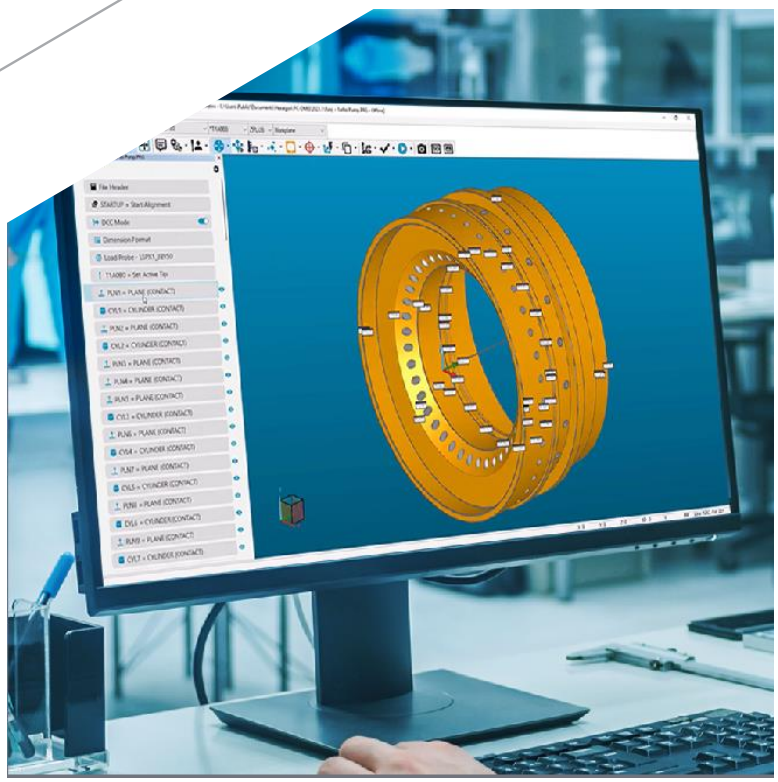
HEXAGON

PC-DMIS Laser 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 Laser: Introduction 8

Attributes for Laser Measurement 9

Getting Started 10

M&H LS-R Laser Sensor 10
Step 1: Install and Launch PC-DMIS 12
Step 2: Define the Laser Probe 14
Step 3: Define Setup Options for the Laser Sensor 16
Step 4: Calibrate the Laser Sensor 48
Step 5: Check the Calibration Results 58

Using the Probe Toolbox in PC-DMIS Laser 61

Laser Probe Toolbox: Position Probe Tab 62
Laser Probe Toolbox: Laser Scan Properties Tab 64
Laser Probe Toolbox: Laser Filtering Properties Tab 71
Laser Probe Toolbox: Acceptance Angle Filter Tab 77
Laser Probe Toolbox: Laser Clipping Region Properties Tab 78
Laser Probe Toolbox: Feature Extraction Tab 80
Laser Probe Toolbox: CWS Parameter Tab 102
Laser Probe Toolbox: Laser AF Multiple Creation Tab 107

Execution Modes 110

Using Asynchronous Execution Mode 110
Using Sequential Execution Mode 112

Using Sound Events 113

Using the Laser View 114

Using the Scan Line Indicator 115

Understanding the Visualisation Tools 117

Pointcloud Scanning Colors 120

Using the Laser Toolbars 121

Pointcloud Toolbar 121
QuickCloud Toolbar 128
Mesh Toolbar 128
HxGN Robotic Automation Toolbar 131

Using Pointclouds 132

Manipulating Pointclouds 133
Pointcloud Graphical Representation 135
COP Command Mode Text 139
Pointcloud Point Information 140
Laser Data Collection Settings 142
Using the Simulate Pointcloud Function 157

