From the **Type of Operation** area, select the **Calibrate CJoint** option. Selecting this item enables the **Wrist Calibration** area if the appropriate wrist entries from the **Option** section in the PC-DMIS Settings Editor are set to 1 (either `DEAWrist` or `RENISHAWWrist`).
5. Ensure that you have a star probe configuration connected, with a tip pointing in the X+ direction.
6. From the **Available Tools** list, select a sphere qualification tool.
7. Type the desired measurement values for the sphere qualification.
8. Set the number of hits, the probe pre-hit and retract distances, and various speeds.
9. Fill out the **Wrist Calibration** area. In order to calibrate the C joint, you need to measure at least three C angle positions. This gives you the ability to specify the angles for calibrating the A, B, and C axes. The **Start**, **Increment**, and **Angle** boxes for the **C** row are for calibrating the C joint.
    - **C Angle Start** defines the starting C angle used for the calibration of your wrist's C joint. The default value is -180.
    - **C Angle End** defines the ending C angle used for the calibration of your C joint. The default value is 180.
    - **C Angle Increment** defines the angle increment between the starting and ending C angles. Angles are calibrated for your C joint device for the starting, ending, and additional angles based on the increment value. For example, an increment angle of 10 degrees adds angles for calibration every 10 degrees between the starting and ending angle. The default value is 10.

Be sure that the wrist's **A Angle Start** and **B Angle Start** are set to 0 when you calibrate a C joint device.

10. When you are ready, click the **Measure** button to start the calibration process.

Calculate Error Map for CJoint

Normally, wrist devices are calibrated at relatively small angle increments (such as 20 degrees). When you calibrate a continuous third axis of a wrist, with the **Calibrate CJoint** option selected, PC-DMIS automatically creates a wrist error compensation file named *ccomp(s).dat* that it uses, together with *abcomp(s).dat*, to correct for angular errors in the three axes wrist.

After calculating the error map, the results are stored on your computer’s hard drive so that any time you use the wrist, it will take advantage of the improved angular accuracy.

Recalculating the Error Map

Generally, you should only need to calculate the error map periodically (once a week or less). However, you should perform a wrist calibration each time the mounting of the wrist changes and at other times recommended by hardware and vendor information. This varies based on device construction and manufacturer recommendations.

Working in Operator Mode

Working in Operator Mode: Introduction

The Operator Mode limits the options that will be available when operating PC-DMIS. Once the restrictions are in place, the operator can only open and execute the measurement routine.

To start PC-DMIS in Operator Mode, from the **Start** menu, select **All Programs | <Version> | <Version> Operator** , where <version> represents your PC-DMIS version.

When PC-DMIS launches in Operator Mode, only the options necessary to run the measurement routine are available.

The main topics in this chapter include:

- File Open Options
- Operator Mode Menu Options
- Using the Marked Sets Window in Operator Mode

File Open Options

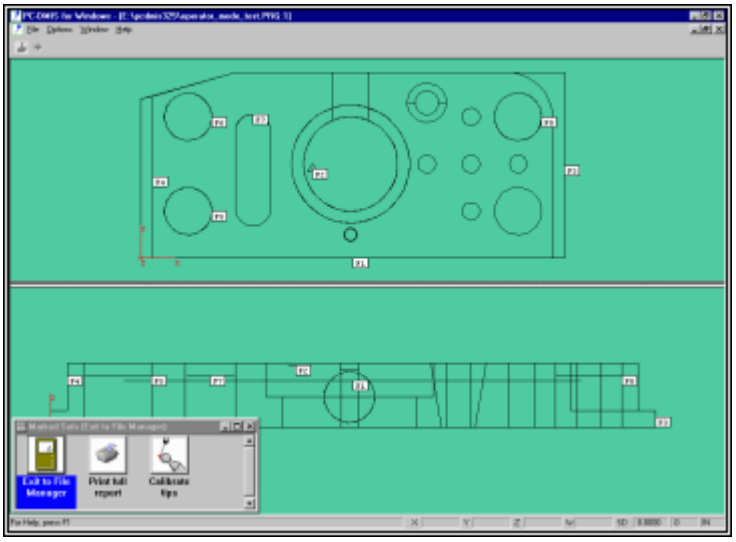
The **Open** dialog box (**File | Open**) is a standard Windows file open dialog. You can activate a measurement routine in Operator Mode by double-clicking the measurement routine file name or by selecting the measurement routine file name and clicking the **Open** button.

The following options have been *disabled* from this dialog box in Operator Mode:

- **Importing to PC-DMIS**, accessible when *not* in Operator Mode by right mouse clicking on a measurement routine and selecting Import to PC-DMIS from the pop-up list.
- **Exporting from PC-DMIS**, accessible when *not* in Operator Mode by right mouse clicking on a measurement routine and selecting Export from PC-DMIS from the pop-up list.
- **Editing the Part Name**, Serial Number, or Revision Number on the measurement routine's PC-DMIS property page, accessible when *not* in Operator Mode by right mouse clicking on a measurement routine and selecting Properties from the pop-up list.

Operator Mode Menu Options

Once a measurement routine has been activated, the main PC-DMIS screen is displayed with the following menu options.



Graphic Display window in Operator Mode

File

The **File** menu allows you to open an existing measurement routine, close the current measurement routine, export the current measurement routine, or exit the software.

Open

The **File | Open** menu option allows you to switch between measurement routines anytime by invoking the **Open** dialog box and allowing you to select a different measurement routine. If the measurement routine you select is different from the currently executing part, PC-DMIS automatically saves and closes the active measurement routine before loading the newly selected measurement routine. If however, you try to select and initiate the active measurement routine a second time from the **Open** dialog box, PC-DMIS offers two choices:

- Discard all changes made to the active measurement routine since the last Save operation and reload the selected measurement routine without the changes.
- Cancel the entire operation and return to the **Open** dialog box.

You can only have one measurement routine open at a given time in Operator Mode.

Close

Selecting **File | Close** closes the measurement routine and saves the measurement values of any executed marked sets.

Quit

Selecting **File | Quit** closes the current measurement routine *without saving* any recently taken measurement values from executed marked sets. Only pre-existing data will be retrievable.

Export

To export your current measurement routine, choose the **File | Export** menu option. This will display the **Export Data** dialog box. Select the data type to export as, and a directory location, and then click **OK**. For more information on exporting, see "Exporting CAD Data" in the "Using Advanced File Options" chapter.

Exit

To quit PC-DMIS, all other routines, and log off the computer, choose the **File | Exit** menu option. PC-DMIS automatically saves the current measurement routine before exiting.

Edit

The **Edit** menu option allows you to access the **Probe Readouts Setup** dialog box. You can use this dialog box to set up how information is displayed. See the "Setting Up the Probe Readouts Window" topic in the "Setting your Preferences" chapter for information.

View

The **View** menu allows you to show or hide the following windows while in Operator Mode:

Graphic Display window – Selecting this option shows and hides the Graphic Display window. See "The Graphic Display window" in the "Navigating the User Interface" chapter.

Preview Window – Selecting this option shows and hides the Preview window. See "Using the Preview Window" in the "Using Other Windows, Editors, and Tools" chapter.

Probe Readouts – Selecting this option shows and hides the Probe Readouts window. See "Using the Preview Window" in the "Using Other Windows, Editors, and Tools" chapter.

Marked Sets Window – Selecting this option shows and hides the Marked Sets window. See "Using the Marked Sets window" in the "Using Other Windows, Editors, and Tools" chapter. See "Using the Marked Sets Window in Operator Mode".

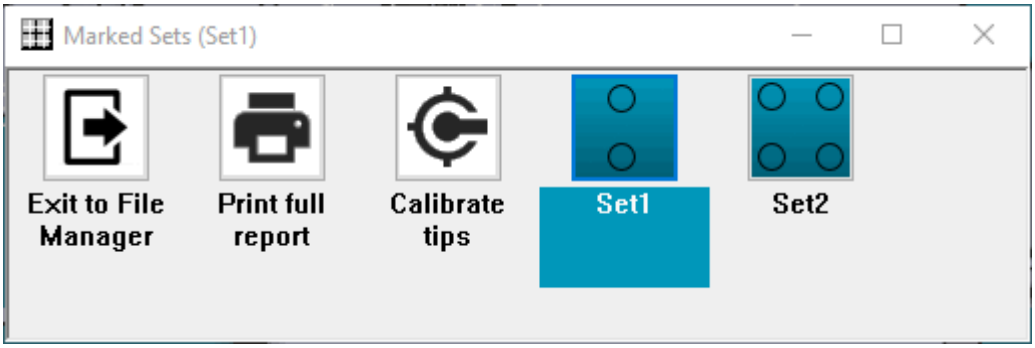
Window

The **Window** menu provides operations to manage multiple open measurement routines and windows. See the "Navigating and Displaying Multiple Windows" chapter.

Help

The **Help** menu provides all the menu items as in PC-DMIS's standard mode of operation.

Using the Marked Sets Window in Operator Mode



Marked Sets Window

As soon as you open a measurement routine, the Marked Sets window automatically launches. The following options are available to the Marked Sets window in Operator mode:

- Exit to File Manager
- Print full Report
- Calibrate Tips
- Any Predefined Marked Sets

Exit to File Manager

The **Exit to File Manager** button displays the **Open** dialog box. From the dialog box, select a measurement routine to open.

Print Full Report

The **Print full report** icon prints the complete report to the output source that was setup in the "learning" portion of the measurement routine.

If the output is sent to a file, it is saved in an RTF format. If this is the first inspection report to be saved since powering up the system, PC-DMIS requires you to enter a value in the **New File #** box. If **Auto** is checked, after the initial file is saved PC-DMIS saves all future files incrementally from the initial number that was entered (as long as the system has not been powered down). The displayed number can be changed at any time simply by entering another value.

To print the current Edit report, choose the **Print Full Report** button (or press F4). The Edit window output is sent to the printer and/or a pre-selected file.

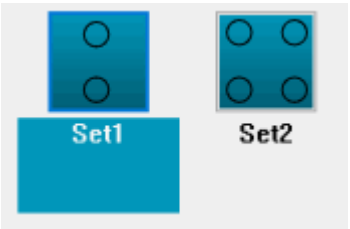
Calibrate Tips

The **Calibrate Tips** button calibrates all tip angles for all probes in the current measurement routine. This process informs PC-DMIS of the location and diameter of each probe tip.

PC-DMIS does not track the calibration of probes. Be sure to verify that a probe is re-calibrated if it is changed in any way.

Predefined Marked Sets

Predefined marked sets may appear in your Marked Sets window. These are features that have been programmed and marked for execution by the measurement routine programmer and are combined into a set for the operator to execute.



Example of predefined marked sets.

To execute and measure features in a marked set, simply click on the set from the Marked Sets window and follow any instructions provided by PC-DMIS.

Using Shortcut Keys and Shortcut Menus

Using Shortcut Keys and Shortcut Menus: Introduction

This appendix covers shortcuts to many of the menu options, dialog boxes, and commands that you can use in PC-DMIS. Using shortcuts can help speed up your programming time and increase productivity and efficiency.

The main topics covered in this appendix are:

- Shortcut Keys Reference
- Shortcut Menus Reference

Shortcut Keys Reference

This table provides a reference of the available shortcut keys in the English language. If the description of a shortcut key is preceded by *italicized text*, the item in italicized text must be the active window or focused item for the shortcut key to function properly. For specific information regarding the function of each option, see the appropriate section of the documentation.

Non-English Shortcut Keys: Since keyboards differ from language to language, some shortcut keys may not be the same for non-English builds of PC-DMIS. To see what shortcuts are associated with a non-English build of PC-DMIS, on the **Customize** dialog box, consult the **Keyboard** tab. To access the **Customize** dialog box, right-click on the toolbar area and select **Customize**. For information on using the **Keyboard** tab, see "Customizing Shortcut Keys" in the "Navigating the User Interface" chapter.

Key	Description	Key	Description	Key	Description
F1	Accesses the online Help.	Ctrl + A	*Edit window:* Selects all text. Form and Report Editors: Selects all the objects.	Ctrl + F1	Places PC-DMIS into Translate Mode.
F2	*Edit window:* Opens the **Expression Builder** dialog box if the cursor is on a line that allows expressions.	Ctrl + C	*Edit window:* Copies the selected text. Form and Report Editors: Copies the selected objects.	Ctrl + F2	*Graphic Display window:* Places PC-DMIS into 2D Rotate Mode. *Edit window:* If in Command Mode, this shortcut inserts or removes a bookmark on the current line.
F3	*Edit window:* Marks or unmarks the command for execution. If your cursor resides on an External Object, F3 switches between print and execute modes.	Ctrl + D	Deletes the current feature.	Ctrl + F3	Places PC-DMIS into 3D Rotate Mode and opens the **Rotate** dialog box.
F4	*Edit window:* Prints the contents of the Edit window.	Ctrl + E	Executes the selected feature or command (for those commands that support this shortcut).	Ctrl + F4	Places PC-DMIS into Program Mode.
F5	Accesses the **Setup Options** dialog box.	Ctrl + F	Opens the **Auto Features** dialog box.	Ctrl + F5	Places PC-DMIS into Text Box Mode.

Key	Description	Key	Description	Key	Description
F6	Accesses the **Font Setup** dialog box.	Ctrl + G	Inserts a `READPOINT /` command into the Edit window.	Ctrl + Tab	Switches between certain open windows to make the next window active. For example, the Basic Script Editor window, Graphic Display window, and the Report Window. The windows must be open for this to work.
F7	*Edit window:* Within a selected toggle field, cycles forward to the next alphabetical entry.	Ctrl + J	*Edit window:* Jumps to a referenced command.	Ctrl + Shift	Hides the selected graphical analysis arrow.
F8	*Edit window:* Within a selected toggle field, cycles backwards to the last alphabetical entry.	Ctrl + K	Saves the selected dimension in the Edit report.	Ctrl + End	*Edit window:* Moves the cursor to the end of the current measurement routine.

Key	Description	Key	Description	Key	Description
F9	*Edit window:* Opens the dialog box associated with the command at the cursor's position. (For the `MODE` command in the Edit window, this key switches between the DCC Mode and Manual Mode.)	Ctrl + L	Executes the currently selected block of commands.	Ctrl + Home	*Edit window:* Moves the cursor to the beginning of the current measurement routine.
F10	Opens the **Parameter Settings** dialog box.	Ctrl + M	Inserts a `MOVEPOINT` command into the Edit window.	Ctrl + Alt + A	Opens the **Alignment** dialog box.
F12	Opens the **Fixturing Setup** dialog box.	Ctrl + N	Creates a new measurement routine.	Ctrl + Alt + L	Creates an automatic alignment using the QuickAlign functionality.
Shift + Right-click	Opens the **Scale Drawing** dialog box.	Ctrl + O	Opens a measurement routine.	Ctrl + Alt + P	Opens the **Probe Utilities** dialog box.

Key	Description	Key	Description	Key	Description
Shift + Tab	*Edit window:* Moves the cursor backwards to the previous user-editable field.	Ctrl + P	Prints the Graphic Display window.	Ctrl + Click	In Text Box Mode, performing this action on a feature or label ID in the Graphic Display window moves your cursor to that feature in the Edit window. Performing this action with the **Analysis** dialog box open selects any associated dimensions.
Shift + Arrow	Highlights all text as the cursor moves.	Ctrl + Q	*Edit window:* Executes the current measurement routine.	Ctrl + Shift + H	*Edit window:* Highlights the selected feature in the Graphic Display window.
Shift + F5	*Edit window:* Changes a dimension's hit display between the Cartesian and polar coordinates. A 'P' character indicates polar display mode.	Ctrl + R	Opens the **Rotate** dialog box.	Ctrl + Shift + U	*Edit window:* Clears the highlight of the selected feature from the Graphic Display window.
Shift + F6	*Edit window:* If in Command Mode, opens the **Color Editor** dialog box.	Ctrl + S	Saves the current measurement routine.	Up Arrow	*Edit window:* Moves the cursor to next available element above the current position.

Key	Description	Key	Description	Key	Description
Shift + F10	*Edit window:* Accesses the **Jump To** dialog box.	Ctrl + T	*Edit window:* Assigns the current command (or selected commands) to Arm1, Arm2, or both arms.	Down Arrow	*Edit window:* Moves the cursor to next available element below the current position.
END	Ends the measurement of a feature. *Edit window:* Moves the cursor to the end of the present line.	Ctrl + U	*Edit window:* Executes the measurement routine from the cursor's location.	Right Arrow	*Edit window:* Moves the cursor to next available element to the right of the current position. In Summary Mode, this shortcut expands a collapsed list.
Home	*Edit window:* Moves the cursor to the beginning of the present line.	Ctrl + V	*Edit window:* Pastes the Clipboard's contents. Form and Report Editors: Pastes the copied objects.	Left Arrow	*Edit window:* Moves the cursor to next available element to the left of the current position. In Summary Mode, this shortcut collapses an expanded list.
Tab	*Edit window:* Moves the cursor forward to the next user-editable field.	Ctrl + X	*Edit window:* Cuts the selected text. Form and Report Editors: Cuts the selected objects.	Alt + "-" (minus)	Pressing Alt and then the minus key deletes the last hit in the hit buffer.

Key	Description	Key	Description	Key	Description
Esc	Aborts any process (other than data entry) if pressed prior to pressing the Enter key.	Ctrl + Y	*Edit window:* Resumes a previously cancelled execution.	Alt + C	Displays the **ClearanceCube** dialog box.
Delete	*Edit window:* See 'Backspace'.	Ctrl + Z	Activates the Scale to Fit function.	Alt + H	Accesses the Help menu.
Backspace	*Edit window:* Deletes any highlighted characters. If nothing is highlighted, then functions as in a normal editor. If the item cannot be deleted, an error message is displayed.	Ctrl + Enter or Return	*Edit window:* In Summary Mode, this keyboard shortcut allows you to select a command to add to the Edit window.	Alt + J	*Edit window:* Jumps back from a referenced command.
Enter or Return	*Edit window:* Creates a new blank line that is automatically deleted if not completed before the cursor is moved from the line. Selects a command.	Ctrl + Click	*Graphic Display window:* Multi-selects CAD objects (like surfaces). If a CAD object is already selected, this shortcut deselects that object.	Alt + P	*Graphic Display window:* Draws the current path of the probe for the entire measurement routine.
Shift + F4	Opens the Machine Interface Setup.	Ctrl + Left-mouse drag	Makes the dialog box or toolbar drag without docking into the interface when you release the mouse.	Alt + Shift + P	*Graphic Display window:* Draws the current path of the probe for the features immediately before and after the cursor position.

Key	Description	Key	Description	Key	Description
Shift + Right-click	*Report Window Label Object:* Displays the **Report** dialog box.	Ctrl + Right-mouse drag (Also click and drag the center wheel button)	*Graphic Display window:* Rotates the CAD model in 3D.	Alt + F3	*Edit window:* Opens the **Find** dialog box.
Shift + Click	*Graphic Display window:* Creates an Auto feature based on the highlighted CAD element.	Ctrl + Shift + F	*Edit window*: This works only in Command Mode. This option toggles the display of selected feature commands and their IDs in the Graphic Display window. You can also use this on a GROUP command to toggle the display of features within the group.	Alt + Backspace	*Edit window:* Undoes the last action taken in the Edit window.

Key	Description	Key	Description	Key	Description
Ctrl + F6 or Ctrl + Shift + F6	Switches between certain open windows to make the next window active. For example, the Graphic Display window and the Report Window. The windows must be open for this to work.	Alt + Click	*Graphic Display window:* With the **CAD Assembly** dialog box open, if you press and hold the Alt key and then left-click a CAD object in the Graphic Display window, this shortcut highlights its corresponding component in the tree view.	Shift + Backspace	*Edit window:* Redoes the last undone action in the Edit window.
				Alt + Right-mouse drag	*Graphic Display window:* Rotates the CAD model in 2D.
				Alt + Click	*Graphic Display window:* Toggles the marked status of the underlying feature in the Edit window.

Related Topics:

Command Mode Shortcut Keys

Summary Mode Shortcut Keys

Shortcut Menus Reference

Shortcut menus are contextual menus that appear when you right-click on certain areas of the user interface. This section explains how to access the various shortcut menus and what the various menu options do.

Shortcut Menus in Standard Windows Dialog Boxes

Beyond the standard commands that exist in the Standard Windows dialog boxes (such as the **Open**, **Save**, and **Print** dialog boxes), PC-DMIS, in certain circumstances, adds additional functionality to shortcut menus. Many of these are explained in the "Using Basic File Options" and "Using Advanced File Options" chapters.

When opening, saving, or using other file operations, right-click on the measurement routine file name listed in the dialog box to bring up these shortcut menu options.

PC-DMIS Import
Imports data from an input file into the selected measurement routine. See "Importing CAD Data or Feature Data" in the "Using Advanced File Options" chapter.

PC-DMIS Export
Exports data to an output file from the selected measurement routine. See "Exporting CAD Data or Feature Data" in the "Using Advanced File Options" chapter.

Open
Opens a measurement routine. See "Opening Existing Measurement Routine" in the "Using Basic File Options" chapter.

Cut
Cuts a measurement routine and moves it to the Clipboard; this is used with Windows **Paste** command to place the cut file into another folder.

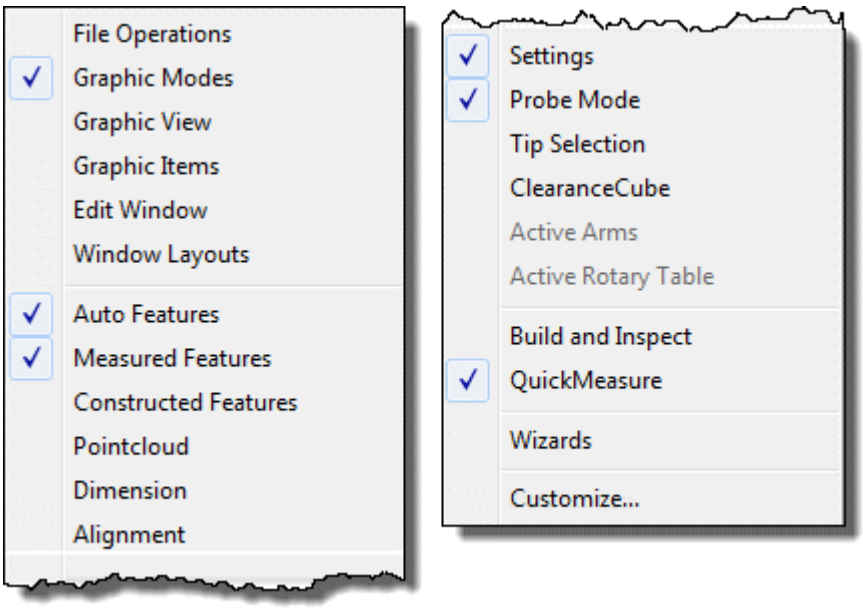
Copy
Copies a measurement routine to the Clipboard; this is used with the **Paste** command to place the copied file into another folder. See "Performing File Operations" in the "Using Basic File Options" chapter.

Rename
Renames a measurement routine.

Delete
Deletes a measurement routine. See "Performing File Operations" in the "Using Basic File Options" chapter.

Shortcut Menus on the Toolbar Area

When you right-click on the **Toolbar** area, PC-DMIS displays a list of the toolbars and other options that you can instantly access. These options include:



The following options are available:

File Operations
Adds or removes the File Operations toolbar to or from the toolbar area.

Graphic Modes
Adds or removes the Graphic Modes toolbar to or from the toolbar area.

Edit Window
Adds or removes the Edit Window toolbar to or from the toolbar area.

Window Layouts
Adds or removes the Window Layouts toolbar to or from the toolbar area.

Auto Features
Adds or removes the Auto Features toolbar to or from the toolbar area.

Measured Features
Adds or removes the Measured Features toolbar to or from the toolbar area.

Constructed Features
Adds or removes the Constructed Features toolbar to or from the toolbar area.

Pointcloud
Adds or removes the Pointcloud toolbar to or from the toolbar area.

Dimension
Adds or removes the Dimension toolbar to or from the toolbar area.

Alignment
Adds or removes the Alignment toolbar to or from the toolbar area.

Settings
Adds or removes the **Settings** toolbar to or from the toolbar area.

Probe Mode
Adds or removes the **Probe Mode** toolbar to or from the toolbar area.

Tip Selection
Adds or removes the **Select Tip** toolbar to or from the toolbar area.

ClearanceCube
Adds or removes the **ClearanceCube** toolbar to or from the toolbar area.

Active Arms
Adds or removes the **Active Arms** toolbar to or from the toolbar area.

Active Rotary Table
Adds or removes the **Active Rotary Table** toolbar to or from the toolbar area.

Build and Inspect
Adds or removes the **Build and Inspect** toolbar to or from the toolbar area.

QuickMeasure
Adds or removes the specific **QuickMeasure** toolbar (CMM QuickMeasure Toolbar, Portable QuickMeasure Toolbar, Vision QuickMeasure Toolbar, QuickCloud Toolbar) to or from the toolbar area.

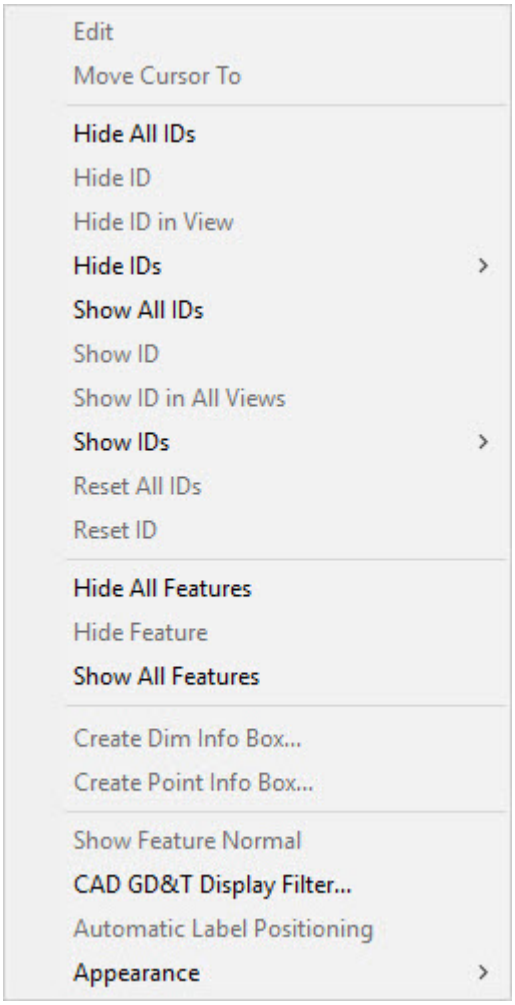
Wizards
Adds or removes the **Wizards** toolbar to or from the toolbar area.

Customize
Customizes the toolbars and the menus to meet specific needs. See "Customizing the User Interface" in the "Navigating the User Interface" chapter.

For information on these toolbars, see the "Using Toolbars" chapter.

Shortcut Menus in the Graphic Display Window

The following shortcut menu is available inside the Graphic Display window (**View | Graphic Display Window**) when you right-click on empty space.