Simulate COP from CAD ... 160
Using the Animation Parameters for Pointcloud Simulation ... 161
Pointcloud Operators ... 165
Manipulating Pointcloud Operators ... 165
Pointcloud Thickness Colormap ... 167
Edit the Color Scale ... 173
SELECT ... 180
CROSS SECTION ... 187
SURFACE COLORMAP ... 217
POINT COLORMAP ... 229
CLEAN ... 235
PURGE ... 241
FILTER ... 242
Pointcloud EXPORT ... 245
RESET ... 249
EMPTY ... 250
Pointcloud IMPORT ... 251
BOOLEAN ... 254
Gages ... 255
Caliper Overview ... 255
2D Radius Gage Overview ... 269
Pointcloud Alignments ... 276
Pointcloud/CAD Alignment Dialog Box Description ... 277
Creating a Pointcloud/CAD Alignment ... 279
COPCADBF Command Mode Text ... 283
Creating a Pointcloud/Pointcloud Alignment ... 284
COPCOPBF Command Mode Text ... 288
Note on Adding or Updating Alignments in the Edit Window ... 289
TCP/IP Pointcloud Server ... 289
Extracting Auto Features from Pointclouds ... 293
Defining a Laser Auto Feature by Clicking on a Pointcloud ... 293
Example of How to Use the Depth Option to Create an Auto Circle Feature ... 296
Executing Scan-Extracted Auto Features ... 297
Aligning Measured Auto Features to CAD ... 298
Extracting Auto Features from a Mesh ... 299
Extracting a Laser Auto Surface Point from a Mesh ... 301
Creating Auto Features with a Laser Sensor ... 306
Implementation of QuickFeatures in PC-DMIS Laser ... 306
Extracting QuickFeatures or Auto Features when Created and Linked to a COP ... 307
Extracting with CAD Data ... 307
Common Laser Auto Feature Dialog Box Options ... 308
Laser Surface Point ... 313
Laser Edge Point ... 321
Laser High Point ... 325

Laser Plane ... 329
Laser Circle ... 332
Laser Slot ... 340
Laser Flush and Gap ... 346
Laser Polygon ... 361
Laser Cylinder ... 364
Laser Cone ... 372
Laser Sphere ... 378

Clearing Auto Feature Scan Data ... 380

Scanning Your Part Using a Laser Sensor ... 381
Introduction to Performing Advanced Scans ... 382
Common Functions of the Scan Dialog Box ... 382
Changing the Scan Speed ... 398
Performing a Linear Open Advanced Scan ... 400
Performing a Patch Advanced Scan ... 403
Performing a Perimeter Advanced Scan ... 406
Performing a Freeform Advanced Scan ... 410
Performing a Grid Advanced Scan ... 412
Performing a Manual Laser Scan on DCC Machines ... 414
Setting Machine Speed for Scanning ... 415

Simulate Scanning by Importing a Pointcloud ... 416

Handling Laser Sensor Errors with On Error Command ... 418

Using the Mesh Commands ... 420
Creating a Mesh Feature ... 423
Creating a Mesh Operator ... 430
Import Mesh in STL Format ... 462
Export Mesh in STL Format ... 463
Empty a Mesh ... 464
Mesh Alignment ... 465
Receive a Mesh from OptoCat ... 478

Receive a Pointcloud or Mesh from the HHScan Application ... 479

HxGN Robotic Automation ... 481
Introduction to HxGN Robotic Automation ... 481
PC-DMIS Workflow Using HxGN Robotic Automation ... 482
Configuring HxGN Robotic Automation with the Connector Application ... 483

Glossary ... 487

Index ... 489

PC-DMIS Laser: Introduction

This documentation describes how to use PC-DMIS with your laser sensor to measure features on a part, or to collect data. Laser sensors allow you to collect millions of points of data in one or more pointcloud (COP). PC-DMIS then uses these pointclouds for surface contour maps, export to reverse engineering packages, and creation of constructed features and Auto features. This documentation discusses how to use PC-DMIS with non-contact laser sensors to collect and interpret those pointclouds.

PC-DMIS Laser supports these hardware configurations:

> **!** Starting with PC-DMIS 2019 R2, PC-DMIS no longer supports the Perceptron laser sensor. While you may still be able to install PC-DMIS 2019 R2 and later, PC-DMIS displays an error if you attempt to execute measurement routines that use the Perceptron sensor. For additional information, please contact Hexagon Technical Support.