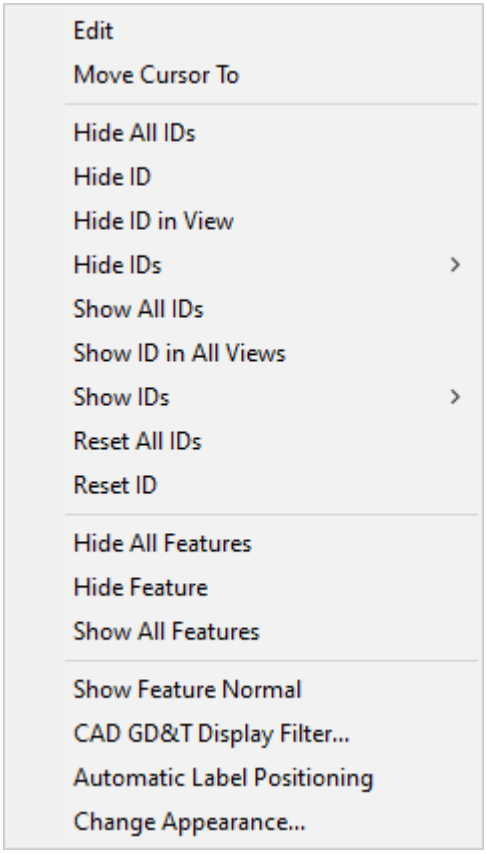
Option	Discussed in Topic
Edit	Feature Shortcut Menu
Move Cursor To	Feature Shortcut Menu
Hide All IDs	Feature Shortcut Menu
Hide ID	Feature Shortcut Menu
Hide ID in View	Feature Shortcut Menu
Hide IDs	Feature Shortcut Menu
Show All IDs	Feature Shortcut Menu
Show ID	Feature Shortcut Menu

Show ID in All Views	Feature Shortcut Menu
Show IDs	Feature Shortcut Menu
Reset All IDs	Feature Shortcut Menu
Reset ID	Feature Shortcut Menu
Hide All Features	Feature Shortcut Menu
Hide Feature	Feature Shortcut Menu
Show All Features	Feature Shortcut Menu
Create Dim Info Box	Feature Shortcut Menu
Create Point Info Box	Feature Shortcut Menu
Show Feature Normal	Show Feature Normal Option (This topic is in the "Editing the CAD Display" chapter.)
CAD GD&T Display Filter	Feature Shortcut Menu
Automatic Label Positioning	Feature Shortcut Menu
Appearance	Feature Shortcut Menu

Additional menu options become available only if you right-click on specific areas. For more information, see the topics below.

Feature Shortcut Menu

To access this shortcut menu, from the Graphic Display window, right-click on a feature ID label, Diminfo box, Pointinfo box, or CAD GD&T callout. You can also see many of these options if you right-click anywhere in the Graphic Display window.



Right-clicking on a GD&T callout displays many of these same menu items; however, only the "show" and "hide" menu items function in this context.

The following options are available:

Edit
This menu option opens the appropriate dialog box for the underlying command in the Edit window so that you can use the dialog box to edit that command.

For Dimension Info boxes with legacy graphics labels, this opens the **Edit Dimension Info** dialog box as usual. For information, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

For Dimension Info boxes with new graphics labels, this opens the **Properties** dialog box for that label so that you can see the pathway to the underlying template in the **Template Path** field.

You can change what graphics labels the software uses with the **Use legacy graphics labels** check box in the **Feature Appearance** dialog box. For more information on this dialog box, see the "Editing Feature Appearance" topic in the "Editing the CAD Display" chapter.

Move Cursor To
Select this menu option to move the Edit window's cursor to this feature's command line in the Edit window. This *only* affects the location of the cursor in the Edit window; the location of the probe in the Graphic Display window remains unchanged.

Hide All IDs
Hides all Feature Labels, Dimension Infos, and Point Infos for all features.

Hide ID
This menu option hides a single ID label in the Graphic Display window. A hidden label still exists; it's merely hidden from view.

An ID Label in this topic refers to any number of identification labels. They could be any of the following:

- Feature ID
- Dimension info text box
- Point info text box
- Feature Control Frame
- CAD GD&T callout

Hide ID in View
This hides the selected feature's ID label only in the current view. In other views, it remains visible.

An ID Label in this topic refers to any number of identification labels. They could be any of the following:

- Feature ID
- Dimension info text box
- Point info text box
- Feature Control Frame
- CAD GD&T callout

Hide IDs
This submenu lets you hide the different feature ID labels. ID labels that you hide still exist and are merely hidden from view.

You can choose to show or hide these types of ID labels:

- **Feature Labels** - Hides all Feature Labels for all features.
- **Feature Labels visible in view** - Hides all Feature Labels for all features visible in the current view.
- **Dimension Infos** - Hides all Dimension Infos for all features.
- **Point Infos** - Hides all Point Infos for all features.

Show All IDs
Shows all Feature Labels, Dimension Infos, and Point Infos for all features.

Show ID in All Views
This shows the selected feature's ID label in all views.

An ID Label in this topic refers to any number of identification labels. They could be any of the following:

- Feature ID
- Dimension info text box
- Point info text box
- Feature Control Frame

Show IDs
This submenu lets you show the different feature ID labels. For information, see the Hide IDs documentation above. The show menu items do the opposite of those "hide" menu items.

Reset All IDs
This option moves all ID labels back to their original locations next to their respective features.

Reset ID
This option moves a single ID label back to its original location next to the feature.

Hide All Features
This option hides all features on a part and removes them from the Graphic Display window. The features still exist and are merely hidden from view.

Hide Feature
This option hides a single feature and removes it from the Graphic Display window. The feature still exists and is merely hidden from view.

Show All Features
This option displays all previously hidden features in the Graphic Display window.

- If you select **Hide All Features**, **Hide Feature**, and **Show All Features**, the icon also turns on or off in the Summary Mode Edit window.
- If you select **Hide ID** and **Show ID**, the icon turns on or off only for the Point Infos, Dimension Infos, Datums, Pointclouds, Mesh, and Basic scan commands but not for the features.

Show Feature Normal
For information, see "Show Feature Normal Option" in the "Editing the CAD Display" chapter.

CAD GD&T Display Filter
This menu option displays a dialog box allowing you to filter what CAD GD&T callouts are displayed. It appears in the shortcut menu if your CAD model contains embedded CAD GD&T information. See "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.

Automatic Label Positioning
This menu option allows you to perform a one-time automatic positioning of feature ID labels around the edges of the current view.

Change Appearance
This menu option opens the **Change Label Options** dialog box. With this dialog box, you can change the appearance of the different labels in the Graphic Display window. There are three different label types: feature ID labels, Dimension Info boxes, and Point Info boxes.

The dialog box contains these items:

Background Color
This box opens a **Color** dialog box so that you can change the label's background color.

Background Transparency
This slider or the box to its right adjusts the transparency level of the label's background color. The value can be from 0 (opaque) to 100 (completely transparent).

Line Color
This box opens a **Color** dialog box so that you can change the label's border color.

Show Shadow
This check box displays a small shadow under the label.

You can choose to apply your changes to all label types or to just the current label type.

To make your changes the default settings, click the **Default** button.

Feature Shortcut Menu Additions in Text Box Mode

In Text Box Mode ( ), PC-DMIS adds these additional options to the standard feature shortcut menu:

Show ID
Shows the label of the selected feature. (This is only available if you right-click near a feature with its label hidden.)

Create Dim Info Box
This option creates a Dimension Information text box from the highlighted feature. An **Edit Dim Info** dialog box appears and shows the available dimensions from which you can create the Dimension info text box.

If no dimensions are associated with the feature, PC-DMIS automatically creates a default Location dimension for the feature (see "Dimensioning Location" in the "Using Legacy Dimensions" chapter). For information on how to use the **Edit Dimension Info** dialog box, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

Create Point Info Box
This menu option creates a Point Info text box from the highlighted feature. An **Edit Point Info** dialog box appears and shows the feature (and any available dimensions) from which you can create the Point Info text box. For information on how to use the **Edit Point Info** dialog box to create a Point Info text box, see "Inserting Point Info Boxes" in the "Inserting Report Commands" chapter.

To access these added options:

1. Select the **Text Box Mode** icon. For information, see "Text Box Mode" in the "Editing the CAD Display" chapter).

2. Right-click on a feature or feature's ID label in the Graphic Display window to display the shortcut menu.

Dimension/Point Info Menu in Text Box Mode

When in Text Box Mode ( ), if you right-click on a Dim Info box or Point Info box, PC-DMIS adds these two options to the standard feature shortcut menu:

Edit
This option allows you to edit the selected Dimension Info or Point Info text box. A slightly modified **Edit Dimension Info** or **Edit Point Info** dialog box opens. In addition to the standard buttons along the bottom of the dialog box, an additional **Apply To All** button is available.

Click **Apply To All** to have any changes made in the dialog box apply to *all* Dimension Info or Point Info boxes:

Only the changes made from the original to the modified info box will be applied across all Dim Info or Point Info boxes. Settings on the original info box that already differ from other info boxes will not be applied to the other info boxes.

For additional information see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

Delete
Select this menu option to delete the selected dimension or point info text box. The corresponding POINTINFO or DIMINFO command in the Edit window will also be deleted.

To access these added options:

1. Select the **Text Box Mode** icon (see "Text Box Mode" in the "Editing the CAD Display" chapter).
2. Right-click on a **Dim Info** or **Point Info** box in the Graphic Display window.

Related Topics

Layout

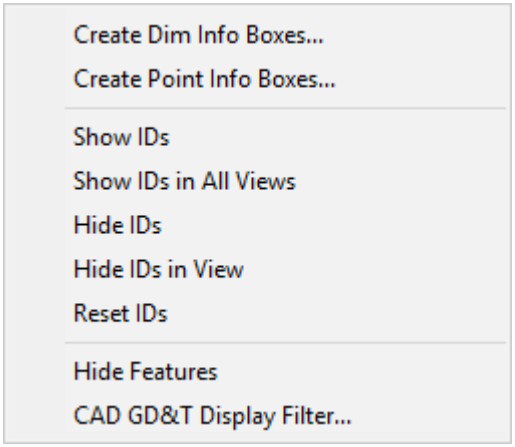
Box-Select Shortcut Menu

This shortcut menu only works in **Text Box Mode**.

To access this menu, follow these steps:

1. From the **Graphic Modes** toolbar, select the **Text Box Mode** icon ( ). (For information on this mode, see "Text Box Mode" in the "Editing the CAD Display" chapter.)
2. Box-select one or more features to select all graphical features or feature ID labels inside the box.

A shortcut with these options appears:



The following options are available:

Create Dim Info Boxes
This menu option creates Dimension Info text boxes from the selected features.

PC-DMIS displays the **Edit Dimension Info** dialog box for one of the highlighted features. The dialog box shows the available dimensions from which you can create the first Dimension Info box.

Once you select the desired options in the **Edit Dimension Info** dialog box, click either the **OK** or **Create** button (**Apply** button when editing) to create the Dim Info boxes.

PC-DMIS creates the first Dim Info box from any selected dimensions in the **Edit Dimension Info** dialog box.

PC-DMIS then creates subsequent info boxes from all dimensions associated with each feature.

If no dimensions are associated with any of the features, PC-DMIS creates a default Location dimension for that feature (see "Dimensioning Location" in the "Using Legacy Dimensions" chapter). For information on how to use the **Edit Dimension Info** dialog box, see "Inserting Dimension Info Boxes" in the "Inserting Report Commands" chapter.

Create Point Info Boxes
Select this menu option to create Point Info text boxes from the highlighted features. PC-DMIS displays the **Edit Point Info** dialog box for one of the highlighted features. This dialog box shows the feature (and any available dimensions) from which you can create the Point Info text box.

Once you've selected the desired options in the **Edit Point Info** dialog box, select the **OK** or **Create** button (**Apply** button when editing) to create the info boxes.

PC-DMIS creates the first info box from the selected feature or dimension (or dimensions) in the **Edit Point Info** dialog box. PC-DMIS creates the

subsequent Point Info boxes from all dimensions associated with each feature and the feature itself. A Point Info box is created for all dimension and feature hits. For information on how to use the **Edit Point Info** dialog box to create a Point Info text box, see "Inserting Point Info Boxes" in the "Inserting Report Commands" chapter.

Show IDs, Hide Ids
Select one of these menu options to show or hide all the selected features' ID labels. If you choose to hide the ID labels, the ID labels still exist and are merely hidden from view.

Show IDs in All Views
This option shows all the selected ID labels in all of the available Graphic Display window view panes.

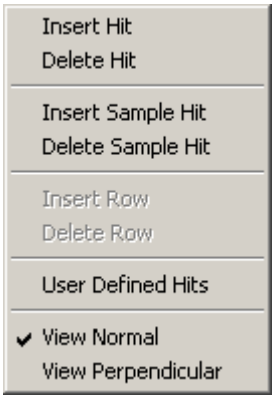
Hide IDs in View
This option hides all the selected ID labels in the current Graphic Display window view pane.

Reset Ids
This option moves all selected ID labels back to their original locations next to their respective features.

Hide Features
This option hides all selected features on a part, removing them from the Graphic Display window. The features still exist and are merely hidden from view.

CAD GD&T Display Filter
This menu option displays a dialog box allowing you to filter what CAD GD&T callouts are displayed. It appears in the shortcut menu if your CAD model contains embedded CAD GD&T information. See "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.

Auto Feature Path Lines Shortcut Menu



To access this shortcut menu, select the **Show Hit Targets Toggle** button inside an **Auto Feature** dialog box (**Insert | Feature | Auto | <Feature Type>**).

The following options are available:

Insert Hit
Inserts an additional hit into the Auto feature.

Delete Hit
Deletes the selected hit from the Auto feature.

Insert Sample Hit
Inserts a sample hit for the Auto feature.

Delete Sample Hit
Deletes the selected sample hit from the Auto feature.

Insert Row
When working with a feature that uses multiple rows of hits, such as a cylinder or cone, this inserts another row of hits.

Delete Row
When working with a feature that uses multiple rows of hits, such as a cylinder or cone, this deletes the selected row of hits.

User Defined Hits
This menu option becomes automatically selected whenever you manually make a change to a hit or path using the mouse.

If you then deselect this menu option, PC-DMIS reverts your change back to what it was before.

Additionally, if you have this option selected for a hit and then change to the feature's perpendicular view with **View Perpendicular**, you can adjust the depth or height for that specific hit.

View Normal
This changes the CAD view of the feature to a top-down view.

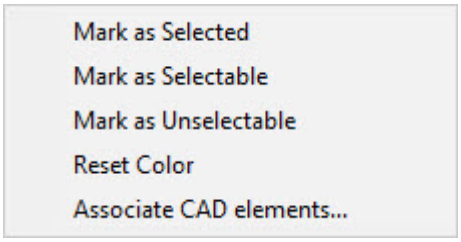
View Perpendicular
This changes the CAD view of the feature to a side view, ideal for defining a feature's depth or adding additional levels of hits.

CAD GD&T Callouts Shortcut Menu

You can use this shortcut menu to set CAD GD&T Callouts to different colors to better communicate the state of the current callout (selected, selectable, or unselectable).

To access this menu, the software must be in GD&T selection mode. To enable this mode, from the **Graphic Modes** toolbar, enable **GD&T selection mode** .

You can then right-click on a CAD GD&T callout to see the menu options:



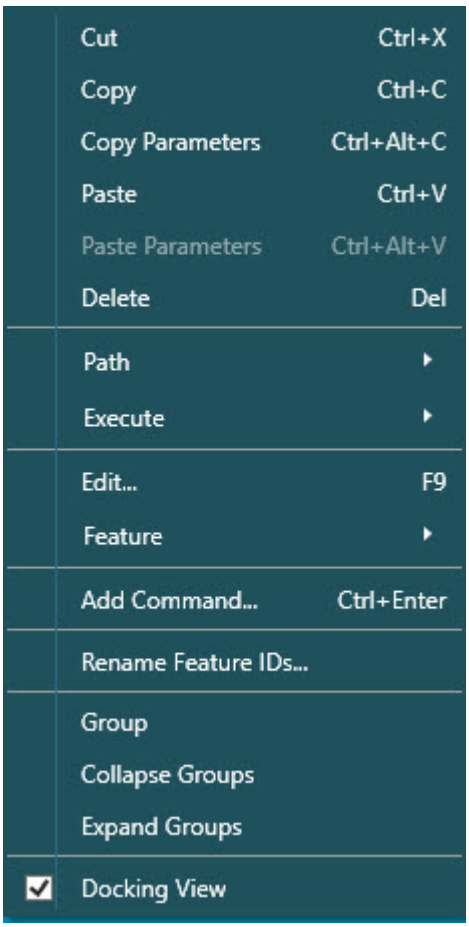
- **Mark as Selected** - This changes callout's color to a gray color to signify that you have selected this callout.
- **Mark as Selectable** - This changes the callout's color to the selectable color to signify that this callout is available for selection.
- **Mark as Unselectable** - This changes the callout's color to the invalid (unselectable) color to signify that this callout is invalid and cannot be selected.
- **Reset Color** - This changes the color of callout's color back to original color when you imported the model.
- **Associate CAD elements** - This opens a dialog box so that you can manually link GD&T CAD callouts to CAD elements on the model. For more information, see "Associating CAD GD&T Callouts with CAD elements" under "Working with CAD GD&T Callouts" in the "Editing the CAD Display" chapter.

Shortcut Menus in the Edit Window

The PC-DMIS Edit window contains the following shortcut menus depending on which Edit window mode you select.

Summary Mode Command Shortcut Menu

When you select a command item in the Edit window's Summary Mode (see "Working in Summary Mode" in the "Using the Edit Window" chapter) and right-click on the command item, the command shortcut menu appears. This menu has the following options:



Summary Mode Command Shortcut Menu

The following options are available:

Cut (Shortcut Key stroke: Ctrl + X)

This option cuts the selected command to the Clipboard and removes the command from the measurement routine.

Copy (Shortcut Key stroke: Ctrl + C)

This option copies the selected command to the Clipboard.

Copy Parameters (Shortcut Key stroke: Ctrl + Alt + C)

This option copies the selected parameters to the Clipboard.

Paste (Shortcut Key stroke: Ctrl + V)

This option pastes a command that you previously cut or copied from the Clipboard and places it below the selected command.

Paste Parameters (Shortcut Key stroke: Ctrl + Alt + V)

This option pastes the copied parameters from the Clipboard into the Edit window at the cursor position.

Delete (Shortcut Key stroke: Del)

This option deletes the selected command and any sub-commands out of the measurement routine.

Path | Path Lines

This option shows only the path lines for a range of selected features. See "Showing, Animating, and Moving Path Lines" in the "Editing the CAD Display" chapter.

Path | Collision Detection

This option shows path lines and performs the collision detection for a range of selected features. See "Detecting Collisions" in the "Editing the CAD Display" chapter.

Path | ClearanceCube | Off

Turns the ClearanceCube feature to Off.

Path | ClearanceCube | On

Turns the ClearanceCube feature to On.

Path | ClearanceCube | Starting Face

This option enables you to select the Starting Face to align the ClearanceCube to. The options are -X, +X, -Y, +Y, -Z, +Z, and Off.

Path | ClearanceCube | Ending Face

This option enables you to select the Ending Face to align the ClearanceCube to. The options are -X, +X, -Y, +Y, -Z, +Z, Use Tip Vector, and Off.

Path | Optimize Path

This option only appears if you right-click on a `GROUP` command. It performs a path optimization on the features inside of the group. For more information, see "Path Optimization for Single Arm with Touch Probe" in the "Using Inspection Plans in PC-DMIS" appendix.

Execute | Execute From Cursor

This option starts the measurement routine at the point of the current cursor location.

Execute | Execute Block

This option executes a designated block of code.

Edit (Shortcut Key stroke: F9)

This option displays the dialog box for the current command object.

Feature | Update Nominals From CAD

This option updates the measurement routine nominals with the imported CAD nominals.

Feature | Override FindNoms

This option overrides the default FindNoms behavior of PC-DMIS during Learn mode and Execute mode. For more information, see "Overriding Found Nominals" in the "Editing a Measurement Routine" chapter.

Feature | Reset Measured Values to Nominal

This option sets all measured values to the measurement routine nominal values.

Feature | Center in CAD

This option pans and rotates the part's CAD model as needed to center the current feature in the Graphic Display window. Once centered, the feature flashes a few times. You cannot undo this pan-and-rotate operation.

Feature | Highlight in CAD (Shortcut Key stroke: Ctrl + Shift + H)

This option highlights the selected feature in the Graphic Display window. It also shows and highlights the feature label if it is hidden. To clear the highlight of the first feature, highlight a second feature.

Feature | UnHighlight in CAD (Shortcut Key stroke: Ctrl + Shift + U)

This option clears the CAD feature that you highlighted through the **Highlight in CAD** menu item.

Feature | Move Probe to Feature

This option moves the animated probe in the Graphic Display window to the CAD element for the selected feature. It becomes available for selection only in Offline mode and only if you first select a feature command. Other commands are not supported.

Feature | Move To SPH_Tip1

This option only appears when you right-click on an Auto feature. It moves the probe to the center of the selected Auto feature. This action functions the same as the **Move To** button in the **Auto Feature** dialog box (**Insert | Feature | Auto | <Feature Type>**).

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Feature | Test SPH_TIP1

This option only appears when you right-click on an Auto feature. It performs a test execution of the selected Auto feature. This action functions the same as the **Test** button in the **Auto Feature** dialog box.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Add Command (Shortcut Key stroke: Ctrl + Enter)

This option enables you to add a command to the Edit window from the **Add Command** dialog box.

STEP 1: Use the search box to type the first few letters of the command. PC-DMIS displays the filtered list.

STEP 2: Select the command from the list.

STEP 3: Press either Ctrl + Enter or Enter to place the new command into the Edit window.

- Press Ctrl + Enter to place the command after the command block used to open the shortcut menu.
- Press Enter to place the command inside the command block used to open the shortcut menu. The Enter key only functions in this manner if the command you are adding is a type of command that PC-DMIS can insert into another command block. Otherwise, PC-DMIS inserts it after the current command.

Rename Feature IDs

This option enables you to rename multiple feature IDs at the same time.

Group

This option inserts the list of contiguous commands into a user-defined group.

Collapse Groups

This option collapses all expanded user-defined groups. Other expanded items remain expanded.

Expand Groups

This option expands all collapsed user-defined groups. Other collapsed items remain collapsed.

Docking View

This option determines whether or not the Edit window is docked or undocked.

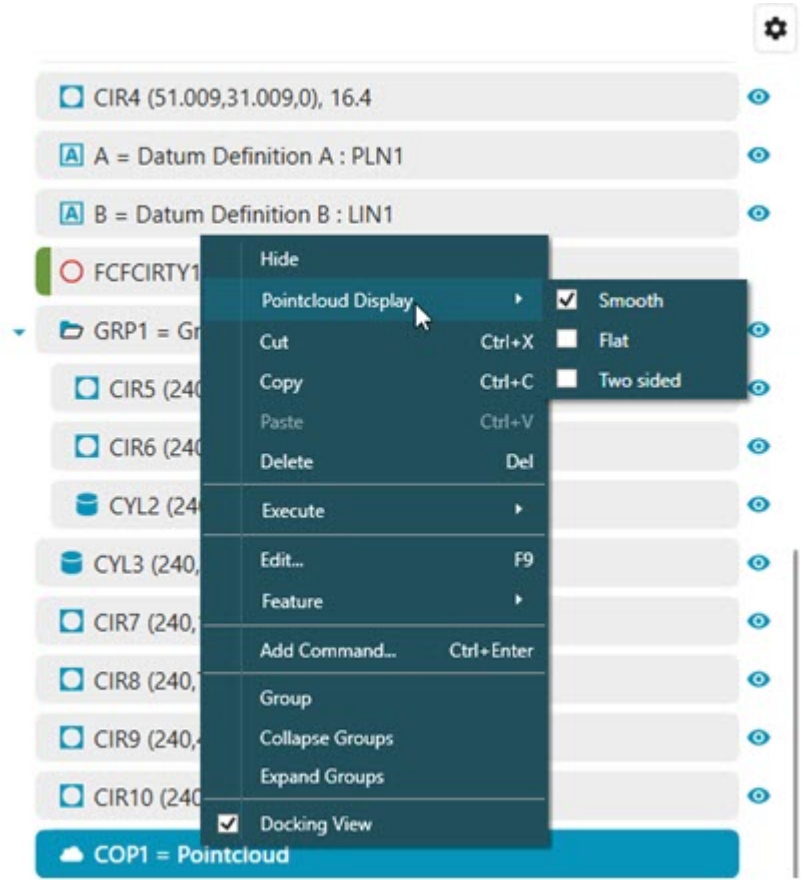
- If you deselect this option, the Edit window floats over the Graphic Display window.
- If you select this option, you can dock the Edit window to the sides or top or bottom of the Graphic Display window.

  For details on how to dock and undock user interface elements, see "Docking and Undocking User Interface Elements".

When you select a command item in the Edit window in Summary Mode (see "Working in Summary Mode" in the "Using the Edit Window" chapter) and right-click on the command item, the options on the command shortcut menu change according to the type of command you selected: general, pointcloud, pointcloud cross section operator, or Mesh.

Pointcloud Commands

The following shortcut menu appears for pointcloud commands:



Pointcloud Command Shortcut Menu

The following options are available:

Show or **Hide**

This option toggles the labels between shown and hidden. This option is also available for the Mesh commands.

Pointcloud Display

- **Smooth** - This option provides a shaded appearance using the defined COP color.
- **Flat** - This option shows the COP in a non-shaded graphical view. This selection requires the least amount of graphical memory.
- **Two sided** - This option shows a shaded appearance where the scanned side of the part is the defined COP color, and the non-scanned side is a contrasting color.
- **Mesh display** - This option shows the pointcloud as a Mesh display.

The **Mesh display** option is only available if you have the Mesh license and you scanned the COP using the **Mesh display** option (Portable only). For details, see "Pointcloud Display Section" in the PC-DMIS Laser documentation.

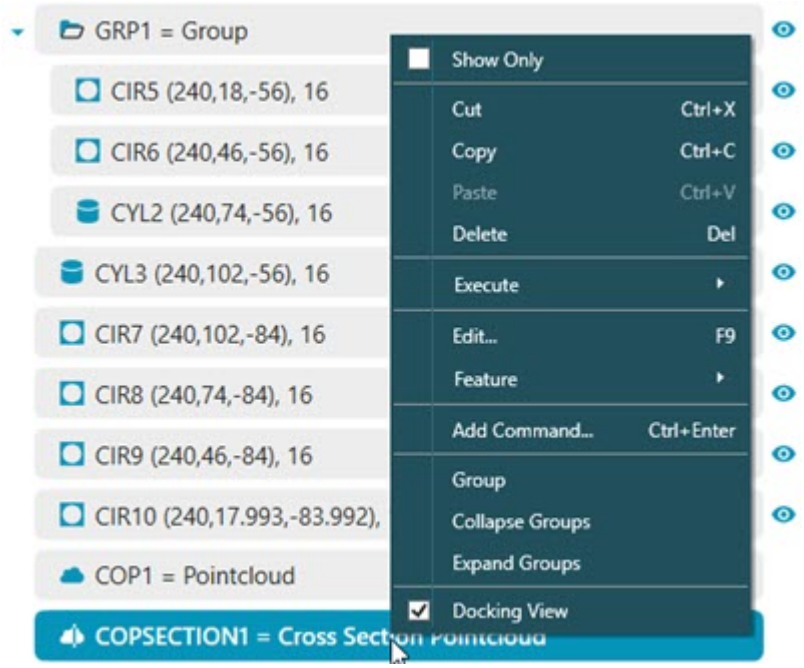
The **Mesh display** is a display setting only. The underlying data is a pointcloud.

However, the mesh display is lost, and the display reverts to points if you edit the COP (for example, if you perform any COP operation on the pointcloud).

For more information about the **Pointcloud Display** options, see "Pointcloud Graphical Representation" in the PC-DMIS Laser documentation.

Pointcloud Cross Section Operator Commands

The following shortcut menu appears for pointcloud cross section operator commands:



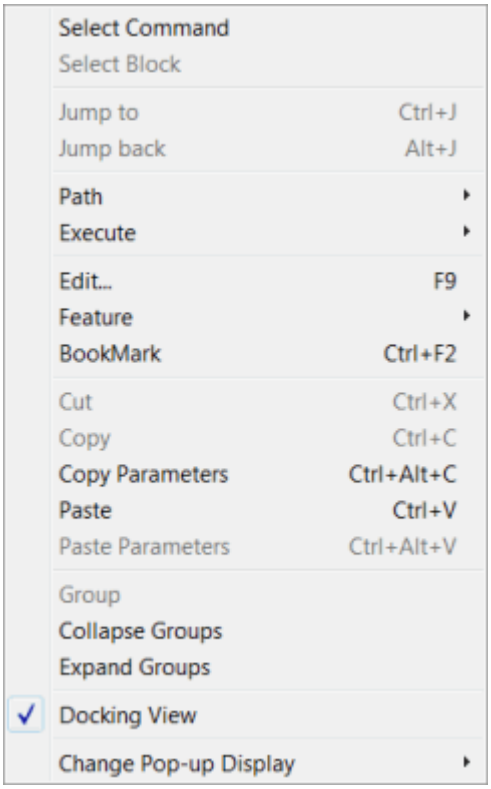
Pointcloud Cross Section Operator Command Shortcut Menu

Show Only

This option displays the 2D view of the selected cross section. When you enable the option, PC-DMIS displays a check mark to the left of it. For more information, see "2D View of Cross Sections" in the PC-DMIS Laser documentation.

Command Mode Shortcut Menu

The following shortcut menu options are available when PC-DMIS is in Command Mode. For information on these modes, see "Working in Command Mode" in the "Using the Edit Window" chapter.



The following options are available:

Select Command
This option highlights the command that the mouse is positioned over.

Select Block
If the mouse is over a start or end block object, it selects the entire block.

Jump To <Feature> (Shortcut Key stroke: Ctrl + J)
This option jumps from the current command to the referenced feature ID.

Jump Back (Shortcut Key stroke: Alt + J)
This option jumps back to the command that you were at before when you selected the **Jump To** option.

Path | Path Lines
This option shows only the path lines for a range of selected features. See "Showing, Animating, and Moving Path Lines" in the "Editing the CAD Display" chapter.

Path | Collision Detection

This option shows path lines and performs the collision detection for a range of selected features. See "Detecting Collisions" in the "Editing the CAD Display" chapter.

Path | ClearanceCube | Starting Face

The **ClearanceCube | Starting Face** option determines what ClearanceCube face PC-DMIS moves to before measuring the feature. The available settings are -X, +X, -Y, +Y, -Z, +Z, and Off.

Path | ClearanceCube | Ending Face

The **ClearanceCube | Ending Face** option determines what ClearanceCube face PC-DMIS moves to after measuring the feature. The available settings are -X, +X, -Y, +Y, -Z, +Z, and Off.

Path | Optimize Path

This option only appears if you right-click on a `GROUP` command. It performs a path optimization on the features inside of the group. For more information, see "Optimizing the Path" in the "Using Inspection Plans in PC-DMIS" appendix.

Execute | Mark (Shortcut Key stroke: F3)

This option toggles the mark state of the object for execution. For more information on marking, see "Mark" in the "Editing a Measurement Routine" chapter.

Execute | Set as Start Point

This option determines the starting location at which measurement routine execution begins when you select the **File | Partial Execution | Execute From Start Points** menu option. See "Setting Start Points" in the "Editing a Measurement Routine" chapter.

Execute | Execute From Cursor

This option starts execution from the current location of the insertion point in the Edit window.

Execute | Execute Block

This option executes the currently marked block of commands.

Execute | Breakpoint

This option inserts a breakpoint at the cursor's location. For more information on breakpoints, see "Using Breakpoints" in the "Editing a Measurement Routine" chapter.

Edit (Shortcut Key stroke: F9)

This opens the dialog box that enables you to edit the command that the cursor resides on.

Feature | Update Nominals From CAD

This option updates the routine Nominals with the imported CAD Nominals.

Feature | Override FindNoms

This option accesses the **Override FindNoms** dialog box so that you can override the default find nominals behavior for the points you right-clicked on in the Edit window. See "Overriding Found Nominals" in the "Editing a Measurement Routine" chapter.

Feature | Reset Measured Values to Nominal

This option sets all measured values to the routine nominal values.

Feature | Center in CAD

This option pans and rotates the part's CAD model as needed to center the current feature in the Graphic Display window. Once centered, the feature flashes a few times. You cannot undo this pan and rotate operation.

Feature | Highlight in CAD

This option highlights the selected feature in the Graphic Display window. It also shows and highlights the feature label if it is hidden. Highlighting a second feature clears the highlight of the first feature.

Feature | UnHighlight in CAD

This option clears the highlighted CAD feature made using the **Highlight in CAD** menu item above.

Feature | Move Probe to Feature

This option moves the animated probe in the Graphic Display window to the CAD element for the selected feature. It becomes available for selection only in Offline mode and only if you first select a feature command. Other commands are not supported.

Feature | Hide Feature(s)

This option toggles the display of selected feature commands and their IDs in the Graphic Display window. You can also use this on a GROUP command to toggle the display of features within the group.

The shortcut for this is Ctrl + Shift + F.

Feature | Test <Feature>

This option only appears when right-clicking on an Auto feature. It performs a test execution of the selected Auto feature. This functions the same as the **Test** button in the **Auto Feature** dialog box.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

Feature | Move To <Feature>

This option only appears when right-clicking on an Auto feature. It moves the probe to the center of the selected Auto feature. This functions the same as the **Move To** button in the **Auto Feature** dialog box.

WARNING: When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

BookMark (Shortcut Key stroke: Ctrl + F2)

This option inserts a bookmark at the cursor's location. For more information on bookmarks, see "Using Bookmarks" in the "Editing a Measurement Routine" chapter.

Cut (Shortcut Key stroke: Ctrl + X)

This option cuts the selected Edit window text to the Clipboard. For more information on how to cut in the Edit window, see "Cut" in the "Editing a Measurement Routine" chapter.

Copy (Shortcut Key stroke: Ctrl + C)

This option copies the selected Edit window text to the Clipboard. For more information on how to copy in the Edit window, see "Copy" in the "Editing a Measurement Routine" chapter.

Paste (Shortcut Key stroke: Ctrl + V)

This option pastes text from the Clipboard into the Edit window at the cursor's location. For more information on how to paste in the Edit window, see "Paste" in the "Editing a Measurement Routine" chapter.

Group

This option inserts the list of contiguous selected commands into a user-defined group.

Collapse Groups

This option collapses all expanded user-defined groups. Other expanded items remain expanded.

Expand Groups

This option expands all collapsed user-defined groups. Other collapsed items remain collapsed.

Docking View

See the description given in the "Summary Mode Command Shortcut Menu".

Change Pop-up Display | Expression Value

1. Select **Expression Value**.
2. Let your cursor rest over an expression.
3. A small pop-up window appears and displays the current value of the expression.

Change Pop-up Display | Data Type Information

1. Select **Data Type Information**.

2. Let your cursor rest over a data field.

3. A small pop-up window appears and displays information about that data field, including the data type description, data type number, and type index.

The information displayed in this pop-up window can be used with the GETTEXT expression. See the "Using Expressions and Variables" chapter.

Change Pop-up Display | Command Information

1. Select **Command Information**.

2. Let your cursor rest over any Edit window command.

A small pop-up window appears and displays command description information and the command type number. The information displayed in this pop-up window can be used with the GETCOMMAND expression. See the "Using Expressions and Variables" chapter.

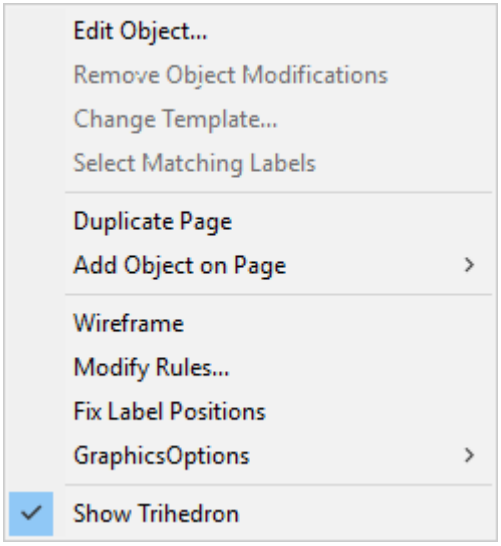
Object

This menu option remains unavailable for selection until you insert an external object into the Edit window. For information on how to do this, see "Inserting External Objects" in the "Adding External Elements" chapter.

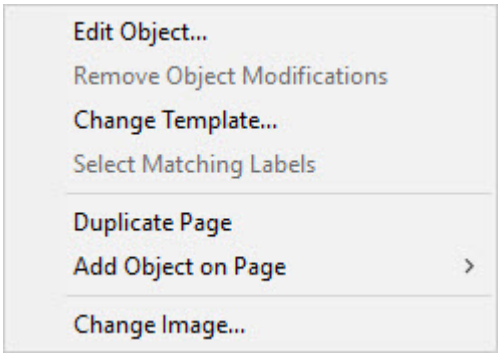
Once you select the inserted external object in the Edit window and then right-click on it, this menu changes to reflect options unique to that particular object. These may include options such as opening the object, editing the object, or modifying the object's properties.

Shortcut Menus in the Report Window

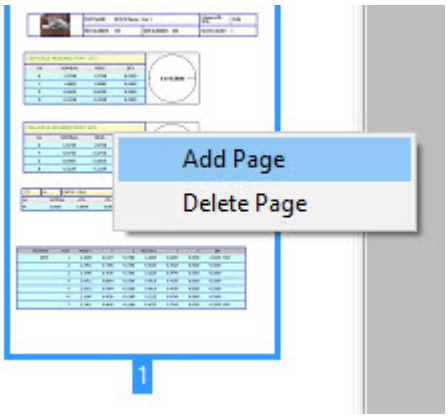
If you right-click in the Report window, PC-DMIS displays a shortcut menu with menu items that affect how objects appear in the Report window. Certain menu items only appear if you right-click on specific underlying objects in the Report window. The various menu items you can access from the Report window's shortcut menu are discussed in the table below:



Report Graphic Display - Shortcut Menu



Report Header - Shortcut Menu



Shortcut menu with mouse pointer over a custom report's thumbnail image in the Preview Pane

Edit Object
This option allows you to modify the current object in the Report window from its default state.

- If you right-click on a label object in the Report window, the **Label Properties** dialog box appears, so that you can modify that label.

- If you right-click on the TextReportObject (essentially any white space text at the end of your report or any non-table-like reporting text), the **Report** dialog box appears so that you can determine what gets included in your report.

You can press **Shift** and right-click on any label object to display the **Report** dialog box.

For more information, see "Changing the Report Window's Contents" in the "Reporting Measurement Results" chapter.

Hide Selected Labels
This option hides any selected labels used with the CadReportObject. Select the labels you want to hide and then right-click on one of the selected labels for this menu item to appear. To select multiple labels at a time, drag a box around the desired labels with the mouse or hold down the Ctrl key and click on the desired labels.

Show Hidden Labels
This option shows all hidden labels. Right-click on the CadReportObject for this menu item to appear.

Remove Object
This option removes a selected label object or section cut profile object that you added using the **Add Object on Page** submenu.

You should not use the **Delete** key when you try to remove selected objects; doing so deletes selected commands in the Edit window instead.

Remove Object Modifications
This option returns the object you right-click on to its original state and removes any modifications you may have made using the **Edit Object** menu item or made resizing or repositioning supported objects.

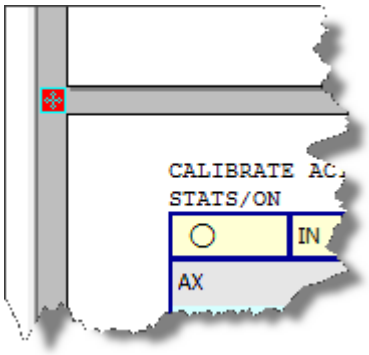
Change Template
This option displays an **Open** dialog box, so that you can change the template used to display and format the data from the underlying command. Only those templates specified to work with the command appear in the dialog box.

If you have multiple labels selected when you use this option (such as after you use the **Select Matching Labels** menu item), PC-DMIS applies the label change to all the selected labels.

Select Matching Labels
This option appears when you right-click on a label. This option selects all labels on the current page that use the same label template as the label on which you clicked.

Duplicate Page
This option creates an identical page of the one on which you right-click immediately following the current page. You may find this helpful to organize labels that can't fit on one page across multiple pages.

Duplicated pages show a red cross icon at the top-left area of the page.

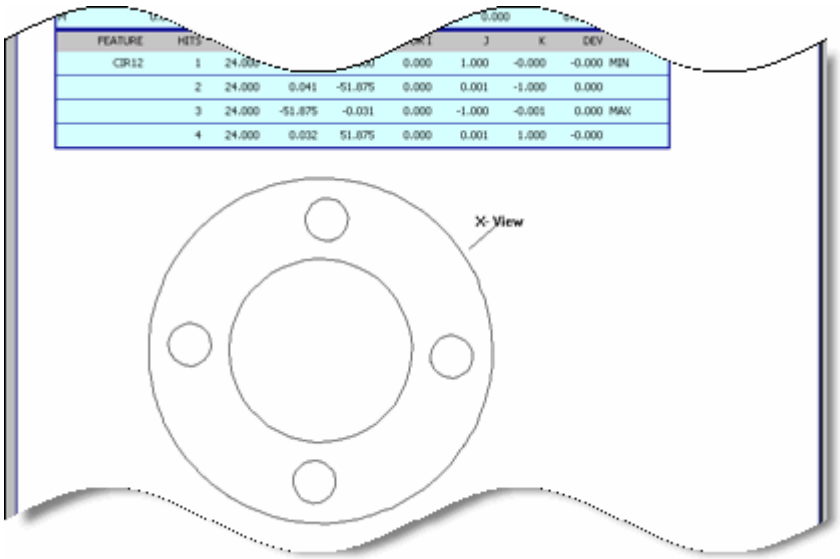


Example of a cross icon indicating a duplicated page.

To remove a duplicated page, right-click on it and select **Remove Duplicated Page**.

Remove Duplicated Page
This option only appears in the menu when you right-click on a previously duplicated page. Select it to remove the duplicated page from the report.

Add Object on Page | Section Cut Profile
This option inserts a section cut profile of your part model into the report.



A sample section cut profile.

After you select this option, use your mouse and drag a box in the Report window to insert the section cut profile. For detailed information, see the information in the "SectionCutObject" topic in the "Reporting Measurement Results" chapter.

You can use this method to insert multiple cutaway views into your report.

Change Image
This option only appears in the menu when you right-click on a header label template (such as FILE_HEADER.LBL). When you choose this option, you can use an **Open** dialog box to choose a new image for the report's header. After you locate the new image and click **Open**, the report uses that image. This action also updates the underlying label template (.lbl) with the new image.

Add Object on Page | Custom Label
This menu item lets you insert a custom label object on the fly into the active report. This type of label insertion process is not tied in any way to the Rule Tree Editor.

Select this menu item and then drag a box in the Report window. An **Open** dialog box appears so that you can insert a selected label template on the fly into the current page of the report.

Remove CAD Report Object Pan/Zoom/Rotate MOD
This menu item lets you remove just pan, zoom, and rotate modifications for the CadReportObject. Other modifications (such as section cuts, various label definitions, solid or wire displays and so forth) are not affected.

Select Labels
This menu item allows you to easily select multiple labels across different sections in your report.

When you select this item, PC-DMIS displays the **Select Labels** dialog box. This dialog box displays the labels in your report that are tied to a CadReportObject. Select any labels from this dialog box; when you click **OK** on the dialog box, PC-DMIS selects the corresponding labels in the report.

You can limit the labels that appear in the dialog box by feature type, dimension type, or all commands.

For more information, see "Changing the Report Window's Contents" in the "Reporting Measurement Results" chapter.

Wireframe
This menu item toggles the display of the selected CadReportObject by displaying the model as either a wireframe or a solid. This menu item only appears if you right-click on a CadReportObject in your report.

Modify Rules
This menu item displays a simplified version of the **Rule Tree Editor** dialog box. You can use the **Disable** or **Enable** options on this dialog box to turn rules on or off.

Show Previously Unused Labels
This menu item works with labels on a CadReportObject in a duplicated page.

If you turn off a set of labels on the CadReportObject on one page and then duplicate that page, you can use this menu item on the duplicated page to get it to show the labels not used on the original page.

Fix Label Positions
This menu item creates a position modification for each label so that the next time you open the report, PC-DMIS won't recalculate label positions.

GraphicsOptions | None
This menu item hides all labels on the CADReportObject.

GraphicsOptions | Labels
This menu item shows all labels on the CADReportObject.

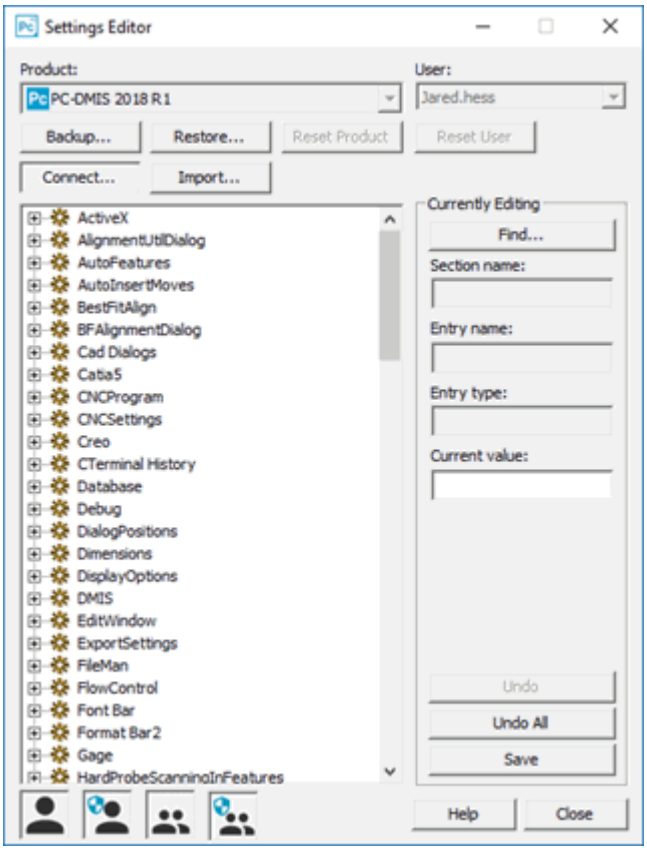
Add Page (applies to Custom Reports only)
This menu item adds a new page to the end of your custom report.

Delete Page (applies to Custom Reports only)
This menu item deletes that page from your custom report. You must always have at least one page in your report.

Modifying Setting Entries

Modifying Setting Entries: Introduction

You can use the PC-DMIS Settings Editor application to change entries that control various PC-DMIS settings.



Settings Editor Dialog Box

To launch the PC-DMIS Settings Editor and its documentation:

1. Close the PC-DMIS application if open.
2. Access the PC-DMIS **Settings Editor** shortcut from the program group where you installed PC-DMIS.
3. Once the Settings Editor opens, click **Connect** to have it connect to the main application. This may take some time.
4. Once the Settings Editor connects, press F1 (or click the **Help** button) to get information on how to use the Settings Editor.

   For more information, see the "PC-DMIS Settings Editor" documentation.

If this is your first time using the Settings Editor since PC-DMIS was installed, PC-DMIS needs to remain open as you click **Connect**. After the first connection succeeds, you can later run the Settings Editor by itself.

Glossary

95 % Confidence Factor
Given a Gaussian distribution (or filter) of errors, an adequate number of hits, and the resulting standard deviation, you can be 95% confident that the actual dimension will be less than the measured value.

AB angles
The position of these A and B axes determine the angles a probe will use during tip calibration. These are also referred to as AB positions. * A is the vertical wrist angle. A = 0 when the tip is pointing straight down, and it is normal to the probe. * B is the rotation angle. B = 0 will vary according to machine type and brand and also probe head type.

Active Tip
The active tip refers to the part of the probe (also referred to as the stylus) that is currently loaded for use. Several tips can be considered active at a time and are referenced in the Active tip list.

Active Tool
This refers to the calibration tool that is used to calibrate the active tip. See also Tool.

ACTL
This means Actual, meaning the "measured" size or location information as opposed to the "theoretical" size or location information.

Actual Point (APT)
Out of the multiple readings that the controller takes during the probing cycle, the Actual Point is the one that is returned from the controller.

Alignment Block
This is a section of measurement routine code in the Edit window beginning with the "Label = ALIGNMENT/START,..." command line and continuing to the "ALIGNMENT/END" command line.

Alignment offset
This is the distance of the part from the X, Y, and Z (0, 0, 0) location.

Alignment Rotation Feature
This Rotate button (found in Alignment dialog box) allows you to rotate the part about a particular axis. This is used for creating alignments.

Analog Probe
This type of probe is an electronic device capable of performing a continuous contact scan of a part's surface without having to leave the part's surface in order to record measurement data. When not performing scans, the probe functions similar to a Touch Trigger Probe (TTP).

Anti-aliasing
A technique used in computer software to smooth jagged edges that appear on curved or diagonal lines drawn on a computer screen.

Arguments
These are conditions used in a subroutine or equation, such as an IF THEN statement. IF a certain argument (or condition) is met THEN a certain result follows.

Assignment
Assignment is the process of giving the value of an operation, set of operations, or a real number to a variable and is accomplished by use of the assignment operator (=). The assignment operator is automatically created when the assignment option is selected (from the menu bar, select the Assignment option).

Asterisk (*) Meta-Character
The asterisk (*) meta-character will match, or take the place, of any character or characters in a search.

Autojoint
A connection where a key is used to engage the male and female halves of this coupling. You manually use a key to engage and disengage the Autojoint for systems with no probe racks. For rack systems, this is done with a rack-and-pinion-style device to turn the key.

Axes
This is plural for axis. See Axis.

Axis
An axis is one of the reference lines in a system of coordinates. The various axes (plural for axis) are XPLUS, XMINUS, YPLUS, YMINUS, ZPLUS, and ZMINUS. An axis can also be defined as the derived median line along the length of a cylinder, cone, slot, or ellipse.

Axis Counters
These are digital readout hardware devices connected to your machine that give a readout of the machine's axes X, Y, and Z positions.

Ball Tip
Tip shaped as a small ball.

Balloon IDs
See "Characteristic IDs".

Baricenter
The center of gravity for a hole of any shape.

Basic Script
A basic script is a series of basic commands written in the programming language BASIC.

Baud
This is the rate at which data can be transmitted or received by a computer. This is usually defined as a number of bits per second (bps).

Bend Radius
This is essentially an outside cylinder (pin) measurement for sheetmetal. It measures the center location and size.

Best Fit
This is a mathematical process that calculates feature or alignment parameters by minimizing certain error conditions between measured and theoretical points or features.

Best Guess Algorithms
These are mathematical calculations that PC-DMIS uses to determine what feature type is being measured based on the number of hits taken. Based on the results of the calculations, PC-DMIS makes its best guess to determine the feature type. Incorrect guesses can be overridden using the Override command on the main menu.

Block Edit
This refers to moving a portion of text from one area of the Edit window to another area.

Boss
A "boss" is usually an external cone or cylinder that has a flat top to it that a bolt may thread into. Sometimes this is also termed as a "Stud".

Boundary Crossings
This numerical value (used in the Number of Boundary Crossings box) tells PC-DMIS how many times the probe's ball center can cross the given condition's surface (planar, spherical, cylindrical) before stopping a given scan. Once the ball center crosses the condition's surface for the nth time (where n = the number entered), then the scan stops.

Boundary Plane
The Boundary Plane is created perpendicular to the Boundary Plane Vector with the same XYZ coordinates as the start point of the scan. The Boundary Plane is often used to determine when to stop a scan by indicating a maximum number of times the scan may cross the boundary plane.

Boundary Plane Vector
The Boundary Plane Vector is determined initially by the direction from the start point of a scan to the direction point of the scan.

Boundary Points
These are sample points which will determine the limits of the area to be contained within a scanned line or surface.

Box Tolerancing
Box tolerancing is a variation of box-selecting. Box tolerancing can be performed with an open Dimension dialog window and box-selecting a group of measured features. The tolerance options in the Dimension Dialog box can then be used to apply the same tolerances to all of the features that are selected.

Box-Selecting
Box-selecting is a function within the Graphic Display area that allows you to draw a box with the mouse around a group of features. This highlights those features in a Feature ID list box.

Branching
Branching is the path (not the probe path but the logical path or flow that an executing measurement routine takes) followed during execution of a measurement routine which may be variable as a result of some logical statement, such as IF, ELSEIF, ELSE, DO, and WHILE, that causes the measurement routine to execute differently depending on the value of some input or variable that is computed or measured in the measurement routine.

CAD Coordinates
The position, expressed as X, Y, and Z values, of a feature in a CAD file.

CAD file
A CAD file is a graphical image file created using Computer Aided Drafting which contains information about a part or parts and their features' dimensions, orientations, and sizes. These files are usually saved in one of the following formats: IGES, DFX, DES, STEP, and XZYIJK.

Cal Wrist

Abbreviation for Calibrate Wrist. Used to calculate the probe head offsets for each wrist position that is used.

Calculate Boundary
Abbreviation for Calculate Boundary Points. Used with Perimeter Scans to determine the limits of travel during the scanning operation.

Calibration
This process simply tells PC-DMIS the location and diameter of the probe tip. It is also referred to as "qualification".

Calibration Tool
A Calibration Tool is a calibrated artifact used to qualify a probe tip. It is normally a precision tooling ball. It is also known as a "Qualification Tool".

Capability
This is a calculated number based on measured data and the tolerances (USL and LSL). It tells you how well a process can produce parts with respect to the specified tolerances.

Capability in Equation Form
It is the minimum between: (USL-XBarBar)/(3*sigma) and: (XBarBar - LSL)/(3*sigma). Where XBarBar is the mean of the subgroup's means for the data in consideration. In order for the process to be labeled as "capable", the calculated capability (above equation) must be greater than the user entered capability threshold. If a process is not capable it is considered "incapable".

Capability Threshold
This is the number by which the process's capability is determined. The capability and the capability threshold are two different values.

Car Body Alignment
Most Car Body (and Aircraft) alignments have a coordinate system that is off in space. In the case of a car, it is usually down in the lower center of the car, under the instrument panel. Dimensioning of features takes place from that point.

CD
Collision Detection.

Centroid
The center most point of a feature.

Characteristic IDs
Circular labels on a blueprint next to a GD&T callout. Also, referred to as "Balloon IDs".

Characteristic Point
This is a measurement used by the NC-100 video probe that is essentially an Angle Point measurement. It has two measurement types.1) Measures a point at the intersection of two surfaces (this measurement type is most like an angle point).2) Measures two points, one on each surface at a specified distance from the intersection.

Child
A 'child' is a command that is dependent upon another command in order to function.

Circ Move
Circular Move.

ClearP

An abbreviation for Clearance Plane. It is also a command entered in the Edit window to define the plane the probe must clear to avoid collision with the part.

Cmk
Machine capability process. This value is used to determine how well a given process can match the specification limits for the process.

CMM
Coordinate Measuring Machine.

Collision Tolerance
Indicates a measurable value from the surface of the CAD data. It is designed to alert you to any collisions between the probe and part.

Column locator
Another term for "Gripper".

Column passing lane
The column passing lane is a small corridor behind the usable fixture volume where columns are moved from one arm to the other and pulled from the rack for placement on the machine table.

Comm
Communications port.

Continuous Contact Scanning
A Continuous Contact Scan can be performed only when using either an analog probe, hard probe, or some laser probes and optical probes. In this type of scan the probe tip is brought in contact with the surface of a part and moved in a linear fashion without leaving the surface of the part until the scan or a portion of the scan is complete.

Controller
Every CMM has a controller. The controller drives the servos to move the machine, reads the scales to keep track of the position, interfaces to the actual probe, etc. A given CMM (or machine) can have any one of several different types of controllers.

Coordinate System
A coordinate system consists of an origin and X,Y, and Z axes.

Coplanar
Coplanar means that features reside in the same plane.

Cpk
Process capability index. This value is used to determine how well a given process can match the specification limits for the process.

CTE
Coefficient of Thermal Expansion

Cut Plane
The Cut Plane is a theoretical plane which is created perpendicular to the Cut Plane Vector with the same XYZ coordinates as the start point of the scan. The Cut Plane is used to enable a scan to be created with all points being at the same level relative to the Cut Plane Vector.

Cut Plane Vector
The cut plane vector is the cross product of the initial touch vector and the line between the start and end boundary points on scans. If there is no end point, then the line between the start and direction point is used.

CutVec (Cut Vector)
See Cut Plane Vector.

DataPage
DataPage(tm) is a statistical processing software package which links seamlessly with PC-DMIS.

Datum
A datum is an "imaginary" and "perfect" user-defined feature used as a reference point from which to measure other features on a part.

Datum Reference Frame
A datum reference frame is an alignment created from the current set of datums. It is defined by the order of your A, B, C, etc. datums.

DCC
Direct Computer Control.

DCC Mode
This mode puts the Coordinate Measuring Machine into Direct Computer Control (or DCC). With this mode turned on, the computer is in control of many CMM functions.

DCI
Direct CAD Interface

DCT
Direct CAD Translator

DD
Diameter of Datum.

Delimiter
The verb, to delimit, simply means to separate pieces of information. A delimiter then is a symbol that separates distinct pieces of information. In written English for example, the period, question mark, and exclamation mark are delimiters that delimit (or separate) the sentences.

Delta
Used to indicate an option where change or an amount of change is expected to occur.

Dependent curve
A Dependent curve is dependent on the feature set it was created from. This means if the feature set is changed later on, then the constructed curve may also change.

Dependent surface
A Dependent surface is dependent on the feature set it was created from. This means if the feature set is changed later on, then the constructed surface may also change.

Depth
This defines the distance below the surface of a part where the measurement will be taken.

DES
If importing a DES file type, this stands for "Data Exchange Standard". If working with statistical databases, DES stands for "Data Evaluation System".

DF
Diameter of Feature.

DFL

Deflection mode

Diameter
The maximum chord length of a circle, cylinder, or sphere. In the case of PC-DMIS, the chord length will be applied to a best fit feature unless specified otherwise. Imagine a round feature with a line passing through the center of it. The length of this segment is referred to as the diameter. It is sometimes referred to as the thickness or width of the feature.

DIMS
Dimensional Inspection Measurement System file format. This is the format of the PC-DMIS measurement routine files.

Direction Point
The Direction Point of a scan sets the direction of the scan. The scan will continue from the Start Point heading in the direction of the Direction Point until it reaches the End Point.

Disk Tip
Tip shaped as a small disk.

DMO
A DMIS Output file (.dmo). PC-DMIS can save a measurement routine's results to this file in the DMIS output format.

DOF
Degrees of Freedom.

DRF
Datum Reference Frame

DRO
Digital Read Out.

DSL
This is an abbreviation for a Digital Scanning Laser probe assembly.

DSN
Database Source Name. This is the database name for a database created inside ODBC or inside of DataPage+.

DST
This is an abbreviation for a Digital Scanning Tactile probe assembly.

DTT
This is an abbreviation for a Digital Touch Trigger probe assembly.

DXF
Drawing Interchange File.

Element
An element is simply another name for a feature.

ELM1
This abbreviation stands for Element 1.

ELM2
This abbreviation stands for Element 2.

ELOGO.DAT
This data file is used to format the Edit Report's footer. It is only used on the very last page of the Edit Report.

End Ang
Ending Angle.

End effector
An artifact that is placed against the part to hold it stationary in at least one axis. Typically, this is a sphere with a defined diameter. Spheres are used because they will only contact the part at a single location. Other types of end effectors can be pins (for mating holes), cylinders for locating edges, and specialty pieces specifically tailored to the part being fixtured.

End Point
This is the Ending Point of a scan. When the scan execution reaches this point, it stops.

End Touch Vector
The compensation vector for the last point of a scan.

End-of-ram
This term refers to the XYZ coordinates at the end of the arm (or ram) only, not figuring in the XYZ coordinates of the probe tip.

EndVec (End Vector)
See End Touch Vector.

EOF
End of File

Error Map
All CMM's are constructed with inherent inaccuracies. After construction, most CMM manufacturers laser verify their machines to record these inaccuracies. These errors are then stored electronically in a computer file (called abcomp.dat) which can be accessed by PC-DMIS to enhance the accuracy of the CMM. In essence, the computer file is an Error Map of the CMM. See Volumetric Compensation.

Export
The process of converting the drawing information contained in a PC-DMIS measurement routine to a CAD standard output file such as IGES.

Expression
An expression is a user defined condition used with PC-DMIS'S flow control commands. If the condition is met or not, you can determine what action PC-DMIS takes.

Extrude
When a probe is moved from one location to another, geometric volume is created through the move. The term "extrude" is used to define the volume of space that a probe will be in as it moves from one point to the next. The volume is then checked for collision with the part or table.

Faro Arm
This refers to the articulating arm CMM produced by Faro Technologies, Inc.

FBM
Feature Based Measurement

FCF
Feature Control Frame

Feature Height
The distance from the base to the top (center) of the feature.

Feature Length
The measured value along its greatest dimension.

Feature Pointer
A Feature Pointer is a variable type that works directly with an existing feature, providing direct access to that feature. For example, the statement ASSIGN/V1 = CIR1 would create a feature pointer to the feature CIR1 and assign it to variable V1. V1 could then be used to access CIR1. CIR1.X would access the measured x component of the centroid of CIR1.

Feature Set
A Feature Set is a collection of previously measured or created elements into one constructed element. The items making up the Feature Set do not necessarily have to be of the same feature type.

Feature Width
The measurement of a feature from side to side. The measured value along a feature's shortest dimension.

FILE I / O
This menu option stands for File Input / Output. Data can be Input (written) to or Output (read) from these files.

FindNoms
Find Nominal Values. When this option is selected from the Nominals option of a scan dialog, PC-DMIS will pierce the closest selected CAD surface in an attempt to gather the theoretical CAD data for each scanned data point. This information will then be used to calculate the deviation of each point.

Fixed Delta
An option for manual scanning with a hard probe where data points will only be gathered at specific, or 'fixed,' increments between each other.

Fixed probe
A fixed probe is synonymous with a "hard probe". It is a probe that doesn't use a touch trigger.

Flat feature type
Flat feature types are planar features. See Flat features.

Flat features
Flat features are defined as points, planes, and lines. See Flat feature type.

Flat Guess
This is a mode of operation utilized when using a hard probe to set order of precedence for the software to guess the type of feature being measured. Flat Guess will attempt to solve for a plane before a circle, cylinder, cone, or sphere. See also Round Guess and Guess Mode.

Floating toolbar
A floating toolbar is a toolbar that is able to be moved out of its set position and allowed to "float" at a desired location on the screen.

Flow Control
This refers to options within the software that allow you to control the direction of the measurement routine.

Fly Mode

This is an option referring to how PC-DMIS will move the probe around the part during the measurement process. To be utilized, the CMM must have a DCC controller capable of Fly Mode operation.

FOV

Field of Vision or Field of View. This refers to what you can see through the eye of a camera, an integral portion of video probes.

Gage

A virtual or physical tool used to perform measurements on a feature.

Gage Diameter

A circle feature that you construct at a given height along a cone. See also "Gage Point".

Gage Point

A point used on a tapered surface or cone where you define a height value (Z) in order to check a point location (X and Y) on the cone; or, where you define a diameter value to check the size of a cone at that diameter; or, where you define a point location (X and Y) to measure the cone's height (Z) at that location. In PC-DMIS, this second definition is accomplished when you construct a Circle feature from a Cone by specifying a height, diameter, or sphere value (for more information, see "Constructing a Circle from a Cone").

Gap & Flush

GAP = Distance (in the same plane) between two mating sheet metal parts. FLUSH = Height difference between two mating sheet metal parts. For example, if you're looking at a top view of an automobile, the distance between a car's fender and hood is the gap (distance between the two parts in one plane). If you look at a side view of the car the flush for the fender and the hood would be the height difference.

Garage

Locations in the probe rack that hold pieces of the probe assembly. Garages are also known as "ports" or "slots".

Gauge

See Gage.

Gaussian Filter

See 95% Confidence factor.

GD&T

Geometric Dimensioning and Tolerancing. This is a standardized, international language that uses a recognized symbology to communicate a part's design specifications.

Geometries

Geometries are features or geometrical shapes such as lines, circles, etc.

Global setting

A global setting is a statement or command that is added to the measurement routine that remains in effect for the remainder of the measurement routine unless it is changed by another modal command.

Gripper

This device, used with flexible fixturing, is placed on the end of the ram. It "grips" (or holds) columns in order to either remove them from their rack and position them on the worktable, or remove them from the worktable and place them back in their rack.

The gripper also uses air injectors (much like air hockey) to lift the column off of the worktable.

Guess Mode
When a fixed probe is used, PC-DMIS can't determine the type of feature that is being measured. Guess Mode allows you to indicate whether the feature is flat or round reducing the possibility of an incorrect guess.

Hard probe
A hard probe is a probe without a touch trigger or removable stylus.

HD
High Definition or High Density.

HEADER.DAT
This data file is used to provide formatting information for the Edit Report's headers. This includes all the pages after the first page.

High Point Feature
A High Point Feature is the type of feature created when using the High Point option of Auto features. (From the main menu Access the Auto features dialog box and then select the High Point tab) The High Point is the point along a specific vector through a surface which has no other point at a higher distance along that vector when compared to the centroid of the surface.

Hit
Contacting the stylus tip with the part. Also known as probing.

Hit Buffer
The hit buffer stores probe hits before you create a feature from those hits. The stored hits can be deleted by pressing the ALT and '-' (minus) key combination.

Hit Retract
This number is a user specified distance which tells the probe at what point to speed up again, after retracting from a probe hit on a part.

Home position
This XYZ position is the 0,0,0 machine location that the probe moves to each time the machine is powered on or off.

I/O
Input/Output

I/O channels
A numbered device in the controller that allows you to set the state to 1 or 0. Compatible devices can then be connected to each channel. One common example is to connect one channel to an air supply for an external device. Setting the channel to 1 turns on the air and setting it to 0 turns it off.

ID
Identification or Identity. Also Label or Feature Name.

IGES
International Graphics Exchange Specification file format.

Import
The process of retrieving a CAD file from a database and processing it into the PC-DMIS measurement routine.

Indent
A value set in from the edge of the feature (to the point location).

Independent curve
An Independent curve no longer depends on the feature set that was used to create it. If the feature set changes, the independent curve will not change.

Independent surface
An Independent surface no longer depends on the feature set that was used to create it. If the feature set changes, the independent surface will not change.

Indexable
The ability of the probe wrist to be positioned at certain predefined (indexed) angles. These positions are mechanically set at regular increments on indexing probe heads. These indexes can vary from 15 to less the 0.1. When a wrist is indexable it means that it has the capability of being moved to different positions within the increments available on that specific wrist.

Indirection
The value of the variable pointed to by the given variable.

Init Sample Hits
The initial sample hits are taken when learning a particular feature. Sample hits are used to determine the surface vector into which the measured feature projects.

Initial Touch Vector
The compensation vector for the start point of a scan.

InitVec (Initial Vector)
See Initial Touch Vector.

INTOF
Intersection of the given feature.

Iterative Alignment
This alignment takes a number of points which approximates the ideal (or nominal). From the points, a mathematical computation repeatedly adjusts or tries to approximate the alignment to the nominal. In a sense, it does a "best fit" computation with the points.

Kinematic Mount
A kinematic mount refers to a set of usually 3 precision tooling balls, receivers for the balls, and a fastening device which is used to provide a highly repeatable connection point for styli, probes, or probe heads. Kinematic mounts are frequently found on devices which may employ an automatic changer and will preclude the need for requalification.

Knot
A knot is a part of the mathematical definition of a spline. They are associated with the control points of a spline and help to define the shape of the spline.

Learn Mode
This term is used to describe the state of PC-DMIS when creating or appending to a measurement routine's instructions in the Edit window.

Learning
This term is often used to describe the process of creating instructions which appear in the PC-DMIS Edit window. This can be done through typing in the actual entries, selecting menu command from the menu bars, or by touching measurement points on a physical part with the CMM and pressing the jog box's DONE button or the keyboard's END key.

LEVEL - 3 +

This means that three or more features are needed to perform the Level command for an iterative alignment.

LEVEL set
This is the set of features needed to perform the Level command for an iterative alignment. See LEVEL -3 +.

Line Distance
The length of a line segment between two features.

Literals
Operands whose symbols literally describe their value. "3" is a literal of type integer. "3" means the number three. It never means the string character â€œ3â€ or â€œthreeâ€. A variable, such as "V1" does not literally denote its value, rather it is a label or holding place for a value. "V1" could hold a value of "2", "3", or "4" or any other number of object type. Literals often have very specific functions and meanings and at times can be used in place of arguments. See also "String Literals".

LK systems
LK systems are CMMs or CMM controllers produced by LK, as opposed to SHARPE models.

LMB
Least Material Boundary

LMC
Least Material Condition.

Logo.dat
This data file contains information that formats a document header for the first page of the Edit Report. It may contain information such as date, time, etc.

Looping
The act of repeating any portion of the measurement routine a predetermined number of times.

Machine Capability
This is the same as the process capability (see Capability Threshold), except standard deviation is calculated in a different way. It this case standard deviation is calculated based on the individual data, not subgroup data. This is essentially the only difference.

Machine Coordinates
The position, expressed as X, Y, and Z values, of a feature or object within the measurement volume of a CMM with respect to the machine's zero reference position.

Manual Probing Point
This refers to the fact that the operator manually moves the machine when taking the hit rather than the machine moving itself in DCC Mode.

Mark Set
This highlights a group of features that will be measured when the measurement routine is executed.

Mark Used
The Mark Used function (selected from the Probe Utilities dialog box) will scan a PC-DMIS measurement routine and mark or highlight the probe angles that are used or referenced by the measurement routine. This will then allow an operator to easily determine which probe angles or tips need to be qualified.

Master Arm Mode

This mode allows one arm, designated by the user as the "Master", on a multi arm CMM, to take precedence over another arm, designated as "Slave", in order to avoid collision of the arms. This is also known as Master / Slave mode.

Material Thickness

Material Thickness is a property which can be associated to CAD files. Many times, especially on parts constructed of sheetmetal, a CAD file will represent only one side of the material. Therefore, in order to accurately measure and dimension on the opposite side of the part, a correction thickness must be applied.

MCR

Module Change Rack

MDI

Multi-Document Interface. This is a program or user interface that allows more than one file open at a time (PC-DMIS for example allows you to have more than one measurement routine open at a time, making it an MDI application).

Measurement Routine

The measurement routine (or part program as it used to be called) is a file with textual commands and descriptions of how to measure or inspect an object. Each measurement routine has a unique name with a .prg file name extension. The measurement routine is created by a CMM programmer and is sometimes executed or ran by someone other than the person who created it. The measurement routine can be, but is not always, associated with a CAD model. If it is associated with a CAD model, the CAD file has the same file name as the measurement routine with a .CAD extension.

Measurement Volume

The measurement volume is the area in the machine that can actually be used for measurement. Although the machine dimensions may be large, the actual space that is available to use for measurement can be much smaller. The measurement volume can increase or decrease depending on the probe configuration (type of wrist, probe body & stylus) being used.

Meta-Character

A meta-character is a character that acts as a wild card for another alpha-numeric character or characters. There are two meta characters available in PC-DMIS: the Asterisk (*) meta-character and the Question mark (?) meta-character.

Microns

A micron is a unit of measurement, one millionth of a meter.

Minus-Lower tolerances

Gives a plus tolerance in the Minus Tolerance field. (i.e. 1.000 + .003 / +.001).

MMB

Maximum Material Boundary

MMC

Maximum Material Condition.

MMIV

This stands for the Micro Measure IV(tm) measurement software package.

MOD

Modular.

Model-space

This is the 3D coordinate space to which surface and curve geometry is mapped.

Modes
Modes are different program states available to PC-DMIS. Each mode offers specific functions. The modes available to PC-DMIS include: Program Mode, Translate Mode, DCC Mode, Manual Mode, and Text Box Mode.

MOHL
Mouse Over Highlighting. You can hover your pointer over CAD elements to highlight them.

Mount point
A defined XYZ location used during certain fixture and probe changer operations. The location is away from the rack, where the machine can move before and after the change cycle. It is a safe point with direct access from all the slots as well as to and from the measurement routine itself. When using multiple racks, the probe would also need to be able to travel from one mount point to the other without colliding with a rack, the part, a clamp, or some other obstruction.

Movement Range
The actual space (volume or area) that can be used on a machine for measurement. It is also known as Measurement Volume.

Moving Average
This is the average value of a set of data points that moves over time. Example: Given a subgroup size of 3 (the number of data points used in calculating a moving average) and the following set of data: [1,2,3,4,3,4,5,4,3,2,3] the first moving average is calculated from the first three points: [1,2,3]. Its value is 2. The second moving average is calculated by moving over one step to the right within the data, giving the following data: [2,3,4]. Its value is 3. The next 3 points are: [3,4,3]. Its average is 3.33. The next 3 are: [4,3,4]. This is done up to the end of the data.

Mrad
This is an abbreviation for milliradians. A milliradian is an angular distance of one thousandth of a radian.

MSE
Measurement Strategy Editor. This editor allows you to save groups of settings for each Auto feature.

Nested subroutines
These are subroutines which are called from (or nested within) another subroutine.

Newtons
A "newton" is a unit of force. One newton is the force needed to accelerate a mass of 1 kilogram to a speed of 1 meter per second per second.

Normal To
The phrase "normal to" a certain feature or geometrical element, simply means "perpendicular to" or at a 90 angle to some other feature.

Notch
A feature type which is similar to one-half of a square ended slot.

OCR
Optical Character Recognition

ODBC
Open Database Connectivity

Offset Feature

PC-DMIS allows three types of Offset Features, Offset Points, Offset Lines and Offset Planes. An Offset Feature is constructed from one or more other features. Each feature comprising the Offset Feature may then have an offset or correction value associated with it. Thereby, each feature contained in the offset feature may or may not actually coincide with the new constructed feature.

OLE

This stands for Object Linking and Embedding.

On Error

On Error is a function of some CMM controllers where if an electronic probe contacts or doesn't contact a part unexpectedly, the PC-DMIS measurement routine will branch to a separate set of instructions to be executed. This function may be called from the main menu by selecting the On Error menu item.

Open GL

Open Graphics Library. This refers to a library of graphic routines used to aid in the display of graphical information.

Operand

The part of an equation that is being operated on by an operator. In the equation "2+3", the numbers 2 and 3 are operands and the plus sign (+) is the operator.

Optical Probe

An Optical probe is a probe that uses optics to determine its location. For example, a laser probe would be considered an Optical Probe.

ORIGIN set

This is the set of features needed to perform the Set Origin command for an iterative alignment. See SET ORIGIN - 1.

Outliers

Outliers are "wild points" and usually result from glitches in the measuring process rather than representing actual material. You may find it useful to identify and ignore such points.

PA

This abbreviation stands for Polar Angle. Also seen as PANG. The Polar Angle is used in cylindrical coordinates and is used with Polar Radius. See PR.

Pang

This abbreviation stands for Polar Angle. Also seen as PA. The Polar Angle is used in cylindrical coordinates and is used with Polar Radius. See Prad.

Parameter-space

This is the 2D coordinate space of a surface's parametric domain. For example, one corner of a surface is defined at parameter position (0,0) and the opposite corner is at (1,1). Varying this parameter between these two positions defines the entire surface. Given the geometric definition of surface, a parameter-space position can be mapped to a model-space point.

Parent

A 'parent' is a command (or information from a command) used in another command.

Part Coordinates

The position, expressed as X, Y, and Z values, of a feature on a physical part with respect to an origin location.

Part Program

The old name for a measurement routine. See Measurement Routine.

PC-DMIS
The term "PC-DMIS" is derived from the acronym "DMIS", which stands for Dimensional Measuring Interface Standard.

Perm
Abbreviation for permanent. See Perm Sample Hits.

Perm Sample Hits
The number of sample hits which will be used in measuring a particular feature during execution of a measurement routine.

PH9
This is the model number of a particular Renishaw probe head. The abbreviation stands for Probe Head 9. This term is also used to refer generically to articulating motorized wrist type probe heads.

Pierce Point
An intersection point found on a CAD surface using the measured hit coordinates and approach vector. It is equivalent to a ray that uses the approach vector and starts from the hit's XYZ location and then, using the approach vector, pierces the surface at an appropriate point.

Pins
Pins are removable features that stick out of a part (as opposed to holes which go into the part). Pins are similar to "studs".

Pitch
Pitch is the distance moved along the axis of the feature in one revolution.

PlaneVec (Plane Vector)
See Boundary Plane Vector.

PMI
This is an abbreviation for the term "Product Manufacturing Information" and relates to design, manufacturing, quality or any other information that is included with 3D CAD model files.

PMM
This is a CMM brand manufactured by Leitz.

PNT TOL
This abbreviation for vector point tolerance is used in the Edit window (as PNT TOL = n) to specify the vector point tolerance for manual touches in an iterative alignment.

POI
Point of Interest

Point to Point Digitizing
Point to Point Digitizing is a process of gathering the data about the surface of a part for reverse engineering purposes. Data is gathered by scanning the part with a machine and software capable of generating discrete measurement points. Once a sufficient number of points are gathered, they are taken as a group and processed to generate the electronic surfaces of a part which can be used by CAD software to complete a design.

POLAR
Refers to a Polar (also seen in the Edit window as POLR) coordinate system containing U and V coordinates. U represents the polar radius and V represents the polar angle or vector.

POLR
An abbreviation for Polar.

Polyline
In computer graphics, a polyline is a continuous line made up of one or more line segments. A polyline is sometimes treated as a single object, or it may be divided into its component segments.

Port
Locations in the probe rack that hold pieces of the probe assembly. Ports are also known as "slots" or "garages".

Portlock
A hardware lock that connects to a USB or your printer port that is programmed to allow PC-DMIS and specific purchased options to run. PC-DMIS will not run on your computer unless you have a valid portlock attached.

Pose
The combined position and orientation of a rigid body or coordinate system: three translations and three rotations.

Potentiometers
These are instruments that measure electromotive forces.

PPAP
PPAP (Production Part Approval Process) defines requirements for production part approval.

PR
Stands for Polar Radius. Also seen as Prad. The Polar Radius is used in cylindrical coordinates with Polar Angle. See PA.

Prad
Stands for Polar Radius. Also seen as PR. The Polar Radius is used in cylindrical coordinates with Polar Angle. See PA.

Precedence
Precedence in algebraic expressions or assignment operations refers to the order in which operations occur. For example, multiplication has a higher precedence than addition therefore, the expression "2 + 3 X 6" evaluates to 20 since 2 is added to the result of 3 X 6. Not following precedence might result in the errant result of 30.

Pre-Hit
This is number which determines at what distance a probe will reduce speeds while approaching a part to take a hit.

Probe
The sensing unit that is attached either to the Ram of the CMM or the indexable wrist. The probe requires a stylus be mounted to it in order to gather measurement data (except in the case of a hard probe).

Probe Changer
A rack placed in the measurement volume of the CMM. The probe changer secures multiple probe types for use in a single measurement routine. The CMM can switch between probe types without the need of operator assistance.

Probe Depth
The Probe Depth is the location of the probe along the Z axis, normal to the CMM's table.

Probe Extension
A Probe extension is a cylindrical piece that is inserted between the probe head and the probe itself.

Probe Shanking
This is the accidental contact of the probe's shank with the part or a feature on the part.

Probing Cycle
This is the cycle of operation an analog probe/controller goes through to take a point.

PROE
ProEngineer CAD file format.

Profile
"A profile is the outline of an object in a given plane (two-dimensional figure). Profiles are formed by projecting a three-dimensional figure onto a plane or by taking cross sections through the figure."-ASME Y14.5M-1994 Dimensioning and Tolerancing. Often on CAD drawings, when you look at a surface in different views, it won't look like a surface, but a curve, line, or a collection of individual points. These are the result of this projection of a three-dimensional figure onto a plane or by taking cross sections through the figure.

Profile Error
This is the deviation of the actual or measured profile from the theoretical or nominal profile

Program Mode
This mode allows you to create your measurement routine in the Edit window. It also shows the image of the probe in the Graphical Display window.

Pt
Abbreviation for Point.

Qualification Tools
See Calibration Tool.

Qualify
To qualify also means to calibrate, or determine the position (of a probe tip, for example).

Question Mark (?) Meta-Character
The question mark meta-character (?) acts in the same way as the asterisk (*), except that the question mark meta-character will match only 1 alpha-numeric character.

Radian
A radian is a unit of angle measurement such that an angle of one radian has an arc length along the edge of the circle equal in length to the radius.

Ram
This refers to the end of the arm to which the probe is attached. On a horizontal machine, the Ram is the horizontal beam on the machine and usually moves in the X or Y axis of the machine. On a vertical machine (commonly called a "bridge" machine) the ram is mounted vertically and moves in the Z axis of the machine.

Read POS
Read Position. This function will get the current XYZ coordinates from the CMM controller.

RECT

Rectangular. Used where the referenced coordinate system should be expressed in rectangular (XY) or Cartesian (XYZ) form.

References
An expression syntax that refers to the value of data. All of the syntax used to access different data elements of the measurement routine. "CIR1.X" is a reference to the measured x value of the centroid of the feature called CIR1. "LINE.LENGTH" is a reference to the length of LINE. "C1.INPUT" is a reference to the input of the comment named C1.

Relearn Mode
This is a scan option for execution of a scan. If the Execute option is selected to RELEARN, then every time a scan is re-executed the compensation vectors for each scan point are recalculated.

RFS
Regardless of Feature Size.

RLE
RLE stands for Run Length Encoding and is a way of compressing bitmap files.

RMB
Regardless of Material Boundary

Rmeas
Relative Measurement. Can be used to create a new Auto feature at coordinates which are derived from the actual measured coordinates of a previously measured feature.

ROI
This is a computerized box displayed on the computer screen for the NC-100 video probe. Measurement of the feature must be within the box.

ROTATE - 2 +
This means that two or more features are needed to perform the Rotate command for an iterative alignment.

ROTATE set
This is the set of features needed to perform the Rotate command in an iterative alignment. See ROTATE - 2 +.

Round feature type
A round feature type is a circular or curved feature. See Round features.

Round features
Round features are defined as circles, spheres, cones, and cylinders. See Round feature type.

Round Guess
This is a mode of operation utilized when using a hard probe to set order of precedence for the software to guess the type of feature being measured. Round Guess will attempt to Solve for a circle, cylinder, cone, or sphere before a plane. See also Guess Mode.

RTF
Rich Text File.

Scan Point Density
The Scan Point density determines the number of hits that the CMM returns for each millimeter.

Scripting
See Basic Scripting.

Section lines
Section Lines are CAD entities that can be superimposed on a CAD drawing. Though they contain no dimensional or drawing information, they can be referenced by a Section scan to provide a linear scan across one particular section of a part. These are reference lines used by engineers when specifying locations of part features.

SET ORIGIN - 1
This means that one feature is needed to perform the Set Origin command for an iterative alignment.

Setting Gage
This is a piece of hardware used to calibrate the ACR3 Probe Rack. The setting gage is inserted into the specified port on the probe changer. You would then use the setting gage to take hits on the port without having to use the probe body.

Setting Gauge
See Setting Gage.

SFT
Soft probing cycle

Shank
This is the shaft portion of the stylus that extends from the measuring tip to the stylus mounting joint. In the case of a shank tip, there is no precision tip (see shank tip). The shaft which supports the qualification tool is also referred to as a shank.

Shank Qual
Shank Qualification is utilized with sheetmetal styli to provide the axis direction of the shank as well as the center point of the spherical tip end.

Shank Tip
Tip shaped like a straight shaft.

Shanking
A term used when the probe is triggered by contacting the part with the shank of the stylus as opposed to the tip of the stylus. When this occurs, erroneous data is gathered.

Sheet Bodies
This is Unigraphic's modeling engine's terminology for surfaces.

Sheetmetal Stylus
A sheetmetal stylus is designed for measuring very thin part. The sheetmetal stylus will have semi-spherical ball on its end with a short section of shank which is coaxial and concentric to the center of the ball.

Sheetmetal Thickness
See Material Thickness.

SHSP
Stylus Holder Setting Piece

Sigma
The Greek letter often used to represent standard deviation.

Slot
Locations in the probe rack that hold pieces of the probe assembly. Slots are also known as "ports" or "garages".

SNSDEF
This statement is a DMIS major word used to defined sensors in the DMIS language.

Soft Probing
Soft Probing (or SFT) is Leitz terminology for a frequently used probing mode used to measure soft materials.

Solid Bodies
This Unigraphic's modeling engine's terminology for solids.

Spacer
A user defined value indicating the offset distance between the edge of a feature and a sample point.

SPC
This stands for Statistical Process Control. In PC-DMIS SPC charts can be created and updated from a defined database.

Spline
A spline is a specific type of curve or surface. Splines are used in modeling because of their mathematical properties which make them easier to use in a wide variety of ways. Splines can be used in both curves and surfaces.

Square-up
"Square-up" means to align (or rotate) the part in the Graphical Display window until that part lines up and is parallel to an edge of the screen.

Standard Deviation
This is the root mean square of the deviation from the mean. In equation form it is this: sqrt((sum((xi-m)^2))/n-1) where: m = the mean value n = the number of data points xi = the ith data point.

Start Ang
Starting Angle.

Start point
The Start Point in a scan is the beginning point of the scan's execution.

State Mode
The four states of operation are Translate Mode, 2D Rotate Mode, 3D Rotate Mode, and Program Mode. These four options can be individually selected from PC-DMIS'S toolbar icons.

Statistical Output File
This file contains printable stats for a measurement routine. Information is saved in the file named "xtats11.tmp" usually in the PCDMIS directory or the location that you chose to install PC-DMIS.

STEP AP203 & AP214
STEP stands for Standard for the Exchange of Product Model Data. STEP was forwarded to the international community in 1988. It is a standard for the computer-interpretable representation and exchange of product data. The objective of STEP is to provide a neutral medium capable of describing a product throughout its life cycle.

Step Mode
Step Mode is a variation in execution of a measurement routine in which the CMM will execute only one block of commands at a time. User intervention is required for the program to continue. In effect, this is 'stepping' through the measurement routine.

Stitch Type Scan

This is the type of scanning performed on DCC CMMs which have Touch Trigger Probes. During the scan, the CMM will contact the part, move away the retract distance, move to the next prehit point, then move in and contact the part. This process continues to repeat until the scan is complete. The term Stitch Type comes from this process which is similar to the functioning of a sewing machine.

STL

Stereolithography format. An ASCII or binary file used in manufacturing, provides a list of the triangular facets that describe a computer generated solid model.

Studs

Studs are non-removable features that stick out of a part (as opposed to holes which go into the part). Pins are similar to "studs".

Stylus

Also referred to as "Probe Stylus" is a cylindrical shaft (shank) with a high precision tip. The stylus screws into the probe body and is usually interchangeable with other Styli (plural for stylus).

Stylus Changer

See Probe Changer.

Stylus Extension

A Stylus extension is a cylindrical piece that attaches to the probe body and to the stylus. It is used to extend the length of the measuring tip. Multiple extensions can be used at one time.

Stylus Holder

The Stylus Holder is a part of the probe assembly that the software drops off or picks up from a probe changer. It is the component that the system exchanges within the probe changer. There are many places in the probe changer calibration process where the software prompts to make sure the stylus holders are removed from the probe changer ports before PC-DMIS proceeds with the measurement of the probe changer. This is necessary in situations where the ports need to be empty so the system can reach them during the measurement process.

Subroutine

A subroutine is a sub-program within the measurement routine. These sub-programs contain commands from the current file or an external file and that are able to be used repeatedly.

Surface of Revolution

This is a surface which may be generated by rotating a curve plane about an axis in its plane.

System Calibration

This is the process of checking the accuracy of a CMM and making adjustments mechanically to the CMM or electronically to the VolComp file to reduce the inaccuracy.

TAPER TIP

Tip that has been tapered to a conical shape which gets smaller toward the end.

TARG

This stands for target. It is the size or location information at which the CMM will try to measure.

Tessellation

The act of breaking up a surface into a group of polygons in order to shade a graphical image.

THEO
This stands for theoretical. It is the nominal size or location information.

Thread
The portion of the tip that is screwed into another part is referenced by the size of thread. The tread is the corkscrewed band that holds the parts together.

Time Delta
This user specified value is used to weed out hits taken with manual scans using a hard probe that come in faster than the specified time increment. The increments are in seconds.

Tip
This is the portion of the stylus that comes into contact with the part being inspected/measured. Styli may have several types of tips. The tip can be a ball, disc, or a pointer.

Tip Changer
See Probe Changer.

TKJ Pin
This refers to the Tesa Kinematic Joint pin. A wrist that has the TKJ connection has a nub that the TKJ locks onto. The "nub" is the TKJ Pin.

TOG
TOG is a "toggle" field in the Edit window. To toggle through the available options within the Edit window, place the cursor over toggle field and press F7.

Tool Changer
See Probe Changer.

Touch Trigger Probe
This type of probe automatically triggers a hit when it touches the part.

TP
Acronym for the old "True Position" term. Now known as "Position" in the software.

Tracefield
This two field command is used in PC-DMIS's Edit window. The first field represents the name or title of the tracefield. The second field displays the current value of the tracefield (such as "Temperature : 75" or "Shift : afternoon"). Tracefield data is often included with measurement data when a measurement run (or transaction) is sent to a database.

Translate Mode
This places PC-DMIS into a program state which allows you to move the part on the screen as well as change the part's display size in the Graphic Display window, etc.

Trihedron
A graphical representation of the planes meeting and forming a point. Also termed a trihedral.

TTP
See Touch Trigger Probe.

T-Value

The distance of the measured hit from the nominal point along the nominal CAD surface vector. In technical terms, it is the combination of the difference in the 3 axes for the hit, essentially the square root of the sums of the squares.

UG

UniGraphics CAD file type.

UPR

This stands for Undulations Per Revolution.

Variable

An object used to hold a value. A variable has a name and a value. The name is used to access the value of the variable. The name is constant, the value can be changed. For example, the statement ASSIGN/V1 = 2 creates a variable with a name of V1 and a value of 2. ASSIGN/V2 = V1 + 2 accesses the value of V1. If V1 still had a value of 2 when this assign statement was executed, V2 would then have a value of 4. A variable is a value that refers to integer, real, string or point operands.

Variable Delta

Variable Delta is a type of scan that can be accomplished only with a hard probe. During a Variable Delta scan, data points are read only when the requirements of a specific minimum time increment, and a minimum distance increment are achieved.

VDAF

VDAF stands for VDA Surface Data Interface format. VDAFS was published as a German national standard in 1986. It is used by the German Motor Manufacturers Association (VDA) to exchange 3D CAD model data.

VDAFS

See VDAF.

View ID

View ID is a user defined name for a view that the user has saved in memory.

VolComp File

The "comp.dat" file. For a CMM to measure accurately, this file must be located in the PC-DMIS operating directory of the computer used to operate CMM's which are Volumetrically Compensated. See Volumetric Compensation (VolComp).

Volumetric Compensation (VolComp)

Volumetric Compensation is the process of accessing a CMM's Error Map and applying the error as a correction factor to measurements in order to enhance the accuracy of the CMM. This is accomplished automatically by PC-DMIS once a CMM has been properly initialized. (See also Error Map.)

Weight

A value calculated from a tolerance used when calculating a best fit. The smaller the tolerance the less room there is for error. A smaller tolerance, then, carries a greater "weight" or importance in a best fit calculation.

Wild Cards

Wild Cards are made up of two meta-characters, the asterisk (*) and the question mark (?). The asterisk meta-character (*) will match, or take the place, of any character or characters in a search. The question mark meta-character (?) acts in the same way as the asterisk (*), except that the question mark meta-character will match only 1 alpha-numeric character.

Wire Bodies

This is Unigraphic's modeling engine's terminology for wireframe features (wire, lines, curves, etc.).

Workplane

The Active and Pass Through Planes define the workplane. PC-DMIS allows you to indicate the specific plane into which a feature is projected when it is measured. The term "work" is the same as "current" or "active" when referring to the workplane.

Wrist Map

See Error Map and Volumetric Compensation.

Wrist Warning Delta

This option can be found on the Part/Cmm tab of the Setup Options dialog box. If a CAD selected feature to be measured does not have a vector which agrees with the active probe angle within the Wrist Warning Delta limit, a warning is generated.

X, Y, and Z Center

These values show the location of the center of the ball tip.

XBarR

XBar is a way of writing out the statistical symbol X which is the mean or average. Thus, the XBar chart is a mean or average of a process. This is usually calculated over time and the chart is set up such that the X axis of the chart is time and the Y axis of the chart is the mean value. XBarR is an abbreviation for "XBar and R". This is essentially an XBar chart with additional information. The "R" represents the range of data in any given sample. A sample is defined as a predetermined number of measurements. The range of a sample is calculated by subtracting the Min from the Max of the sample. This is then plotted at the same time position on the chart as the mean of the sample giving two data points for the sample at that time.

XBarS

XBar is a way of writing out the statistical symbol X which is the mean or average. Thus, the XBar chart is a mean or average of a process. This is usually calculated over time and the chart is set up such that the X axis of the chart is time and the Y axis of the chart is the mean value. XBarS is an abbreviation for "XBar and S". This is essentially an XBar chart with additional information. The "S" represents the Standard Deviation of the sample. There is a mathematical equation to determine the standard deviation of a sample.

Z rail

The Z rail is the vertical arm of the CMM.

Index

... button 3105
... Menu 48
.DAT Files 464

2

2 Dimensional Angle 1911
2 Dimensional Option 1925
2 sided lighting 550
2D Feature 264
- Circle 264
- Ellipse 264
- Enable Glow 264
- Polygon 264
- Slot 264

2D Feature Glow 264
- Circle 264
- Ellipse 264
- Enable 264
- Polygon 264
- Slot 264

2D or 3D Best Fit Alignments 1847
2D Plot 2895
- About 2895
- Scan Plot 2895
- Scan Plot:Creating 2896

2D Radius Gage 1881**,** 1881
2D
- Alignment 1852
- Angle 1911
- Best Fit Command Line 1843
- Distance 1925
- Rotate Mode 529

3

3 Dimensional Angle 1911
3-2-1 Alignment 1815
- Step 1: Measure the Alignment Features 1815
- Step 2: Level 1816
- Step 3: Completing the Alignment 1816

3D 539
- Alignment 1852
- Alignment Center of Rotation 1850
- Angle 1911
- Best Fit Command Line 1845
- Distance 1926
- Grid 480

Grid Color 539
Rotate Mode 531
3D Grid 480
3D Mouse 1256
3D PDF 199

4

4-Axis Moving 2381

A

Absolute Speed 271
Acceleration 345
Acceleration tab 340
Accessing 722
ACIS Import 149
ACR1 Probe Changer calibration 1048
ACR3 Probe Changer calibration 1062
Active
Alignment List 1803
Arms Toolbar 906
Probe 326
Rotary Table Toolbar 907
Tip List 920
ActiveX Controls 2550**,** 2613**,** 2855
Actual Measurement 1986
Actual Thickness 1380
Adapters 1062
Add Angles 1294
Add
Angle option 938
Angles 950
Arguments to subroutine 2450
Button 3016
Directory to List 2488
Features to Existing Marked Sets 696
Rotab Angles 386
Tool 934
Adding 480**,** 1801**,** 1891
Adding Items to Favorites 52
Adding Probe Changers 1315
Adjust CAD 273
Adjust Filter Construction 1781
Adjust Part Setup 273
Advanced Button - Scanning 2314
Advanced File Options 143
Advanced Scans 2307
After Choosing the Best Nominal Point 685
Aligning Probe Changers 1315
Alignment Dialog Box 1803
Alignment folder 437

Alignment Routine
Creating
Manual
Measurement
Alignment Toolbar 900
Alignments 628**,** 1797
2D or 3D Best Fit Alignments 1847
3-2-1 Alignment 1815
3D Center of Rotation 1850
Alignment ID 1803
Alignment Name 1803
Alignment Option 751
Alignment Trihedron 1803
Auto Align 1803
Best Fit 1838
Best Fit:Alignment 1838
Best Fit:Alignment Dialog Box 1852
Best Fit:Button 1803
Best Fit:Math Type 1483
Best Fit:Point Fitting 1851
Changing Nominal Values 1875
Command Format 1800
Constraints for 3D Best Fit Alignments 1847
Cylinder 846
Dependent Generic Feature 1793
Equating 1870
Export as GDS 198
Export as PolyWorks Text 198
Feature Weights 1848
Fine Tuning 246
Fixture Tolerance 1836
Force Part Alignment in Car Body 254
Independent Generic Feature 1793
Inside Loops 1863
Iterative 1803
Iterative Alignment Dialog Box 1832
Leapfrog 1870
Level 1832
Level 3 1832
New Alignment Mode 693
Origin 1833
Plane and Two Circles 846
Plane and Two Lines 846
Plane-Line-Point 846
QuickAlign 1817
Recall 1862
Recalling 1860
Rotate and Translate 1847
Rotate Only 1847
Save 1858
Translate Only 1847
Weights 1852

Alignments List 901
Allow fine tuning of Alignment 246
Always Save Stats to File 281**,** 279
Always Track FOV Center 378
Always Update Database 281**,** 279
Ambient Lighting 490
Analysis 2882**,** 2882**,** 2048
Multiplier 2882
Settings 1899
Size Options 2895
Analysis Area 1429
Analysis Object 2555
Analysis Options 2524
Analysis Window 2882
ANALYSISVIEW Command 2886
ANALYSISVIEW commands in loops 260
and Tools 805
and Translate to the Features 1816
and Transparencies 550
ANGLE Dimension 1908
Angle Offset Base Feature 338
Angle Point Field Definitions 1433
Angle Visualization 1903
Angle
Between Nominal 1910
Degrees 287
Dimensioning 1903
GD&T 842
Grid 955
Offset 338, 660, 1803
Relationships 842
Type 1911
Visualization 1903
Angles List box 951
Angles to Calibrate 386
Angular 1062
Angular Adapters 1062
Angular Definition Format 1483
Angularity 1912**,** 2078
Angularity Error by Using the ANGULARITY Option 1913
Animate Probe during Program Mode 248
Animated Probe Changer 1247
Animation Speed 304
Animation Speed for Rotation 551**,** 548
Any Order Execution 221**,** 267
Appendix
Using a Wrist Device 3229
Using Shortcut Keys and Shortcut Menus 3251
Working in Offline Mode 3183

Working in Operator Mode 3245
Applying or Removing a Report Template 2698
Archive Tab 486
ArcReductionAngle 1333
ARCSEGMENTENDINDEX function 3075
ARCSEGMENTSTARTINDEX function 3075
Argument
Add to subroutine 2450
Delete from a CALLSUB command 2458
Delete from subroutine 2450
Description of in a subroutine 2447, 2450
Edit Dialog Box 2451
Name 2450
Passing into a subroutine 2459
Value 2450
Arm1 Probe File Calibration 3136
Arm2 Probe File Calibration 3137
Arranging Open Windows 3180
Array Indices Object 3036
Arrays 3033
Arrow Size 647
Artifact 1294
ASME Y14.5 2156
ASME Y14.5 Math Types 2156
Assemblies 573
Assemblies of Parts 573**,** 3251
Assembly Component Highlight 573**,** 3251
Assembly of a Machine 584
Assign From 3018
Assign To 3018
Assigning a New Command to An Arm 3139
Assigning Existing Commands to a New Arm 3141
Assignment 3018
Asterisk (*) Meta Character 601
Auto Align 1803
Auto Circle Hits 267
Auto Continue Execution if FindHole Fails 246
Auto Create Dimensions 287
Auto Dimension Info 287
Auto Dimension Setup 287
Auto Features 1430**,** 1329
Box Mode 1335
Box-Selecting on Auto Features IDs 604
Command Buttons 1430
Creating 1431, 1335
Field Definitions for 1433
Inserting 1431
Quick Creation of 1335
Auto Features Toolbar 892

Auto Insert Moves 2412
Auto Insert Temperature Compensation in New Routine 261
Auto Line Hits 267
Auto Position CAD 273
Auto scale manual feature during execution 263
Auto Trigger area 347
Auto Zoom 359
AutoCalibrate Probe 962
Automatic Calibration 3134
Automatic File Save 88
Automatic Label Positioning 606**,** 3264
Automatic Measurements 3193
Automatically Adjust Probe Head Wrist 240
Automatically Insert Moves 2412
Automatically Scale to Fit 245
Automating PC-DMIS 23
Automation 23
AutoTrigger 347
Available Calibration Tools 933
Available Dimensions 751
Available Input Parameters 410
Average Values from a Set 1762
Avoidance Move 1433
Avoidance Move Area 2337
Axes 1965
Axes Drop Down list
Axis 2328**,** 2052

B

Back Face Culling 550
Background Color 536
Background Colors 355
Ball 930
Balloon ID Mapping 507
Basic Button - Scanning 2314
Basic File Options 79
Basic Help 807
Basic Hit 752
Basic Hits by Probing 1489
Basic Measurement Format 1468
for a Circle 1471
for a Cone 1473
for a Cylinder 1473
for a Line 1469
for a Plane 1470
for a Point 1469
for a Round Slot 1474
for a Sphere 1472
Basic Scan Dialog Box 2362

Basic Scans 2307
Basic Script 3107
- as a Command 3107
- as a Menu or Toolbar Item 3108
- Edit Menu 810
- Editor 807
- Editor Toolbar 807
- Inserting 3107

Best Fit 1498
- Alignment 1838
- Alignment Dialog Box 1852
- Constructed Circle 1622
- Constructed Cylinder 1709
- Constructed Plane 1589
- Math Type 1483
- Options for Line Profile Dimensions 1976
- Options for Surface Profile Dimensions 1975

Best Fit Alignment Methods 1840
Best Fit Recompensate 1498
Best Nominal Point 685
Blank Lines 753
Blob
- Area 1940, 2910, 3020

Body Axis
- Direction 1 of Patch Scan 2320
- Direction 2 of Patch Scan 2320
- Linear Open 2322
- Rotary 2322
- Technique 2322

Bonus 2220
Bonus Calculations 2220
Bookmark 701
- Clear All 702
- Clear All Icon 885
- Insert 885
- Next 702
- Next Icon 885
- Toggle 701

Both Option (for Position and Location) 1902
Boundary 2156
Boundary Plane Vector 2328
Boundary Points 2319
- Adding and Deleting 2319
- Boundary Points 2315
- Editing 2319
- Using the CAD Data Method 2317
- Using the Measured Point Method 2317

Bounded Line 1389
Box / Circular Mode list 1392
Box-Select Shortcut Menu 3270
Box-Selecting

ID(s) 604
Sheet Metal Features IDs 604
Brackets for Feature Arrays 296**,** 3033
Branching by Using Flow Control 2421
Branching with Subroutines 2447
Breakpoint 699
Breakpoints 698
Insert Breakpoint 885
Insert Defaults 699
Remove All 700
Remove All Icon 885
Remove Breakpoint 885
Remove Defaults 699
Brightness of Highlight 490
Building the Star Probe 917

C

CAD 628**,** 547
Alter Imported / Exported 206
Assembly 573
Coordinate System 642
Deleting CAD Elements 634
Display 477
Display:See Graphic Display Window 57
Equals Part 751
Export 196
GD&T callouts 553
Groups 514
Import History 874
Information 588
Levels 483, 509
Mesh 199
Model updates 644
Nominal Places= 282
Rotating about axes 270
Select 1342
Setup 890
Tol 2326
Tolerances 651
Toolbar 890
Transform 637
Transparency 487, 881
Verify 589
X+ Axis= 273, 270
Y+ Axis= 273, 270
Z+ Axis= 273, 270
CAD Assembly 573**,** 3251
CAD folder 437
CAD GD&T
CAD GD&T Tab 502
CAD Model Views 520

CAD Model
Showing or Hiding 883
Calculate Boundary 2327
Calculate Error Map 3236
Calculating Nominals Example 1569
Calculating Offsets Example 1570
Calculation list 1424
Calculations 2220
Calibrate 3231
a Probe 917, 932
CJoint 3241
Date and Time 943
Dual Arms 3147, 3134
Indexable Rotary Table 386
Infinite Rotary Table 388
Rotary Table 385
Star Tip 3231
the Unit 3236, 3234
Tips in Operator Mode 3249
Wrist 3231, 3236
Calibrate tab - Probe Changer dialog box 1028
Calibrating Probe Assemblies 1294
Calibrating Probe Changers 1315
Calibrating Senmation Probe Changer 1156
Calibration 1294
Calibration Artifact 1294
Calibration Mode 932
Caliper Gage 1881**,** 1881
Call Sub 2452
Call Subroutine Dialog Box 2456
Add New Arguments 2458
Edit Arguments 2458
Calling a Subroutine 2452
CALLSUB Statements in Multiple Arm Mode 2459
Calypso
Calypso Converter
Car Body Alignment 254
Careful study 1989
Cartesian/Polar Toggle 1433**,** 1375**,** 1483**,** 1932
Case / End Case 2432
Case / End Default Case 2433
Causing an Arm to Wait to Prevent Collision 3143
Center of Rotation 544**,** 1850
Center Point and Radius 2332
Center Point boxes 1371**,** 2332
Center Probe 653
Centering the Probe 653
Change Management Options 3214
Change

View 2885
Changing 355**,** 548**,** 1603**,** 1568
Changing Probes 901
Changing the Language 117
Channel 350
Character Sets 672
Characteristic IDs 261**,** 507
Showing or Hiding 883
Check Boxes 63
Check Percent 333
checking for 644**,** 3007
Circ Move check box 245
Circle Feature 1333
ArcReductionAngle 1333
Circle
2D Feature Glow 264
Auto Feature 1431
Changing the Direction 1628
Options 1929
Circular Runout 2149
Circular
Feature Type 1484
Features 1431, 3193
Mode 1392
Circularity 1915**,** 2063
CJoint Calibration 3241
Clamping Value 334
Clear All 689
Clear All Bookmarks 702
Clear All Bookmarks Icon 885
Clear Marked 885
Clear Point Move 352**,** 2396
Clearance Plane 317
Displaying 653
Example 317
Inserting 2396
Showing or Hiding 883
Clearance Plane Example 317
Clearance Plane Toggle 1406
ClearanceCube 908**,** 2384**,** 2397
Using 2397
Using with Moveset 2384
ClearanceCube uses Tip Vector for Start/End Face 258
Clearing Boundary Points 2320
Clearing Template Associated Data 2672
Clip Planes Tab 495
Clipping Plane 497
Adjusting 497
Defining 496
Close File 3247

Cloud of Points 439**,** 897
Pointcloud Toolbar 897
Cloud of Points (COP)
Showing or Hiding 883
CMM
Acceleration 340
Animated 998
Documentation 27
Multiple Arm 3122
CNC 28
Coaxiality 1917
COAXIALITY Dimension 1917
Coaxiality GD&T 1917
Collision Detection 621
Collision List 624
Collision Tolerances 944
Collisions 1252
Probe 1252
Probe Changer 1252
Protecting from 1252
Color Bar 864
Color
Assemblies 573
Background 535
Ball 2580
Change Feature Color 612, 614
Changing on report expression 2788
Changing the Report's Text Colors 2540
Defining Edit Window Colors 353
Depth 439
Edit Dimension Colors 864
Feature 614
Text Label 615
Trihedron 540
Colormap 487**,** 2585
colors.dat 464
Colors
Imported CAD Entities 460
Comm tab - Probe Changer dialog box 1027
Command 2381
Move Point 2381
Moveset 2382
Command Buttons 63
Command Line Format
2D Best Fit 1843
3D Best Fit 1845
Alignment 1800
Angle Point 1433
Auto Circle 1433
Auto Cone 1433
Auto Corner Point 1433

Auto Cylinder 1433
Auto Edge Point 1433
Auto Ellipse 1433
Auto High Point 1433
Auto Line 1433
Auto Notch 1433
Auto Plane 1433
Auto Round Slot 1433
Auto Sphere 1433
Auto Square Slot 1433
Auto Surface Point 1433
Auto Vector Point 1433
Check Distance 755
Command Format 1468
Dimension Command 1890
DIMINFO 2913
Extended Auto Circle 1433
Extended Auto Edge Point 1433
Extended Auto Round Slot 1433
Extended Auto Square Slot 1433
Extended Auto Surface Point 1433
Extended Auto Vector Point 1433
Extension 3014
General Command Format 1496
Generic Feature 1789
Iterative Alignment 1828
Iterative Alignment Rules 1829
Measured Circle 1471
Measured Cone 1473
Measured Cylinder 1473
Measured Line 1469
Measured Plane 1470
Measured Point 1469
Measured Round Slot 1474
Measured Sphere 1472
Measured Square Slot 1475
POINTINFO 2918
PRINT/REPORT Command 2946
Read Point 759
Recall Alignment 1862
Command Lines 1801
Command Mode 748
Display Options 357
Indented Commands 718
Keyboard Functions 771
Command Set property 2681
Command Shortcut Menu 3282
Commands 2381
Dragging and Dropping 738
Inserting 718
Marking for Execution 687
Move Point 2381

Removing 737
Selecting 721
Commands Not Supported Converted to Comments dialog box 86
Commands Related to the Analysis Window 2882
Comments 759
Comment Text 2940
Document 2927
Enable Report 2932
Readouts 762
Report 2926
Show in Report 2932
Common Datums 2005**,** 2006**,** 2006
Best Practices 2006
Guidelines 2006
Common Properties 2658
Compensating for Temperature 409
Compensation method 414
Compile icon 809
Composite Boundary 2327
Concentricity 2096
CONCENTRICITY Dimension 1919
Concentricity GD&T 1919
Cone Feature 1333
ArcReductionAngle 1333
Cone
Changing the Direction 1728
Field Definitions 1433, 1473
Considered features 1989**,** 2052
constraining 2156
constraining degrees of freedom 2156
Constraints for 3D Best Fit Alignments 1847
Construct Generic Feature Dialog Box 1791
Constructed Feature 1498
Constructed Features 628**,** 749
Adjusted Filter 1781
Alignment Line 1551
Apply Tension Factor 1755
Auto Circle 1620
Auto Cone 1724
Auto Cylinder 1706
Auto Ellipse 1656
Auto Line 1548
Auto Plane 1584
Auto Point 1509
Auto Sphere 1740
Best Fit Type 1622, 1589
Best Fit:Circle 1621
Best Fit:Cone 1725
Best Fit:Ellipse 1657
Best Fit:Line 1551

Best Fit:Plane 1586
Best Fit:Sphere 1741
Cast:Circle 1626
Cast:Cone 1727
Cast:Cylinder 1710
Cast:Ellipse 1660
Cast:Line 1555
Cast:Plane 1590
Cast:Point 1514
Cast:Sphere 1745
Changing Nominals Directly to Calculate Offsets 1603
Changing Offsets Directly to Calculate Nominals 1568
Circle Direction 1628
Cone Direction 1728
Constructed Features Toolbar 895
Control Point List 1699
Corner Point 1515
Create Analysis View Command 2886
Create Corners 1756
Curve 1695
Cutoff Frequency box 1780
Cutoff Wavelength box 1780
Cylinder Direction 1711
Cylindrical 1778
Data Used to Construct the Arc 1632
Data Used to Construct the Line 1563
Decimal Placement Conventions 1497
Dependent / Independent Curve 1697
Dialog Box Description 58
Drop Point 1513
Ellipse Direction 1661
Example of Calculating Nominals 1569
Example of Calculating Offsets 1570
Filter Set 1766
In / Out:Circle 1620
In / Out:Cone 1724
In / Out:Cylinder 1706
In / Out:Ellipse 1656
In / Out:Slot 1669
In / Out:Sphere 1740
Intersect:Circle 1625
Intersect:Line 1556
Intersect:Point 1510
Line Direction 1561
Linear option 1767
Mid Line 1557
Mid Plane 1590
Mid Point 1515
Offset:Alignment Origin 1803
Offset:Alignment Rotation 1803
Offset:Alignment Translation 1803
Offset:Calibration Tool 934

Offset:Force 345
Offset:Generic Loops 2433
Offset:Move 2379
Offset:Pattern 1520
Offset:Point 1519
Parallel Line 1558
Parallel Plane 1593
Perpendicular Line 1559
Perpendicular Plane 1591
Pierce Point 1517
Plane Direction 1594
Plane from an Alignment 1585
Point 1505
Point at the Origin 1513
Projected:Cone 1726
Projected:Cylinder 1710
Projected:Line 1560
Projected:Point 1516
Projected:Sphere 1744
Secondary Datum 1570
Secondary Datum:Line 1570
Set of Features 1761
Sphere Direction 1745
Tangent Circle 1628
Tertiary Datum 1520
Tertiary Datum:Point 1520
Translated Plane 1604
Vector Distance Point 1519
Constructed Features Toolbar 895
Constructions 1498
Best Fit 1498
Best Fit Recompensate 1498
Control Pairs 2422
Control Point
Edit 1699
List 1699
Control Points 2351
Control Points Types 2351
Controller Compensates Axes Only 414
Controller Comps Axes and Part 414
Controller Support 416
Controlling Math Types 2254
Controls area 2345
Coordinate System 1483
COP 897
Pointcloud Toolbar 897
COPs
Showing or Hiding 439
Copy 659
Copy a Measurement Routine 94
Copy Icon 808

Copy to Actuals 1483
Corner Point Field Definitions 1433
Create Analysis View Command 2886
Create
Corners 1756
from File 3114
Weights 1852, 1848
Creating a Package 81
Creating PC-DMIS Alignment Routine
Manual
Measurement
Creating Probe Assemblies 1270
Creo 151
Cross Sections
Showing or Hiding 883
Current temperature boxes 411
Curve and Surface Modes 524
Custom Fixtures 1023
Adding 1023
Custom Probe Builder Utility 967
Custom Report Editor 2684
Custom Reports 2644
Customize User Interface 67
CustomMenuItem.dat 464
Cut 658
Cut Icon 808
Cut Plane Vector (CutVec) 2328
Cutoff Frequency box 1780
Cutoff Wavelength box 1780
Cylinder Feature 1333
ArcReductionAngle 1333
Cylinder
Basic Scan 2307
Changing the Direction 1711
Field Definitions 1433
Cylindrical 1778
Cylindrical surface 2004
Cylindricity 1920**,** 2066

D

Dark Theme 52
DAT Files 464
Data Collected 343
Data Fields 677
Data Files 464
Data Type 1793
Data types 2793
Data Used to Construct the Arc 1632
Data Used to Construct the Line 1563

Database
Directories 2488
Options 2487
Datalog 213
DataPage Option 2488
DataPage Variable Name 2488
DataPage+ Option 2493
Date and Time of Calibration 943
Datum 2006**,** 2156**,** 2048
datum boundary 2156
Datum Definition dialog box 2002
Datum definition syntax 2002
Datum feature shift 2156
datum feature types 2004
Datum Features 2156**,** 2052
Datum math 2048
Datum patterns 2018**,** 2156
Datum Reference Frame 2156
Datum Rules 2156
Datum shift 2156
Datums 2005**,** 2006**,** 2006**,** 2002**,** 2156**,** 2052**,** 2004
Common 2006, 2006
Defining 2002
Viewing 588
Datums when Repeating Dimensions 1954
Daturms 2002
DCC
Enter DCC Mode 904
Mode 904
Debugging Measurement Routines Offline 3196
Decimal Placement Conventions 1497
Default 932**,** 2433
Default Dimension Info 1897
Default RMEAS Mode 1462
Default Tolerances 284
0-5 Places 284
Default User Interface 77
DefaultDimensionColors.dat 464
defining 2002**,** 2018
defining and using common 2005
defining and using datum 2018
defining and using single 2004
defining datums 2002**,** 2002
Defining Hardware 917
Defining Machines 998
Defining Probe Angles 1262
Defining probe changers 1025**,** 1262
Environment Builder 1262
Defining Probes 917**,** 1262

Environment Builder 1262
defining single 2004
Defining Star Probes 917
Defining the I++ Client Probe Changer 1111
Degrees of freedom 2156
Delay before reading part temperature box 415
Delete 660**,** 753
Commands 721
Deleting CAD Elements 634
Dimensions 636
Directory from List 2488
Features 635
Part Files 95
Delete Current 2488
Deleting 889**,** 1007
Depth
color depth 439
editing for measured features 1489
for Auto Feature Cylinders 871
For Auto Feature Initial and Perm Hits 871
for Auto Feature Pitch 871
for Auto Feature Sample Hits 871
for BASIC Scans 2307
for DCC Scans 2348, 2343
for Probes 3190
DES Input 3186
DES Option 2490
Description Area 3017
Description Tab 2060
Deselect All 2348
Desktop Settings 439
Detecting Collisions 621
Deviation 1962
Arrow Multiplier 267
Perpendicular to Centerline 1962
Symbols 312
Deviations Only 284
Dialog Box Tabs 65
Dialog Boxes 722
Dialog Editor 812
Diameter 1794
Diameter / Length 941
of ball tip 941
Diameter and Thickness 920
Diameter box 1388
Diameter of the ball tip 941
Diffuse 490
Dimension 628**,** 1889**,** 1890
Colors Window 864
Command Format 1890

Conventions 1892
Delete 636
Dialog Box 1892
Dialog Box Options 1894
Edit Colors 2887
Format 749
Headings 312
List box 63
Output Format 309
Quick Start Toolbar 846
Reporting 1890
Text 311
Toolbar 898
Dimension Colors 864
Dimension info 2048
Dimension Info Boxes 2904
Showing or Hiding 883
Dimension tab 308
Dimensioning Angle 1903
Dimensioning Features 1889
2D Distance 1925
3D Distance 1926
Angle 1903
Angle Degrees 287
Angle Offset 660
Angle Type 1911
Angularity 1912
Axes 1965
Axes for Position Dimensions 1961
Best Fit Options for Line Profile 1976
Best Fit Options for Surface Profile 1975
Both Option (for Position and Location) 1902
Circle Options 1929
Circular Runout 1977
Circularity 1915
Coaxiality 1917
Command Format 1890
Common Options 1894
Concentricity 1919
Control Options for Line Profile 1974
Control Options for Surface Profile 1974
Cylindricity 1920, 2066
Distance 1922
Edit Dimension Colors 2887
Flatness 1930, 2068
Form and Location 1974
Form Only 1974
Form Option (for Position and Location) 1902
Format 1890
Gap Only for Location Dimensions 1938
ID=[Feature] 60
Info Creation Rules 2905

Info Format 2909
ISO Limits and Fits 1942
Line Definition Format 1484
Line Profile 1967
List Area 2918
List box 63
Location 1976, 1932
Location Axes 2910
Location Options 1937
Material Conditions 1895
Options 311
Parallelism 1944
Perpendicularity 1948
Position 1950, 2101
Printing to Inspection Report 1893
Profile 1967
Roundness 1915
Rules for 2D and 3D Distance Dimensions 1925
Runout 1977
Straightness 1981, 2071
Surface Profile 1967
Symmetry 1982
Text Box Mode 533
Through Keyboard Input 1985
Total Runout 1977
Variables 1987
Dimensions 749
Automatically Create in the Edit Window 1893
Using the KEY IN Option 1985
DIMINFO Command 2913
Diminfo Labels
Showing or Hiding 883
Direction 1 Technique 2320
Linear Close Scan 2320
Linear Open Scan 2320
Patch Scan 2320
Rotary Scan 2320
Section Scan 2320
Direction 2 Technique 2320
Patch Scan 2320
Direction Technique areas 2320
Directory Setup 437
Disable 550
Disable High Quality Transparency 551
Disabling or Enabling Home Page 53
Discrete Measurements 3195
Disk Probe 930
Display Check Box 3104
Display coordinates 2052
Display Symbols 647
Display

Absolute Speed 270
Angle Degrees 287
As Icon 3115
As Radius 1963
Brackets for Feature Arrays 296
Controls area 2345
Dimension Info 1896
Dimension Information 1896
Error of Last Measured Feature 359
hits for auto features 1407
Metafile 593, 763
Options 2906
Displaying 539**,** 1970**,** 1972
Displaying Dimensions 1894
Displaying Line Profile Contour Plots 1972
Displaying Profile Contour Plots 1972
Displaying Surface Profile Contour Plots 1970
Distance 1922
Between Hits 2321, 2322
Box 1791
Dimensions Orientation 1928
Nominal 1924
Relationships 1922
to Target 359
DISTANCE Dimension 1923
Shortest Between Two Lines 1927
DMIS Import
Advanced tab 183
Error Log tab 182
General tab 180
DMIS
Label 940
Mode 774
Output File 132
Do / Until 2428
Code Sample 2429
Do Control Calculations 2488
Docking or Undocking Items 78
Document 761
Double Coercion 3092
Double Reference 3020
Draw Points 553
Drawing Surfaces 527
Drop Down Axes in Alignments 1803
Drop Point Column 682
Drop-Down List Box 65
DSL 1270
DSL Probe Assembly 1262
DST 1270
DST Probe Assembly 1262

DTT 1270
DTT Probe Assembly 1262
Dual Arm Measurement Routine Execution 3142
Dual Arms 3121
Calibration:Backup Files 3153
Calibration:Delete Files 3153
Dual Rotary Table 382
DXF Input 3186

E

EB 1262
EBM File 153
Edge Point 1431
Field Definitions 1433
Edge Point Only Mode 239
Edit 54**,** 926**,** 1852**,** 1897
Box 60
Measurement Strategy 444
Tool 937
Window 713
Edit Boxes for the General tab 267
Edit Text Reporting 2517
Edit Window 713
Change Colors 355
Colors 715
Command Mode 748
Core Concepts 714
Default Colors 715
Defining Colors 353
DMIS Mode 774
Feature Creation 1495
Fonts and Colors 702
Indented Commands 718
Layout 357
Navigation 714
Output and Printer Options 122
Printing 120
Scrolling 66
Setup 353
Summary Mode 727
Toolbar 885
User-Defined Groups 779
Using 713
Editing 660**,** 677**,** 720**,** 1489**,** 1860**,** 2052
Editors 805
Ellipse 1431
2D Feature Glow 264
Changing the Direction 1661
Field Definitions 1433
elogo.dat 464
Else / End Else 2425

Code Sample 2426
Else If / End Else If 2423
Code Sample 2424
Emission 490
End Alignment Command 1801
End Angle 1394
End Loop 2433
End Number 2433
End of a File or the End of a Line 3007
End Touch Vector (EndVec) 2328
Ending 3196
Entering Multiple Arm Mode 3127
Environment Builder 1294**,** 1270**,** 1262**,** 1315**,** 1262
Calibrating Probe Assemblies 1294
Calibration 1294
Creating Probe Assemblies 1270
Defining Probes 1262
Getting Started 1262
Probe Assemblies 1270
Probe Changers 1315
Environment Configurator 30**,** 29**,** 33**,** 31**,** 30
Menu 30
Environment Configurator Menu 30
Environment Configurator/Hardware Settings 33
Equally Spaced Angles Data 952
Equate Alignment 751
Equate Alignment Dialog Box 1865
Equate Original Alignment 751
Equating an Alignment 1865
Equating CAD to Measured Part Data 1870
Error Label 1837
Error Messages 2279
Error of Last Measured Feature 359
Evaluating 2220**,** 1989
Evaluating Geometric Tolerances 1989
Evaluating Size 2220
Evaluation phases 1989
Example 317
Clearance Plane 317
Excel 122**,** 215
Excel Template
Excel Viewer 813
Excluding Moves 2392
Exclusion Zone 2393**,** 2392
Execute 219
Any Order 221
Block 221
Feature 219
From 220

From Cursor 220
From Start Points 221
Marked Sets 697
Mini Routine 221
Mode 2335
Routine 218
executebarstate.dat 464
Executing Measurement Routines Offline 3196
Execution 263
Displaying execution time 268
Executing Measurement Routines 218
FindNoms Tolerance 267
List 222
Mode 230
Out of Order 221
Recording execution time 268
Execution dialog box 224
Resizing 231
Execution Tab
Avoidance Move Area 2337
Execution Time 706
About 706
Calibrating Execution Time Parameters 707
Estimating 710
Workflow 707
Exit 810
Exit to File Manager 3248
Exiting PC-DMIS 141
Export 3247
Export folder 437
Exporting Plane Features in IGES 203
Exporting to 122**,** 216**,** 215
Exporting/DWG 196
Exporting
Available Output Options 207
BASIC File 212
CAD Data or Measurement Routine Data 196
CAD Mesh 199
Datalog 213
DataView (AS3) 199
DES 208
DWG 209
DXF 209
Gds 198, 255
Generic Text 210
IGES Write Setup 206
Inspection Plans 215
PolyWorks Text 198
STEP 211
STEP AP214 (Part Model) 200
STL 200

VDAFS File 212
Wavefront File 200
XAML 201
XML 216
XYZ File 201
Expression Builder 3013
Expression Element Type 3013
ID 3014
Expressions 763
Accessing a Report's Object Properties 3098
Add Button 3016
Checking for Correctness 3013
Components 3018
Description Area 3017
Double Coercion 3092
Edit box 3017
Expression Builder 3013
Feature Arrays 3033
for Constructed Scan Minimum Circles 3101
Functions 3047, 3009
Functions As Members of an Array Example 3090
Functions Creating Other Functions Example 3089
Functions Defined Recursively Example 3090
Functions Passed as Variables Example 3089
Functions:Array Functions 3074
Functions:Arrays 3033
Functions:Assign From 3018
Functions:Assign To 3018
Functions:Assigning Values to Variables 3018
Generic Function Example 3088
ID Expression Combinations 3098
ID Expressions 3095
ID Wild Cards 3095
in a Measurement Routine 3009
Integer Coercion 3092
Keeping Expression Values Only 3010
Literals 3019
Operators 3045
Output to a File Using Expressions 139
Typing Expressions 3011
Values 3010
with Branching 3010
with File Input / Output 3010
Extended Sheet Metal Options 1415
Extended
Circle Field Definitions 1433
Edge Point Field Definitions 1433
Round Slot Field Definitions 1433
Square Slot Field Definitions 1433
Surface Point Field Definitions 1433
Vector Point Field Definitions 1433
External Command 3104

as a Menu Item or Toolbar Item 3105
Inserting 3103
External Elements 3103
External Measurement Routine 3111
External Objects
Editing 702
Inserting 2945, 3103

F

Fast Probe Mode 756
FCR25 Probe Changer calibration 1030
Feature Appearance
Graphic Options 615
Feature Control Frames 1989**,** 1989
Datums 2002
Parallelism Information 2083
Position Information 2101
Showing or Hiding 883
Feature ID
Rules 296
Feature Labels
Display ON/OFF 614, 3264
Editing 605, 612
Positioning 606
Setup 296
Shortcut Menu 3264
Showing or Hiding 738, 883
Feature List box 61
Feature Locator 871**,** 1433
Feature mapping 460
Feature Pointers 3032
Feature Selection Highlight 538
Feature Sets as Best Fit Alignment Inputs 1849
Feature Tooltips 605
Feature Type list 1371
Feature Types 2213
Features 2213**,** 2156
Array ID 3095
Arrays 3033
Auto 1329
Color 612, 614
Constructed from the Edit Window 1495
Delete 635
Display 612
Editing 612
Ending 3196
Feature Name 1794
Feature Type 1792
Generic 1789
Hiding 612
Hiding and Showing 612

ID 612
List Area 2918
List box 61
Measured 218, 1467
Paths and Orientation 1407, 1409, 1410
Selecting in Online Mode 605
Shortcut Menu 3264
Theoreticals 1486
Weights 1848
Features Use Dimension Colors 281
File 53
Displaying in a Different Way 3119
Format 958
Menu 2884
Menu for Basic Script 809
Open Options 3245
Operations 89
Operations Toolbar 876
File Box in Call Subroutine 2456
File Input / Output 2965
Dialog Box 2967
Sample Code:EOF and EOL 3008
Sample Code:File Close 2970
Sample Code:File Copy 2999
Sample Code:File Delete 3003
Sample Code:File Dialog 3006
Sample Code:File Exists 3005
Sample Code:File Move 3001
Sample Code:File Open 2969
Sample Code:Read Block 2981
Sample Code:Read Character 2972
Sample Code:Read Line 2975
Sample Code:Read Up To 2985
Sample Code:Recall File Position 2997
Sample Code:Rewind to Start 2993
Sample Code:Save File Position 2995
Sample Code:Write Block 2992
Sample Code:Write Character 2987
Sample Code:Write Line 2989
File Locations 460
Filter
Deviation Above 2921
Hit Interval 2921
None Used 2921
Out of Tolerance 2921
Tab 2365
Type list 1768
Width box 1779
Worst 2921
Filters 2921
Find and Replace 670**,** 674**,** 811**,** 812
Find Hole 1433

Find Nominals 238
During Execution 246
Tolerance 267
Find Noms Override 681
Dialog Box 682
from the Edit Window 686
in Execute Mode 686
in Learn mode 683
Find Vector(s) button 1376
Find What 676
Finding and Replacing Text 669
Finding Nominal Points 238**,** 246
All CAD Surfaces 684
Choosing a Nominal Point Later 684
Previous Surface 683
Priority Surfaces 684
Fine Probing 345
Fixed Dialog Positions 244
Fixed Distance Manual Scan 2307
Fixed Radius 1622
Fixed Time / Distance Manual Scan 2307
Fixed Time Manual Scan 2307
FIXED_RAD 1622
Fixture Tolerance 1836
Fixtures 1023
Showing or Hiding 883
Flatness 1930**,** 2068
FLATNESS Dimension 1930
Flip Line button 1371
Flip Surface buttons 1371
Flip Vector button 1371
Flip XYZ 1371
Flow Control 2421
Calling a Subroutine 2452
Case / End Case 2432
Code Sample:Do / Until 2429
Code Sample:Else / End Else 2426
Code Sample:Else If / End Else If 2424
Code Sample:If / End If 2422
Code Sample:Select / End Select 2430
Code Sample:Subroutine 2463
Code Sample:While / End While 2427
Default Case / End Default Case 2433
Delete Argument from Subroutine 2456, 2450
Do / Until 2428
Else / End Else 2425
Else If / End Else If 2423
Ending a Measurement Routine 2465
Ending Generic Loops 2433
Generic Loops 2433

If / End If 2422
Select / End Select 2430
Subroutine 2452
While / End While 2427
Flush for LinearOpen 2343
Fly Mode 271
Fonts and Colors 702
Footer 724
Formatting Keywords 725
Footers 722
for Auto Features 871
for Marked Sets 122
Force Part Alignment in Car Body 254
Force the Insertion of a Certain Measured Feature Type 1477
Forcing a Given Feature Type 1490
Form and Location 1974
FORM Command 2957
Form Dimensions Tolerance Zones 2532
Form Editor 2685
Form Feed Command 2953
Form Only 1974
Form Option (for Position and Location) 1902
Format Command 749
Forms 2732
Forms in Reports 2732
Freeform Advanced Scan 2307
From Teamcenter 157
Functions 3047
for Command Mode Expressions 3047, 3088
for Report Expressions 2769, 2783
Functions List 3047
Functions and Operators 2769
FUSION 806
FUSION Settings 473

G

Gage 1881**,** 1881**,** 1881
2D Radius 1881, 1881
Caliper 1881, 1881
Temperature 1881, 1881
Thickness 1883, 1881
Gage Axis 1883
Gage scan filter 943
Gage-centered pose 2156
Gages 1881
Gap Only for Location Dimensions 255**,** 1938
Gauss Filter / Cutoff Frequency 1625
Gauss Filter / Cutoff Wavelength 1554
gbarstate.dat 464

GD&T 2062, 1989
GD&T callouts 553
Showing or Hiding 883
GD&T dialog box 2060, 2026
Description Tab 2060
Nominals Tab 2052
Reporting Tab 2048
GD&T Notes
Displaying 883
Showing or Hiding 883
General Command Format 1496
General Tab Check Boxes 238
Generate Scan 2354
Generic Features
Command Format 1789
Creating 1790
Description 1789
Generic Feature Dialog Box 1791
Purpose 1789
Generic Function Example 3088
Generic Loops 2433
Ending 2433
Geometric Tolerance 2271
Geometric Tolerance Command 2220
Geometric Tolerance Description Tab 2060
Geometric Tolerance dialog box 2018, 2060, 2026, 2052, 2048
Description Tab 2060
Geometric Tolerance Types 2062
Geometric Tolerances 2271, 2078, 2063, 2149, 2006, 2006, 2096, 2066, 2018, 2060, 2068, 2062, 2156, 1989, 2083, 2090, 2101, 2116, 2132, 2048, 2071, 2111, 2154, 2279, 1989
Common Datums 2006, 2006
Get Current Temperatures button 416
GETROTABDATA function 3075
Getting Started 1262
Giving Open Windows the Routine Focus 3180
Global 2220
Global Commands 763
Global Size 2220
Global Used 957
Graph
Each Axis 2907
Options 2907
Worst Axis 2907
Graphic Display Options 439
Graphic Display Window 57
2 sided lighting 550
2D Rotate Mode 529
3D Grid Color 539
3D Grid Setup 480

3D Rotate Mode 531
Applying Textures to Selected CAD Objects 494
Applying Textures to the Entire CAD Model 493
Archive Tab 486
Assemblies of Parts 573
Back face culling 550
Background Color 536
CAD Coordinate System 642
CAD Levels 509
CAD Tolerances 651
Clearance Planes 653
Clip Planes Tab 495
Clipping Plane 497
Create Levels 509
Curve and Surface Mode 524
Delete CAD Elements 634
Deleting CAD 634
Deleting the LOADMACHINE Command 1007
Deleting the Machine 1008
Desktop Settings 439
Detecting Collisions 621
Disable Materials 550
Display CAD Information 588
Display Existing Machines 1005
Display Mouse Coordinates 527
Double Buffering 439
Draw as Defined 552
Draw as Wireframe 552
Draw Points 553
Drawing Surfaces 527
Editing the CAD Display 477
Enlarging the Part Image 529
Equating CAD to Measured Part Data 1870
Feature Appearance: Graphic Options 615
Feature Color 612
Feature Display 612
Graphic view toolbar 881
Graphics Modes Toolbar 879
Highlight 538
Highlighting Surfaces and Curves with the Mouse 527
Ignore these objects 551
Label Display 3264
Lighting and Materials 483
Lighting Tab 487
Make CAD 3D 514
Material Tab 490
Modify Display Symbols 647
Motion Options 548
On-Screen Elements 523
Open GL Options 439
Options 439
Output and Printer Options 119

Point Nominal Deviations 644
Positioning Feature IDs 606
Repainting 528
Rotating the Drawing 544
Rotation Options 548
Scaling the Drawing 543
Screen Captures 593
Screen Colors 535
Screen Modes 528
Screen View 479
Setting Up CAD and Graphics 483
Showing and Hiding Graphics 554
Shrinking the Part Image 528
Transforming CAD 637
Trihedron and Trihedron Colors 540
Verify CAD Information 589
Graphic Items toolbar 883
Graphic Options 615
Graphic View toolbar 881
Graphic Window Print Preview 120
Graphical 1900**,** 2882**,** 2048
Graphical Analysis 2882**,** 2048**,** 2524
Graphical Hit Selection 628
Graphical Hit Selection Method 628
Graphical Representation of Auto Feature Vectors 1330
Graphical Representation of QuickFeature Vectors 1330
Graphical Representation of Scan Vectors 2331
Graphics and CAD
Setting Up 483
Graphics Modes Toolbar 879
Grid 539**,** 1377
Grid Advanced Scan 2307
Group
Measurement Strategy 892
Grouping Commands 779
Groups
of CAD Elements 514
of Commands 779
Guessing
Feature Type 894

H

Half Angle for Location Dimensions 1939
Hard Probes
Defining 917
Using 29
Hardware 917
Defining probe changers 1025
Machines 998
Probes 917
Quick Fixtures 1015

Hardware Settings 33
header.dat 464
Headers and Footers (Edit Window) 722
    Formatting Keywords 725
Headers and Footers (Report Window)
    Modifying in .DAT Files 2535
Hexagon Customer Experience Improvement Program 18
HexMF 1262**,** 1262
Hide Probe Component 985
Hide
    Features 3270
    IDs 3270
High Point
    Field Definitions 1433
    Inserting 1431
High threshold boxes 412
Highlight Assembly Component 573**,** 3251
Highlight Color 538
History 874
    CAD Import 874
Hit Type 2343
Hit
    Adding Hit Lines 752
    Arrays 3038
    List 2917
Hits 752
    Deleting from the Hit Buffer 771
    Editing Target List
    in U 267
    in V 267
    Number for Leapfrog Operation 1873
    Number for Measured Features 1482
    on a Feature 3195
    on a Surface 3195
    Showing 1407
    Taking Hits 3195
Home Page 52**,** 52**,** 52**,** 52**,** 47**,** 48**,** 39**,** 44**,** 263
    ... Menu 48
    Adding to Favorites 52
    Creating a Routine 49
    Disabling or Enabling 53
    Opening a Routine 51
    Removing a Routine from Recent 52
    Removing from Favorites 52
    Right Pane 44
    Routine Locations 53
    Settings 47
    Theme 52
Home the Unit 3238
Horizontal CMMs Rules 27
Hot Keys (see Shortcut Keys) 3251

HPL-Laser
ACR3 Probe Changer 1062
HR-MP Probe Changer calibration 1089
HR-P Probe Changer calibration 1171
HR-R Probe Changer calibration 1180
HR-X1-TR calibration 1228
HR-XS Probe Changer calibration 1133
HyperView Report Editor 2504

I

I/O Channels tab 350
I/O Concepts 2966
I++ Client Probe Changer 1111
Icon that runs PC-DMIS in Reverse Axes mode 3145
ID 2314
Box 1371
Expression Combinations 3098
Expressions 3095
Manage 523
Rules 296
Setup tab 296
Showing or Hiding 533, 883
Wild Cards 3095
ID=[Feature] 60
If / End If 2422
Code Sample 2422
IGES
Compatibility 3185
Entities Supported 3185
Ignore CAD to Part 241
Ignore Motion Errors 239
Ignore these objects 551
IgnoreMotionErrors command 239
IJK Vector
Angle Vector 1384
Edge Report Vector 1415
Edge Vector 1381
Generic Feature Vector 1793
Line Vector 1383
Normal Vector 1373
Pin Vector 1415
Punch Vector 1415
Rotary Scan Vector 2332
Surface Report Vector 1415
Surface Vector 1373
Update Vector 1415
illumination.dat 464
Image Snipping 2644
Import folder 437
Import OCR GD&T From File 2266

Import Options 216
Important Notes 2386
Imported CAD Entities
Colors 460
Importing 156
ACIS 149
ASCII File 164
Autodesk Inventor File 154
CAD File 149
CATIA File 150
ChorusNT Measurement Routines 172
ChorusNT Measurement Routines:Step 1: Configure PC-DMIS 173
ChorusNT Measurement Routines:Step 3: Create the PC-DMIS Measurement Routine File and Import the DMIS file 174
ChorusNT Measurement Routines:Step 4: End of automatic Import Phase 174
ChorusNT Measurement Routines:Step 5: Manual Verification of the PC-DMIS Measurement Routine 175
ChorusNT Measurement Routines:Things to Watch for and Some Helpful Tips and Suggestions 175
Color Scheme 356
Creo File 151
DES File 151
Direct CAD File 194
EBM File 153
IGES File 154
Inspection Plan File 195
Inventor File 154
JT File 155
MeasureMax Project 195
Multiple CAD Files 148
NX (Unigraphics) File 156
Parasolid File 156
QIF file 156
Solid Edge 156
STEP File 157
STL File 157
VDAFS File 157
Viewing CAD Import History 874
VISI file 158
Wavefront File 158
XYZIJK File 158
Importing a QIF file 156
In / Out
Circle 1620
Cone 1724
Cylinder 1706
Ellipse 1656
Slot 1669
Sphere 1740
in cad 553
Increment box 1392
Increment Move 2386

Incremental Moves 339
Indent boxes
for auto features 871
for scans 2343
Indenting Commands 718
Indexable Rotary Table 386
Individual Angle Data 952
Infinite Rotary Table 388
Infinitely Indexable Wrist 3236
Initial Direction Vector (DirVec) 2328
Initial Hits for Scans 2343
Initial Vectors for Scans 2328
Inner Bound 2335
Input 761
Input List 1852
Insert Bookmark 885
Insert Breakpoint 885
Insert Defaults 699
Insert Form 2957
Insert Menu 54
Inserting 2957**,** 2953**,** 2945**,** 2882**,** 2904**,** 2915**,** 2923**,** 2881
Inserting Move Commands 2375**,** 2375
Introduction 2375
Inspection Plans 3197
Change Management Options 3214
Exporting 215
Inserting Move Commands Automatically 2412
Parameters and Rules 3204
Void Detection 3225
Inspection Reports
Saving 122
Viewing 813
Install PC-DMIS on All Computers 3122
Integer Coercion 3092
Interference 2156
Interior/Exterior Feature Type 1391
Intersect Features 1625**,** 1556**,** 1510
Introduction 2375
Introduction to Inserting Move Commands 2375
ISO 1101 2156
ISO 1101 Math Types 2156
ISO Limits and Fits 1942
Iterative Alignment
Command Format 1828
Dialog Box 1832
Rules 1829

J

JT Import 155

Jump to and from 679
Jumping to Specified Line Numbers 678

K

Keep Existing Dimension's Axes 255
Keeping Expression Values Only 3010
Key in Method
- Axis Scan 2307
- Center Scan 2307
- Circle Scan 2307
- Cylinder Scan 2307

Keyed in Data
- Angle Point 1431
- Auto Circle 1431
- Auto Plane 1431
- Cone 1431
- Corner Point 1431
- Cylinder 1431
- Edge Point 1431
- Ellipse 1431
- High Point 1431
- Line 1431
- Notch 1431
- Round Slot 1431
- Sphere 1431
- Square Slot 1431
- Surface Point 1431
- Vector Point 1431

Keying in Probe Depth 3192

L

Label
- Custom Templates 2701
- Display 614
- Length 296
- Manage 523
- Ordering 2711
- Template Editor 2682
- Templates 2700

Labeling Method 296
Labels in Reports 2711
Labels
- Background Color 612
- Editing Appearance 612
- Legacy 612
- Line Color 612
- Shadows 612
- Showing or Hiding 533, 883

Language 117
Last Hit 359
layout#.dat 464

layouttoolbar.dat 464
Leapfrog 1870
Learn During Execution 2386
Least Squares 1622**,** 1589
LEAST_SQR 1589
Leg Collision check
Automatically calibrating probe 962
Obtaining CMM limits 275
Resetting values 275
Running 275
Legacy Report 2531
Legacy RMEAS mode 1462
Leitz Clamping Value 334
Length Between Scan Points 1699
Length box 1389
Level
Alignment 1803
Levels
Creating 509
Deleting 509
Modifying 509
Viewing 483
Light Theme 52
Lighting and Materials 483
Lighting Tab 487
LIGHTINGMATERIALS.DAT 464
Line and a Point 846
Line Method
Direction 1 of Patch Scan 2320
Direction 2 of Patch Scan 2320
LinearClose Scan 2320
LinearOpen Scan 2320
ROTARY 2320
Section Scan 2320
Line profile 2116
Line Technique 2321
Line
Basic Scan 2307
Changing the Direction 1561
Definition Format 1484
Field Definitions 1433
Profile 1967
Linear Closed Advanced Scan 2307
Linear Features 3194
Linear Open Advanced Scan 2307
Linear option 1767
Lines over surfaces 482
Lines
Deleting 1892
Editing 1892

LINESEGMENTENDINDEX function 3075
LINESEGMENTSTARTINDEX function 3075
Link 3115
List Box 61
List of Available Tools 933
Literals 3019
Load Probe 758
Loading the Active Probe 398
LOADMACHINE Command 1007
Local 2220**,** 2048
Local Size 2220
Local sizes 2048
Location 1932
Location Axes 2910
Location Dimension Lines 1891
Location Options 1937
Locations 2052
logo.dat 464
Loop Ids 2433
Looping Offsets and Alignments 2433
Loops and ANALYSISVIEW commands 260
Low Force 343
Low Level Matrix Calibration 917
Low threshold boxes 412
LSPX5 / HR-XS Probe Changer calibration 1133

M

Machine Coordinate System (MCS) 359
MACHINE.DAT 464
Machine
- Showing or Hiding 883

Machines
- Assembly 584
- Axes 1004
- Coordinate System 359
- Defining 998
- Deleting 1008
- Display in the Graphic Display Window 1005
- Interfaces 406
- Part/Machine tab 270
- Switching Between Profiles 234

Maestro 1262**,** 1262
Maestro CMM 1262**,** 1262
Make 3D 509
Make CAD 3D 509
Make Variables Globally Visible 253
Managing On-Screen Elements 523
manual feature 263
Manual or DCC Mode 904

Manual Temperature dialog box 422
Manual
Compensation 414
Fine Probing 345
Hit Retract 271
Mode 230
Organization 18
Point Trigger Tolerance 349
Scan Options 2374
Scans 2307
Scans with a Hard Probe 2307
Scans with a Touch Trigger Probe 2307
Map balloon numbers to characteristic IDs 507
Map features to sensors 460
Mark 687
Mark All 688
Mark All Icon 885
Mark Current Feature 885
Mark Used 957
Marked Sets 694
Adding or Removing Features 696
Delete 697
Execute 697
Lock 697
Reposition 697
Sample 694
Window 859
Window in Operator Mode 3248
Marking Commands for Execution 687
Summary Mode 738
Marking External Objects for Execution 3117
Markings 687
Clear All 689
Clear Marked 885
Master / Slave Mode (see Multiple Arm Mode) 3121
Match case 676
Match whole word only 676
material 2156
Material Boundary 2156
Material coefficient boxes 410
Material Coefficients Editor 434**,** 411
Material Conditions 1895
Material Modifiers 2156
Material Tab 490
Materials 550
Math 2048
Math options 2048
Math Types 2156**,** 2254
Mathematical Functions 3067
Max / Min boxes 2324

Max
Angle 2324
Force 342
Increment 2324
Iterations 1837
MAX_INSC 1622
Maximum Acceleration 340
Maximum angle 2324
Maximum increment 2324
Maximum Inscribed 1622
Maximum Number of pages property 2681
Meas All Once 1833
Meas Ang box 1402
Measure 3 1870
Measure Order list 1391
Measure the Alignment Features 1815
Measured Feature 1467
Angular Definition Format 1483
Basic Hit 752
Circle 831
Circle Command 1471
Circular Feature Type 1484
Cone 833
Cone Command 1473
Coordinate System 1483
Copy to Actuals 1483
Cylinder 832
Cylinder Command 1473
Dialog Box 1481
Force Type 1477
Forcing a Given Feature Type 894
Guessing a Feature Type 894
Inserting 1476
Line 829
Line Command 1469
Override Guess 1490
Plane 830
Plane Command 1470
Point 828
Point Command 1469
Round Slot 837
Round Slot Command 1474
Sphere 834
Sphere Command 1472
Square Slot 838
Measured Features 628**,** 751
Measured Features Toolbar 894
Measured Hits Dialog Box 1489
Measured Point Method
Axis Basic Scan 2307
Center Basic Scan 2307

Circle Basic Scan 2307
Cylinder Basic Scan 2307
Measured Points 1890
Reporting Dimensions 1890
Measurement 3196
Measurement Properties area 1390
Measurement Results 2501
Measurement Routine 31
Opening 31
Measurement Routine in Multiple Arm Mode 3139
Measurement Routines 628
Adding seconds to execution time 268
Attaching External Measurement Routines 3103
Automatic Archiving 88
Copying 89
Creating New 79
Debugging in Offline Mode 3196
Displaying execution time 268
Editing 657
Editing from Summary Mode 732
Ending 2465
Executing in Offline Mode 3196
Marking Commands for Execution 687
Migrating Between PH10 and Tesa Star M Probes 985
Mirroring 89
Recording execution time 268
Renaming 89
Saving 84
Saving to a previous version 84
Saving to an older version 86
Schema numbers 86
Measurement Strategy Editor 262**,** 444
Measurement Strategy Group 892
Measurement Strategy Toolbar 892
Measuring Features 1467
Measuring Features Offline 3193
Memory Pages 2488
Menu Bar 53
Menu Item
Moving 69
Remove 70
menu_{language}.dat 464
Mesh Toolbar 897
Mesh
CAD Export 199
Meshes
Showing or Hiding 883
MessageBoxOptions.dat 464
Messages 2279
Methods for Best Fit Alignment 1840

Migration from Prior Versions of PC-DMIS (see XactMeasure Migration) 2250
Migration Report 2250
Min Angle 2324
Min Increment 2324
Min/Max 1852
MIN_CIRCSC 1622
MIN_SEP 1589
Mini routines 785
- Adding seconds to execution time 268
- ClearanceCube 792
- Creating 789
- Displaying execution time 268
- Feature Based Measurement (FBM) License 795
- Important Notes 803
- Recording execution time 268
- Shortcut Menu 790
- Sub Mini Routine 789
- Time Remaining item in Execution dialog box 224
- Using 787
- Using ClearanceCube Key Concepts 792
- Using FBM Key Concepts 795
- Using Group Property

Minimum angle 2324
Minimum Circumscribed 1622
Minimum increment 2324
Minimum Separation 1622**,** 1589
Minor Dia. box 1852
Minus 1966
- Tolerance 1924
- Tols Show Negative 282

Minus size tolerance 2052
Minus tolerances 2052
Mirror CAD 637
Miscellaneous Functions 3075
Mode=MAN / DCC 753
Modify the Dimension Prior to Printing 1893
Modifying 679
- Nominal Values 679
- Target Values 679

Monitors 439
Motion
- Commands 753
- Options 548
- Parameters 307
- Tab 329

Mount Point tab - Probe Changer dialog box 1236
Mouse or Motion Controller 1256
Mouse Over Highlighting 527
Mouse
- track ball style 1256

Mouse-over Highlight 538
Move Command 2375
Move Commands 756**,** 2375
Automatically Inserting 2412
Inserting 2375, 2375
Move Circular Command 2387
Move Clear Plane Command 2396
Move Clear Point Command 2396
Move Exclusive Command 3143
Move Increment Command 2386
Move Point Command 625, 2376
Move Rotary Table Command 2395
Move Sweep Command 2389
Move Sync Command 3143
Move Sync Command:Inserting 2388
Move Exclusion Zone 2392
Move Point 2381
4-Axis Moving 2381
Move Point Command 2381
Move to focus position 258
Move
by an Offset (alignments) 1803
Feature to Reference Plane 244
Speed 753
Speed % 334
to a Feature (alignments) 1803
to Specific Feature's Location 714
Moveset 2382**,** 2386**,** 2382**,** 2384
Using with ClearanceCube 2384
Moveset Command 2382**,** 2386**,** 2382
Important Notes 2386
Using 2382
Moving CAD 273
Moving Dialog Boxes 66
Multiple Arm Calibration Matrix (ArmArm.dat) 3126
Multiple Arm Mode 3126**,** 3121
Dual Arm Calibration Example 3147
Multiple Arm CMM 3122
Multiple Arm CMM:Step 1: Install PC-DMIS on All Computers 3122
Multiple Arm CMM:Step 2: Determine the Arm1 System 3122
Multiple Arm CMM:Step 5: Setup the Multiple Arm Connection 3123
Multiple Arm CMM:Step 6: Connect the Computers 3126
Multiple Arm CMM:Step 7: Enter Multiple Arm Mode 3127
Multiple Arm CMM:Step 8: Calibrate the Multiple Arm System 3127
Multiple Arm CMM:Step 9: Calibrate the Multiple Arm Probe Files 3136
Multiple Arm Machines 1005, 3126
Multiple Arm Machines:Causing an Arm to Wait to Prevent Collision 3143
Multiple Arm Machines:Creating a Measurement Routine Using Multiple Arm Mode 3139
Multiple Arm Machines:Flush for LinearClose 2343
Multiple Arm Machines:Multiple Arm Calibration Matrix (ArmArm.dat) 3126

Multiple Arm Machines:Temperature Compensation 3144
Multiple Arm Machines:This computer (primary) drives all arms 3125
Multiple Arm Machines:This computer is not used in a multiple arm setting 3126
Multiple Arm Machines:This computer relays commands from the main computer 3125
Multiple Arm Measurement Routines 3139
Multiple Arm Measurement Routines:Routine Execution 3142, 3145
Using CALLSUB Statements in Multiple Arm Mode 2459
Multiple User Settings 234
Multiple Windows 3179
Multiplier 1901
Multisection Manual Scan 2307

N

Navigating the User Interface
Introduction 37
New
Alignment Mode 693
Angles List box 951
Coordinate System 751
Icon 807
Level 509
Measurement Routines 79
Probe 917
Probe File 3230
Row 224
Script 809
Newton 342
Next Bookmark 702
Next Bookmark Icon 885
Nickname 943
No Graph 2907
No Probe default item 1241
Nominal 1986
for Angle Between 1910
for Distance 1924
Point 684
Size 1943
Nominal locations 2052
Nominal Values 679
Modifying 679
nominals 2052
Mode Tab 2370
Nominals Directly to Calculate Offsets 1603
Nominals display coordinates 2052
Nominals tab 2052
Noncalibrated Tip 920
Notch Field Definitions 1433
Nullfilter Scan Technique 2324
Number for Return Data 343
Number

Decimal Places 283
Deleted Features 634
Sections to scan 2307
Selected Features 634
Times to Offset 660
Numeric Control 28
NX 156

O

Object 2548
ActiveX 2550
Analysis 2555
Arc 2554
Bitmap 2568
Border 2567
Button 2569
CadReportObject 2574
Checkbutton 2587
ComboBox 2588
CommandTextObject 2586
Dimension Color Key 2589
DimensionInfo 2591
EditBox 2595
EditDblBox 2596
EditLongBox 2596
Ellipse 2597
Frame 2598
Gauge 2598
Graph 2604
GridControlObject 2613
Histogram 2612
Label 2618
Leader Line 2620
Line 2622
ListBox 2623
MultiEditBox 2624
OLE 2624
Page 2665
Pointer 2632
PointInfo 2633
Polyline 2635
Radiobutton 2635
Section 2665
Section/Page 2681
Select 2637
Slider 2642
Spinner 2644
Text 2648
TextReportObject 2645
TextVar 2648
TheFrame/TheView 2663
Object Properties 2656

see Properties in Reporting Objects 3098
with Expression Language 3098
Object Type List 3116
OCR 2271**,** 2266
Geometric Tolerance 2271
Geometric Tolerances 2271
OCR Import 2266
OCR Mode 2266
Offline Mode
Debugging Measurement Routines 3196
Executing Measurement Routines 3196
Introduction 3183
Measuring Features 3193
Prerequisites 3184
Probe Depth 3190
Probes 3189
Offset 2327
+ / - 2327
Force 345
XYZ 934
Offsets Directly to Calculate Nominals 1568
OK to Move 2380
Old Style Report 2531
On Error Supported Interfaces 2446
One CAD file for Multiple Measurement Routines 149
Open 3246
Open Basic 809
Open Icon 807
Open report file when generated 260
OpenGL 439
Options 439
Opening a Measurement Routine 31
Selected Environment 31
Operand Coercion 3092
Operand Coercion and Mixed Type Expressions 3094
Operand Types 3019
Operation 55
Operator Mode 3245
Calibrate Tips 3249
File Open 3245
Marked Sets Window 3248
Menu Options 3246
Operators for Expressions 3045
Operators for Report Expressions 2782
Optical 28
Optical Character Recognition 2271**,** 2266
Geometric Tolerance 2271
Optimize Path 3215
Optimize Path Workflow dialog box 3219
Summary 3222

Optimize Surface 1755
optimized 2156**,** 2048
Optimized Fit 2156
Optimized shift 2156**,** 2048
Option 1 3240
Option 2 3240
Option 3 3240
Options 439
for Scanning 2362, 2312
for Tracefields 2467
Menu 2886
PZR 504
Ordering labels 2711
Orientation for Distance Dimensions 1928
Origin 1833
Boxes 412
Button (alignments) 1803
Offset Distance (alignments) 1803
Out of Order Execution 221
Output To 1902
Outputs Migration 2250
Override button 683
Override Guess 1490
Overriding a Guessed Measured Feature 1490
Overriding Found Nominals 681
Overriding Guessed Features 680

P

Package Measurement Routine 81
Page Numbers
in Reports 2769, 2665
Pan options 504
Parallel GD&T 1944
Parallel Lines 1558
Parallel Planes 1593
Parallel To 1928
Parallelism 1944**,** 2083
PARALLELISM Dimension 1945
PARAM Statements 2957
Parameter Settings
Acceleration tab 340
Dimension tab 308
I/O Channels tab 350
Motion tab 329
Probe Options tab 341
Probe Trigger Options tab 346
Probing tab 325
Rotate Table tab 336
Part Image
Enlarging within the Graphic Display Window 529

Part material coefficients list 411
Part/Machine tab 270
Pass Back Settings from Subroutine 254
Passing
Feature Names into a Subroutine 2459
Numeric Characters into a Subroutine 2459
Text Strings into a Subroutine 2459
Variables into a Subroutine 2459
Variables To and From BASIC Scripts 3108
Paste 659
Icon 808
With Pattern 669
With Pattern Icon 885
Paste With Pattern 660
Patch Advanced Scan 2307
Patch Scans Maintain Last Increment 249
Path Definition Tab 2354
Path Lines
Changing Size 647
Moving by Dragging 625
Viewing Path Lines 873
Path Optimization Summary 3222
Path Regeneration 620
Paths 437
Paths and Orientation 1407**,** 1409**,** 1410
Pattern 660
Pattern matching 676
Pattern Setup - Alignment 660
Pattern Setup - Curve 664
Patterns 2018**,** 2156
Patterns of Features 660
PC-DMIS FUSION 806
Settings 473
PC-DMIS NC 28
PC-DMIS Planner 195**,** 3197
PC-DMIS
CMM 27
Gear 29
Laser 28
Menu Bar 53
NC 28
Overview 17
Portable 29
Setup Options 235
Vision 28
PDF 3D 199
Peek Window
Summary Mode 738
Perimeter Advanced Scan 2307
Perm box for Scans 2343

Perpen To 1928
Perpendicular To 1929
Perpendicularity 1948**,** 2090
PERPENDICULARITY Dimension 1948
Pierce Point Column 682
Planar surface 2004
Plane
Auto Plane 1431
Changing the Direction 1594
Display in Graphic Display Window 258
Field Definitions 1433
Measured Plane 830
Plane-Line-Point Alignment 846
Plus box 1965
Plus size tolerance 2052
Plus Tolerance
Angularity Dimensions 1915
Coaxiality Dimensions 1918
Flatness 1931
Parallelism Dimensions 1946
Perpendicularity Dimensions 1949
Roundness Dimensions 1916
Runout Dimensions 1980
Straightness Dimensions 1982
Symmetry 1985
Point Cloud 439**,** 897
Toolbar 897
Point Functions 3069
Point Info Boxes 2915
Showing or Hiding 883
Point Nominal Deviation 644
Point of Interest Definition and Reporting 1883
Point Only Mode 239
Point Target Radius 1835
Point Trigger Tolerance 349
Point
Coercion 3093
Density 345
Functions 3069
Info Format 2916
Reference 3024
Symbol 647
Pointcloud 897
Toolbar 897
Pointcloud Toolbar 897
Pointer Functions 3071
Pointer to Reference Data 3112
Pointers 3032
POINTINFO Command 2918
Pointinfo Labels

Showing or Hiding 883
Polar/Cartesian Toggle 1433**,** 1375**,** 1483**,** 1932
Polar/Rect 1794
Polar
Option 1768
Vector Compensation 327
Polygon 1431
2D Feature Glow 264
Polyhedral 934
Portable 29
Ports tab - Probe Changer dialog box 1238
Position 1950**,** 2101
Position Axes 2912
POSITION Dimension 1951
Position GD&T 1950
Position Reporting 2101
Positioning Accuracy 344
Positioning On-Screen Elements 523
Positive Reporting 284
PrbRdv 941
Precedence 3046
Predefined Character Sets 672
Preferences 233
Pre-Hit / Retract 753**,** 754
Prehit Distance 330
Press End Key 238
Preview Window 856
Options 857
Sizing 858
Previous temperature boxes 411
Primary Datum Plane 1596
Print 810
Full Report 3249
Icon 808
List 948
Preview 808
Print Command 2945
Print Job Preview 120
PRINT/REPORT Command 2946
Editing 2952
Printer
Choose Printer 118
Edit Window Options 122
Graphic Display window Options 119
Printer Output
Setup and Options 118
Printing Reports 122
Printing
Dimensions to the Inspection Report 1893
from the Edit Window 120

in black and white
the Graphic Display Window 119
the Inspection Report 122
Printout Example of Active Tips 949
Priority Surface Column 682
Pro/Engineer 151
Probe 757
Analog Probe Scanning Parameters 998
Centering on Screen 653
Changing to the (no probe) default 1241
Comp 327, 758
Configuration Preview 929
Description 923
File 3230
File Name 919
Map Files 959, 3236, 3243
Naming rules 919
Position 359
Setup 945
Showing or Hiding 883
Probe calibration
Defining Probes 917
Probe Changer dialog box
Calibrate tab 1028
Comm tab 1027
Defining probe changers 1025
Mount Point tab 1236
Ports tab 1238
Type tab 1026
Using probe-specific port clearance distances 1246
Probe Changer
Showing or Hiding 883
Probe Changers 1315
Adding 1315
Aligning 1315
Calibrating 1315
Calibrating Senmation Probe Changer 1156
Calibrating the ACR1 Probe Changer 1048
Calibrating the ACR3 Probe Changer 1062
Calibrating the FCR25 Probe Changer 1030
Calibrating the HR-MP Probe Changer 1089
Calibrating the LSPX5 / HR-XS Probe Changer 1133
Calibrating the SCP600 Probe Changer 1144
Calibrating the SP600 Probe Changer 1162
Calibrating the TESASTAR-PR / HR-P Probe Changer 1171
Calibrating the TESASTAR-R / HR-R Probe Changer 1180
Calibrating the TP20 Probe Changer 1207
Calibrating the TP200 Probe Changer 1218
Calibrating the Travel Rack HR-X1-TR 1228
Defining probe changers 1025
Defining the I++ Client Probe Changer 1111
Delete probe changer from Graphic Display Window 1249

Full calibration 1028
Partial calibration 1028
Showing an Animated Probe Changer 1247
Probe Compensation
Active 327
Probe Component
Visibility 985
Probe Components 926
Probe Depth 3190
Key-in 3192
on a Cone 3192
on a Feature 3190
on a Sphere 3191
Probe folder 437
Probe Head
Orientation 271
Probe mapping 460
Probe Mode
Fast Probe Mode 756
Probe Mode Toolbar 904
Probe Options tab 341
Probe Tip Diameter 920
Probe Tips List 901
Probe Toolbox 871
Placing and Sizing 872
Probe Trigger Options tab 346
PROBE.DAT 464
PROBECHANGER.DAT 464
Probes 917
Defining 917
Probes list 901
Probing Accuracy 344
Probing Mode 344
Probing Tab 325
Profile Contour Plots 1970**,** 1972
Profile GD&T
Dimensioning Line Profile 1967
Dimensioning Surface Profile 1967
Profile of a Line 2116
Profile of a surface 2132
Profile of line 2116
Program Mode 534
Programmer Comments 2923
Projected Distance
Angularity Dimensions 1915
Coaxiality 1918
Parallelism Dimensions 1946
Perpendicularity Dimensions 1949
Projection Planes list 901
Prompt History 378

Properties in Reporting Objects 2656
(Form Name) 2665
(Object Name) 2658
Accelerator 2658
Alignment 2658
AlignTextLeft 2635
AnchorSnaps 2658
Angle1 2554
Angle2 2554
Arrowhead 2620, 2622, 2635
ArrowheadHeight 2620, 2622, 2635
ArrowHeads 2555
Auto Size 2658
AutoScaleToFit 2555
AutoTicks 2642
AutoZoom 2574
BackColor 2658
Bitmap 2568, 2569, 2587, 2635
BitmapOffState 2587, 2635
BorderDrawn 2595, 2596, 2596, 2623, 2624
BorderLines 2589
BorderShow 2555
BorderStyle 2658
Bottom 2658
ButtonShape 2569
ButtonType 2569
CadNominalStepValue 2555
CaseOrPassword 2595, 2596, 2596
CircularityMaxMinDiameters 2555
Colors 2586, 2645
ComboType 2588
Command Set 2665
CursorPointer 2658
DecimalBase 2644
DeviationLineStyle 2555
DimFeatID 2591
DimID 2591
DisplayID 2642
DoVerbs 2624
Draw wires in shaded mode 2574
EditBuddy 2644
Empty Lines In Text 2586, 2645
Enable 2658
FlexHorizontal 2588
FlexVertical 2588
Font 2658
ForeColor 2658
Format 2648
GotoPath 2568
GraphOption 2591
GridShow 2555
Group 2658

HatchStyle 2658
Height 2665, 2663
HelpContextID 2658
HelpHotButton 2587, 2588, 2595, 2596, 2624, 2642
HighlightColor 2567
HitNumber 2591, 2633
Horizontal Positioning 2663
HorizontalScroll 2623, 2624
IncrAccelerator 2644
IsWedge 2554
LabelOffset 2555
LabelText 2589
Layer 2658
Layout 2568, 2569
Leader Line ArrowHeader Height 2574
Leader Line ArrowHeader Style 2574
Leader Line Color 2574
Leader Line ending at two features for angle 2574
Leader Line ending at two features for distance 2574
Leader Line Feature Mode 2574
Leader Line Style 2574
Leader Line Termination 2574
Leader Line Width 2574
LeaderTermination 2555
Left 2658
LimitsColor 2589
LineStyle 2658
LineWidth 2658
ListID 2623
ListItems 2588, 2635
LocalDecls 2665, 2663
LocalVariables 2665, 2663
LocOrderA 2591
LocOrderD 2591
LocOrderForm 2591
LocOrderH 2591
LocOrderL 2591
LocOrderPAng 2591
LocOrderPD 2591
LocOrderPRad 2591
LocOrderR 2591
LocOrderRS 2591
LocOrderS 2591
LocOrderT 2591
LocOrderUseDimAxes 2591
LocOrderV 2591
LocOrderWorst 2591
LocOrderX 2591
LocOrderY 2591
LocOrderZ 2591
MaxColor 2555
MaximizeBox 2663

Maximum Number of Pages 2665
MaximumEq 2596, 2596, 2632, 2644
MaxMin 2555
MaxMinSize 2555
MeasuredLineColor 2555
MeasuredLineStyle 2555
MinColor 2555
MinimizeBox 2663
MinimumEq 2596, 2596, 2632, 2644
NominalLineColor 2555
NominalLineStyle 2555
NumColumns 2613
NumDropped 2588
NumRows 2613
Object1 2620
Object2 2620
OLEProperties 2624
OrderAuto 2591
OrderDeviation 2591
OrderHitNum 2591
OrderMaxMin 2591
OrderMean 2591
OrderMeasured 2591
OrderNominal 2591
OrderNumberPoints 2591
OrderOutTol 2591
OrderPoint 2591
OrderStdDev 2591
OrderTolerances 2591
OrderVector 2591
Orientation 2658, 2642, 2644
OwnerDrawn 2587
Padding 2658
Page Orientation 2663
Page Size 2663
PC-DMIS Commands 2658
PC-DMIS References 2658
PointInfoLabel 2555
ProfileMeasured 2555
ProfileNominal 2555
ProfileOptions 2555
ProfileTolerances 2555
ReadOnly 2595, 2596, 2596
RemoveSelection 2623
ReportLock 2663
Right 2658
RulerElemX 2555
RulerElemY 2555
RulerElemZ 2555
RuleTree 2574, 2618, 2645
Scale 2555
SelectPointInfo 2555

ShadowStyle 2658
ShowAlignments 2645
ShowBorder 2574
ShowComments 2645
ShowDimensionID 2591
ShowDimensions 2645
ShowFeatureID 2591
ShowFeatures 2645
ShowGraph 2591
ShowHeadings 2591
ShowID 2633
ShowMoves 2645
ShowPointInfo 2591
ShowScreenCaptures 2645
ShowType 2591
Single Click Rule Tree 2574
Sort 2588, 2623
Supported Command Types 2663
TableFormat 2613
TabStop 2569
Tag 2658
Text 2658
TickFrequency 2642
TickStyle 2642
TitleShow 2555
ToolTipText 2658, 2663
Top 2658
TPDiameters 2555
TPOrderD1 2591
TPOrderD2 2591
TPOrderD3 2591
TPOrderDF 2591
TPOrderForm 2591
TPOrderPA 2591
TPOrderPR 2591
TPOrderTP 2591
TPOrderUseDimAxes 2591
TPOrderWorst 2591
TPOrderX 2591
TPOrderY 2591
TPOrderZ 2591
Transparent 2555, 2613
TrihedronShow 2555, 2574
TriState 2587
UseColors 2588, 2595, 2596, 2596, 2623, 2624
UseTabStops 2623
ValueEQ 2632, 2648
ValueID 2658
Vertical Positioning 2663
VerticalScroll 2623
Visible 2658
WantKeyInput 2623

Width 2665, 2663
Wireframe 2574
Word Break 2658
WrapAround 2644
Protecting from Collisions 1252
Probe Changer 1252
Probes 1252
Pulled Hits 29
PZR options 504

Q

QIF 156**,** 2871**,** 2871
About 2871
QIF Report 2871**,** 2871
About 2871
Qual Sphere 934
Qualification
Check 3237
Sphere 934
Tools 934
QualToolData Function 3075
Question Mark (?) Meta Character 602
Quick Fixture toolbar 915
Quick Fixtures
Dialog Box 1015
Inserting 1015
Installing 1023
Toolbar 915
Using Existing 1023
Quick Start 815
Align Toolbar 846
Calibrate / Edit Toolbar 849
Construct Toolbar 840
Constructed Features 842
Dimension Toolbar 842
Main Toolbar 815
Measure Toolbar 824
Quick Ways to Create Auto Features 1335
QuickAlign 1817
Using Scanning Strategies 259
QUICKFIX.DAT 464
QuickMeasure 914**,** 1881
Gage 1881
Gages 1881
Laser 915
Quit 3247

R

Radial Deviation 941
Radius 1794
Read Lock 2488

Read Point 1376**,** 1795
Read Pos for Auto Move Point 2376
Read Position 1795
Read Temperatures from Controller 414
Read Vector From Machine icon 1380
Readouts 762
Readouts Window 359
Recall Alignment Command Line Format 1862
Recall an Alignment 1862
Recall folder 437
Recalling an Existing Alignment 1860
Recover after Accidentally Moving a Part 1869
Rectangle 1794
Redo 658
Redo Icon 885
Reference Angle 1914
Reference temperature boxes 412
Reference Type 1482
References 3020
Referencing One CAD file for Multiple Measurement Routines 149
Regenerate Hit Targets 1483
Regenerating the Path 620
Relationship
for Angle Between Dimensions 1911
for Distance Dimensions 1927
Relative Measure 1462
Relative To 1428
Remove A Menu Item 70
Remove Breakpoint 885
Remove Outliers / Standard Deviation Multiple 1624
Remove Outliers check box 1781
Remove
Edit 3247
Features from Existing Marked Sets 696
Setup 3247
Removing a Routine from Recent 52
Removing Commands 737
Removing items from Favorites 52
Removing the Animated Machine from the Graphic Display Window 1007
Rename 95
Repainting the Screen 528
Replace 674
Replace All button 676
Replace button 676
Replace with 676
Report 760**,** 2250**,** 2052**,** 2871**,** 2871
Forms 2732
Generation Flow Chart 2504
Output 122

Parameters 307
QIF 2871, 2871
Template Editor 2679
Templates 2687, 2527
Watermark 116
Window 2506
Report and Label Templates 2687
Report axis 2052
Report Commands 2881
Report Expression Language 2768**,** 2769**,** 2849**,** 2792
Report Mode 2506
Report output 2048
Report Template 2688
Report Template Selection Precedence 2688
reporting 2018**,** 2220**,** 2871**,** 2517**,** 2871**,** 2501
Adding Notes to Reports 2540
Changing Text Colors 2540
Clearing Template Associated Data 2672
Colormaps 2585
Commands 2881
Custom Label Templates 2701
Custom Report Editor 2684
Dimension 1890
Edit Text 2517
Excel Template
Expression Language 2768
Form Editor 2685
Forms 2732
Forms in Reports 2732
Label Template Editor 2682
Labels 2700
Labels in Reports 2711
Modifying Standard Templates 2534
QIF 2871, 2871
Removing a Report Template 2698
Report Template Editor 2679
Report Window 2506
Sharing Report Templates 2700
Standard Reports 2527
Standard Reports:CADONLY 2527
Standard Reports:CADONLY_LANDSCAPE 2527
Standard Reports:PPAP 2527
Standard Reports:TEXTANDCAD 2527
Standard Reports:TEXTONLY 2527
Templates 2687, 2527
Using .DAT File Keywords 722, 2535
Viewing a Legacy Report 2531
Reporting Objects
About Object Properties 2656
Accessing Properties Using Expressions 3098
Common Properties 2658

The Object Bar 2548
Reporting Options 2524
Reporting Size 2220
Reporting tab 2048
ReportingToolbar.dat 464
Reports 2688
Adding Notes To 2540
Changing Text Colors On 2540
Creating Custom 2746
Creating Templates 2687
Inserting 2688
Legacy Text Only 2531
Standard 2527
Temporary Modifications On 2516
Repositioning Commands 721
Repositioning the Part Image within the Graphic Display Window 529
Reset 1875
Reset Execution List 222
Reset global settings when branching 242
Reset IDs 3270
Reset Loop 2443
Reset to Defaults button 416
Resolving Error Messages 2279
Restore 77
Restore the Default User Interface 77
Results 956
Results box 1875
Retract Distance 330
Retrolinear Only for Location Dimensions 1938
Return Speed 344
Reverse Axes mode 3145**,** 3145
Right Pane 44
RMEAS/DEFAULT 1463
RMEAS/LEGACY 1465
Rotary Advanced Scan 2307
Rotary Table 382**,** 339
Angle 336
Dual 382
Incremental Moves 339
Stacked 382
Tab 336
Rotate 1816
Rotate - 2 1833
Rotate & Translate 1852
Rotate (alignments) 1803
Rotate 2D
by Dragging 530
to an Element 530
Rotate 3D

by Dragging 531
to an Element 532
using Rotate Dialog Box 532
Rotate About 1852
Rotate by an Offset 1803
Rotate Only 1852
Rotate options 504
Rotate the Table 339
Rotate to a Line between Two Circles 1803
rotate with Rotation Widget 547
Rotating the Drawing 544
Rotation 547
Rotation and Other Motion Options 548
Rotation Direction 336
Rotation Options 548
Rotation Widget 547
Showing or Hiding 883
Round Slot Field Definitions 1433
Roundness 1915
ROUNDNESS Dimension 1916
Routine folder 437
RTF and PDF Files 1894
Rule Tree Editor 2714
Rulers 539
Rules 2156
Run Menu 813
Running Arm1 Measurement Routine on Arm2 3145
Runout 1977
Circular or Total 1979
How it is Measured 1978
RUNOUT Dimension 1980
Runout GD&T 1977
Runout to Check a Cylinder End 1979

S

Sample Code
EOF and EOL 3008
File Close 2970
File Copy 2999
File Delete 3003
File Dialog 3006
File Exists 3005
File Move 3001
File Open 2969
Read Block 2981
Read Character 2972
Read Line 2975
Read Up To 2985
Recall File Position 2997
Rewind to Start 2993

Save File Position 2995
Write Block 2992
Write Character 2987
Write Line 2989
Sample Hits 871
Sample Plane Auto Trigger area 348
Save 809
Graphics to Report 2887
Icon 808
Save Analysis View Command 2886
Save As 86
Save Measurement Routine on Execute 248
Save/Alignment Command 1860
Saving Measurement Routines 84
scale and rotate during execution 263
Scale Factor 267
Scale to Fit 2885
Scaling the Drawing 543
Scaling the Model by a Factor 543
Scan Adaptive Strategy 449
How to Attach to a Supported Feature 449
Scan Construction for Perimeter Scan 2326
Scan Speed % 335
Scan Type 2313
Scanning after Adding Control Points 2354
Scanning Your Part
Introduction 2307
Scanning
Acceleration 345
Advanced button 2314
Advanced Scan Dialog Box 2312
Analog Probe Scanning Parameters 998
Avoidance Move 2337
Axis 2328
Basic button 2314
Basic Scan Dialog Box 2362
Body Axis:LinearOpen 2322
Body Axis:Rotary 2322
Body Axis:Technique 2322
Boundary Plane Vector 2328
Boundary Points 2319
Boundary Points:Using the CAD Data Method 2317
Boundary Points:Using the Measured Point Method 2317
Boundary Tab 2368
CAD Data Method 2317
CAD Tol 2326
Calculate Boundary 2327
Center Point and Radius 2332
Clear Plane 2335
Clearing Boundary Points 2320
Common Functions 2362, 2312

Components of a Scan 2311
Control Point Types 2351
Control Points 2351
Cut Plane Vector (CutVec) 2328
Depth 2348, 2343
Direction 1 Tech:for LinearClose Scan 2320
Direction 1 Tech:for LinearOpen Scan 2320
Direction 1 Tech:for Patch Scan 2320
Direction 1 Tech:for Rotary Scan 2320
Direction 1 Tech:for Section Scan 2320
Direction 2 Tech:of Patch Scan 2320
Direction Technique areas 2320
Distance Delta 271
Editing Boundary Points 2319
End Touch Vector (EndVec) 2328
Filter Tab 2365
Filter Width box 1779
Flush for LinearClose 2343
Flush for LinearOpen 2343
Graphics Tab 2348
Hit Type 2343
Hit Type Tab 2367
Increment Box:Patch Scan 2325
Increment Box:Perimeter Scan 2326
Increment Box:Section Scan 2328
Indent boxes 2343
Initial Direction Vector (DirVec) 2328
Initial Hits 2343
Initial Vectors 2328
Inner Bound 2335
Line Technique 2321
Linear Method:Direction 1 of Patch Scan 2320
Linear Method:Direction 2 of Patch Scan 2320
Linear Method:LinearClose Scan 2320
Linear Method:LinearOpen 2320
Linear Method:Rotary Scan 2320
Linear Method:Section Scan 2320
Linear option 1767
Nullfilter Technique 2324
Offset 2327
Options 2362, 2312
Profile Contour Plots 1970
Section Location for Section Scan 2327
Strategies 259
Surface Features 3194
Thinning Parameter U 1755
Thinning Parameter V 1755
Time Delta 271
Top Surface Vector (SurfVec) 2328
UV Scan Settings 2333
Value Limit 2467
Variable Technique 2322

Vectors 2328
Scans as Best Fit Alignment Inputs 1850
Schema numbers for measurement routines 86
SCP600 Probe Changer calibration 1144
Screen Captures 763
Editing 593
Inserting 2960
of the Graphic Display Window 593
Send to a File 593
Send to the Clipboard 593
Send to the Report 2960
Screen
Color 535
Colors 535
Counters 527
Modes 528
OpenGL Settings 439
View 479
Script 807**,** 807**,** 3107
Scrolling the Edit Window 66
Search Path 437
Search
for a specific word or phrase 674
ID 60
Override IJK 934
Second Extension 3016
Select Center 2332
Secondary Datum Line 1570
Section Advanced Scan 2307
Section Location for Section Scan 2327
see Creo 151
see NX 156
Select 1833
Select / End Select 2430
Code Sample 2430
Select Alignment to Export 255
Select Center 2332
Select Last # 61
Select Subroutine 2452
Select Tip Toolbar 905
Selected Environment 31
Opening a Measurement Routine 31
Selecting
Features Using Meta-Character Matching 601
Features Using the Graphic Display Window 599
Options from the Menu Bar with the Keyboard 56
the Last ID(s) 603
Selection Highlight 538
Send skipped items to statistics 260
Senmation Probe Changer Calibration 1156

Senmation: Step 2 - Define the Mount Point 1157
Senmation: Step 3 - Define the Ports 1158
Senmation: Step 4 - Prepare for Calibration 1159
Senmation: Step 5 - Take the Manual Hits 1160
Senmation
Step 2 - Define the Mount Point 1157
Step 3 - Define the Ports 1158
Step 4 - Prepare for Calibration 1159
Step 5 - Take the Manual Hits 1160
Sensor mapping 460
Sensor numbers boxes 410
Set and Reset Options 350
Set as Priority 596
Set Defaults 279
SETROTABDATA function 3075
Setting Preferences
PC-DMIS FUSION 473
Setting up 437
Setting Up CAD and Graphics 483
Setting Up Search Paths 437
Settings 47
.DAT Files 464
Edit Window 353
Graphics / OpenGL 439
Import Options 216
Machine Interfaces 406
Multiple Users 234
PC-DMIS FUSION 473
Probe 917
Readouts Window 359
Scans 2361
Setting Preferences 233
Settings Editor:Notes for Renishaw PHS with Leitz Interface 3229
Settings Editor:Profile Error from a Set 1762
Settings Toolbar 901
Settings Window 850
Settings Window:Options 855
Temperature 409
Setup Options
Dimension tab 279
General tab 236
ID Setup tab 296
Part/Machine tab 270
Shank 917
Sheet Metal (Auto Features)
Advanced Options 1424
Axes 1937
Box-Selecting 604
Extended Options 1415
Feature Properties 1371
Features 1431

Hits 752
Measurement Properties 1390
Measurements 757
Shift 2156**,** 2048
Shortcut Keys 3251
Add to a Command 76
Customizing 76
Remove from a Command 77
Shortcut Keys and Menus 3251
Box-Select Menu 3270
Command Menu 3282
Keys Reference 3251
Menus Reference 3259
Report Mode Menu 3282
Short Cut Menus in the Report Window 3286
Shortcut Menus for Auto Feature Path Lines 3272
Shortcut Menus in Standard Windows Dialog Boxes 3260
Shortcut Menus in the Edit Window 3274
Shortcut Menus in the Graphic Display Window 3262
Shortcut Menus on the Toolbar 3260
Summary Mode Keyboard Functions 746
Show Feature Normal 628
Show Home Page 263
Show negative for minus tolerances 2052
Show optimized shift 2048
Show Probe Component 985
Show tracker parameters in offline 256
Show
All Arrows 2883
All Scan Parameters 2345
Comments 759
Deviations Tolerance 267
Extended Sheet Metal Options 244
Graph 2916
Headings 2916
Histogram 2885
Hit Deviations 245
Hits check box 1407, 2915
Horizontal Lines 2916
Icon 2916
Icon in Text Boxes 248
ID 2916
IDs 3270
Max/Min Deviations 2885
Menu 2885
Move in Machine Coordinates 2380
Report file when generated 134, 260
Scan hits 2345
Startup Dialog 246
Statistics 2885
Temperatures in Celsius 413

Type 2916
Vertical Lines 2916
Showing and Hiding Graphics 554
showing negative 2052
Shrinking the Part Image within the Graphic Display Window 528
Simplified temperature compensation 417
Simultaneous Move 339
Single datums 2004
Single Point Mode 2335
Size 2220
Size dimension 2255
Size of Rotation Widget 540
size tolerances 2052**,** 2048
Sizes 2048
Skip 224
Skip Number 2433
Skipped items to statistics 260
Slot
2D Feature Glow 264
Smooth Bad Data 1756
Smoothing Parameter box 1780
snap to 1377
Snap to Grid 1377
Snipping 2644
Solid Edge 156
SolidWorks 156
solving 2156
Solving datums 2156
Solving Error Messages 2279
SP600 917
SP600 Probe Changer calibration 1162
SpaceBall 1256
SpaceMouse 1256
SpacePilot
Configuring 3Dconnexion 3D Mouse Devices 1256
Spacer 871
Spacer for Scans 2343
Specification 2220**,** 1989
Specular 490
Speed 271
Absolute 271
Rotation 551, 548
Scan 335, 754
Top 271
Touch 334, 754
Sphere
Changing the Direction 1745
Field Definitions 1433
Spherical Tool 933

Spline Filter 1777
Square Slot Field Definitions 1433
Stacked Rotary Table 382
Standard Deviation 313
    Multiple box 1781
Standard Edit Commands 658
Standard Reports 2527
standards 1989
Star Probes
    Defining 917
Start 809
Start Alignment Command 1800
Start and End Angles 1394
    2 Degree Buffer Offset 1394
Start Label 1837
Start Number 2433
Start Points Always Track Execution 259
Start Points
    Boundary Points 2315
    for Auto Features 1384
    for Multiple Arms 3142
    Set Start Point icon 885
Starting Letters 296
Starting Number 296
Statistics/Include or exclude skipped items 260
Statistics
    Always Save Stats to File 281
    Always Update Database 281
    Data Source 2493
    Database Directories 2488
    Database Options 2487
    DataPage Option 2488
    DataPage Variable Name 2488
    Delete Current Stats File 2488
    Delete Directory from List 2488
    DES Option 2490
    Display 2467
    Do Control Calculations 2488
    Feature IDs in Statistical Databases 2433
    Loop Ids 2433
    Memory Pages 2482
    No Display 2467
    Output To 1902
    Statistical Data 751
    Statistics Options Dialog Box 2481
    Stats Off 2482
    Stats On 2482
    Stats Options 2481
    Tracefield Options 2467
    Tracefields 763
    Tracking 2467

Transfer 2485
Update 2485
Update Database Now 2488
Value Limit 2467
Stats File 2488
Status Bar 57
Status Window 862
Step-Next 224
Stop 224
Store Move 2380
STP Files 409
Straightness 1981**,** 2071
STRAIGHTNESS Dimension 1981
Strategies 259
Strategy Editor 444
String
Coercion 3093
Reference 3026
Structural Temperature Compensation 410
Structures 3030
Structures with Variables of Type Point 3031
Subroutine folder 437
Subroutine
Arguments 2451
Calling 2452
Code Sample 2459
Editing 2451
Overview 2447
Summary Mode
Accessing 727
Drag and Drop 738
Editing a Measurement Routine 732
Icon 885
Keyboard Functions 746
Layout 729
Marking Commands for Execution 738
Shortcut Keys 746
Unmarking Commands for Execution 738
Surface 2213
Surface and Curve Modes 524
Surface Feature Types 2213
Surface Features 3194
Surface Grid Density 1755
Surface Point Field Definitions 1433
Surface Profile 1967**,** 2116
Surface Profile Contour Plots 1970**,** 1972
Surfaceless 2213
Surfaceless Feature Types 2213
Swap Vectors button 1382
Switching Between Machine Profiles 234

Switching Between Open Measurement Routines 3179
Switching Probes 901
Symmetry 2111
SYMMETRY Dimension 1982
Syntax 2002

T

T Value 367
Table Avoidance 275
TablesOnly.dat 464
Tagged Features in a Feature Set or Scan 1850
Target Values 679
  Modifying 679
Temperature compensation 409
  Controller Support 416
  Enabled 413
  Get Current Temperatures 416
  High threshold boxes 412
  Input Parameters 410
  Linear 410
  Local Settings 417
  Low threshold boxes 412
  Material coefficient boxes 410
  Origin boxes 412
  Show temperatures in celsius check box 413
  Simplified 417
  Structural 410
  TEMP COMP Command in the Edit Window 416
  Thermal Expansion Origin command 429
  Time Remaining 415
  Using STP Files 409
  with Multiple Arm Calibration 3144
Temperature Gage 1881**,** 1881
Template , 2688
TemplRepPickerList.dat 464
Tension Factor 1755
Tertiary Datum Point 1520
TESASTAR-PR / HR-P Probe Changer calibration 1171
TESASTAR-R / HR-R Probe Changer calibration 1180
Tessellating a Part without a CAD License 195
Tessellation 439
Text Box Mode 533
Text Color 353**,** 2540
Text Report 2527**,** 2531
Textual 1899**,** 2882**,** 2048
Textual Analysis 2882**,** 2048
Texture Mapping 490
Textures 494**,** 550
  Disable 550
Theme 52

Theoretical Values 679
Modifying 679
Theoretical
Center 1852
Thickness 1380
Thickness Gage 1883**,** 1881
Thickness
for Point Only Mode Points 245
of Auto Features 1380
of the ball tip 941
Thinning Parameter U 1755
Thinning Parameter V 1755
This computer (master) drives all arms 3125
This computer (slave) relays commands from the main computer 3125
This computer is not used in a multiple arm setting 3126
Time Remaining 415
Time Remaining item in Execution dialog box 224
Tip 759
Active Tip List 920
Adding Tip Angles 950
Editing 920
Selecting Calibration Order 920
Selecting Tip 920
To Feature 1803
To X Axis 1803
To Y Axis 1803
To Z Axis 1803
Toggle 699
Toggle Bookmark 701
Toggle Breakpoint 699
Toggle icons
Auto Wrist Toggle 1404
Circular Moves Toggle 1407
Clearance Plane Toggle 1406
Manual Pre-Position Toggle 1407
Measure Now Toggle 1377
Polar - Cartesian Toggle 1375
Re-Measure Toggle 1377
Show Filtered Points Toggle 1413
Show Hit Targets Toggle 1407
Show Measured Points Toggle 1412
To Points Toggle 1413
View Normal Toggle 1409
View Perpendicular Toggle 1410
Void Detection Toggle 1411
Toggle Multiple Arm Markings 3141
Tolerance types 2062
Tolerance zone 2048
Tolerance
BClass 1943

Box 683
for Angle Dimensions 1910
for CAD 651
for Concentricity Dimensions 1920
for Distance Dimensions 1924
for Keyboard Created Dimensions 1986
for Line Profile Dimensions 1973
for Location Dimensions 1940
for Position Dimensions 1964
for Surface Profile Dimensions 1972
Radius 349
Zones for Form Dimensions 2532
Toleranced features 1989
Tool 934
Add 934
Delete 937
Edit 937
ID 934
Type 934
tool.dat 464
Toolbar Area 57**,** 897
Pointcloud 897
toolbar.dat 464
Toolbar/Portable 897
Toolbars 900**,** 897**,** 897**,** 915**,** 875
Active Arms 906
Active Rotary Table 907
Alignment 900
Auto Features 892
Basic Script Editor 807
CAD 890
Constructed Features 895
Custom 75
Delete 76
Dimension 898
Edit Window 885
File Operations 876
Graphic Items 883
Graphic View 881
Graphics Modes 879
Measured Features 894
Measurement Strategy 892
Mesh 897
Pointcloud 897
Portable 897
Probe Mode 904
Quick Fixture 915
Quick Start Toolbar 815
QuickCloud 915
QuickFixture 1015
QuickMeasure 914, 1881
Select Tip 905

Settings 901
To Create a Custom Toolbar 75
Window Layouts 888
Wizards 909
Tooltips
for a feature 605
Top Speed (mm/sec) 271
Total Runout 2154
Touch Speed 754
Touch Speed % 334
Touch Trigger Probe 2307
TP20 Probe Changer calibration 1207
TP200 Probe Changer calibration 1218
Tracefields 763
Transform CAD 637
Translate Mode 528
Translate Only 1852
Translated Plane 1604
Transparencies 550**,** 487**,** 881
Travel Rack HR-X1-TR calibration 1228
Treat Theo Values As if Stored in Part Coordinates 252
Triangle Filter 1777
Trigger Force 343
Trigonometry Functions 3067
Trihedron 547
Trihedron Colors 540
Trihedron for Alignments 1803
Trihedron Rotation Interface 547
Trihedron Settings 540
Trihedron Size 540
Troubleshooting 2279
True Position
See the Position dimension 1950
TutorElement Function 3075
Type tab - Probe Changer dialog box 1026

U

Unbounded Line 1389
unconstrained location 2156
Undo 658
Deleted CAD 634
Deleted Dimensions 636
Deleted Features 635
for Basic 810
Unicode 810
Uniform 1777
Unigraphics 156
Unmark specific lines within a feature 688
Unmarking Commands for Execution 738

Summary Mode 738
Update 2485
Update Database Now 2487
Update Dependent Commands 1876
Update theoretical values in while loops 253
Upper Force 343
Use DMIS Polar Convention 253
Use Program Layout for Execution 254
Use
Automotive Deviation Letters 249
CAD Provided ID's for Features 246
Circular Moves on Round Features 245
Datums 1895
DMIS Button in Edit Window 248
Find Nominals Override for Scans 250
ONLY Priority Surfaces for the Find Nominals During Scans 250
Pin 1415
Polar Coordinates 362
Previous Surface 683
Trigger Tolerance 349
User Interface 37
Customizing 67
Navigating 37
Restore 77
User Layouts 889
User-Defined Groups 779
UserLightingMaterials.dat 464
usermachine.dat
Understanding Data Files 464
userquickfix.dat 464
using 2002**,** 2156
Using a Moveset Command 2382
Using ClearanceCube 2397
Using datums 2002**,** 2156
using Geometric Tolerances 1989
Using Moveset Command with ClearanceCube 2384
Using the Function Line to Pass Variables 3109
Using the GetVariableValue and SetVariableValue 3109
Using Toolbars 875
usrprobe.dat 464
UV Advanced Scan 2307
UV Scan Settings 2333

V

Values 720
Variable Arrays 3043
Variable Technique 2322
Variables 1987
Variables with Expressions 3017
Variables 3027

Variations 1572
Vector
Arrow Size 647
Vector Point Field Definitions 1433
Vector1 Check Box 2348
Vector1 Row 2328
Vector2 Check Box 2348
Vector2 Row 2328
Vectors
Editing 2331
Used in Auto Features 1371
verification 1989
Verify CAD 589
View 54**,** 2052
View Feature Normal 628
View Menu 3247
Viewing 483**,** 653**,** 2052
Views List 901
Virtual table 998
Visualization
Angle 1903

W

Wait Check Box 3105
Warnings 2279
Watermark 116
Wavefront 200
Weights 1852
While / End While 2427
Code Sample 2427
Wide screen monitors 439
Widgets
Rotation Widget 547
Width box 1388
Width Meas 1402
Wildcards 670
Window 55
Window Layouts Toolbar 888
Windows 805
With Alignment 1803
With Surface Data 2213
Without Surface Data 2213
Wizards Toolbar 909
Working with 509
Wrist Calibration 3241**,** 3236
Wrist Device 3229
Calculate Error Map 3236
Calibrate the Unit 3234
Calibrating 3231, 3230
CJoint 3241

Home the Unit 3238
Installing 3229
Qualification Check 3237
Star Tip Calibration 3231
Using in a Measurement Routine 3239
Wrist in a Measurement Routine 3239
Wrist Installation 3229
Wrist Warning Delta 271
Write Lock 2488

X

XactMeasure 1989
XactMeasure Migration 2250**,** 2254
Controlling Math Types 2254
Migration Report 2250
Options 2254
Outputs 2250
Simultaneous Tolerances 2250
Suggested Workflow 2250
XAML 201
XML 216
XYZ
ASCII File 3189
Boxes 1793
Center 1386
Center Boxes 1386
Files 158, 194
Move 2378
Nominal 920
Offsets 273, 1520
Point boxes 1372
Start / End Point Boxes 1385

Z

Z-Buffer Depth 439
ZMouse 1256
Zone 2392
Zoom 263
In on the Part 529
Out of the Part 528
Zoom during execution 263
zoom manual feature 263
Zoom options 504