- HP-L-10.6 for DCC
- HP-L-20.8 for DCC and Portable
- HP-L-5.8 for CMM. The supported types are:
  - HP-L-5.8A-SYSTEM (AJ)
  - HP-L-5.8T-SYSTEM (TKJ)

The main topics in this help document include:

- Attributes for Laser Measurement
- Getting Started
- Using the Probe Toolbox in PC-DMIS Laser
- Execution Modes
- Using Sound Events
- Using the Laser View
- Using the Scan Line Indicator
- Understanding the Visualisation Tools
- Pointcloud Scanning Colors
- Using the Laser Toolbars
- Using Pointclouds
- Pointcloud Operators
- Gages

- Pointcloud Alignments
- TCP/IP Pointcloud Server
- Extracting Auto Features from Pointclouds
- Extracting Auto Features from a Mesh
- Creating Auto Features with a Laser Sensor
- Clearing Auto Feature Scan Data
- Scanning Your Part Using a Laser Sensor
- Simulate Scanning by Importing a Pointcloud
- Handling Laser Sensor Errors with On Error Command
- Using the Mesh Commands
- Receive a Pointcloud or Mesh from the HHScan Application
- HxGN Robotic Automation

If you come across something in the software that isn't covered in this documentation, see the PC-DMIS Core documentation.

Attributes for Laser Measurement

Before you get into the details of non-contact laser sensors, you need to understand their attributes to improve the results that you obtain when you use them for measurement. Laser sensors are excellent at gathering large amounts of data quickly. They are also good for measuring parts that may otherwise deform under the pressure of a tactile probe.

However, keep in mind that other factors, such as sunlight, surface finish, surface reflectivity, and surface color can influence measurements taken with laser sensors. To compensate for some of these factors, you can apply filters to the data to handle the influence. You should understand how and why these items affect the measurement results.

Sunlight

Unlike other non-contact systems, laser sensors are not generally affected by standard industrial lighting. Laser sensors work under varied lighting conditions because the sensor's frequency is tuned to its own laser. Only light that has the same frequency as the laser itself can affect the measurement. Because sunlight contains all frequencies of light, it is important to keep sunlight out of the inspection room.

Surface Finish

Because tactile probes are larger than the deviation in most surface finishes, a tactile probe acts as an averaging filter. When the tactile probe contacts the surface, it gives an average of the highest points on the surface. When you use a laser sensor, the light reflects off the surface of the part. How the light reflects depends a lot on the roughness of the surface, even if it does not appear rough to human sight or touch.

Surface Reflectivity

Generally, surfaces with a matte finish work better than those with a glossy finish. A glossy surface finish usually has directional reflection. Based on the angle of the light, you can get too much or too little light. You might even get a "hot spot" (something that looks like a "blob" in the Graphic Display window). This *blob* is actually the image of the light source. The reflection of light might add some extraneous points to the scan line, but the rest of the points are not affected by the reflection. To compensate for surface reflectivity, you can spray the part with an aerosol powder or paint.

Surface Color

Because the laser is light, the surface color can potentially impact the measurement. Similar to how something colored black absorbs heat from the sun, black surfaces on a part absorb the laser's light and make the measurement of those surfaces difficult. Darker colors have more potential for problems than lighter colors. If your part is too dark, you can apply powder coatings to it to make it easier to sample.

It usually takes time and experience with your particular parts and in your specific environment to determine what settings work best for you. You should experiment with the capabilities of your specific sensor to improve the measurement results.



WARNING: Exercise caution when working with laser sensors as they can damage your eyes. Consult your laser sensor documentation for safety issues and procedures for a safe work environment.

Getting Started

Before you use PC-DMIS with your laser device, the basic steps below can help you verify that your system is properly prepared.

The M&H LS-R laser sensor is currently for offline use only and doesn't require any setup procedures. You can find information on this sensor in the "M&H LS-R Laser Sensor" section of the PC-DMIS Laser documentation.

To get PC-DMIS running with your laser sensor, follow these steps:

M&H LS-R Laser Sensor

The M&H LS-R laser sensor is a wireless blue-line laser scanner for machine tools. Data is transmitted via radio to the measuring software through the machine controller.

The currently available M&H LS-R sensors are:

- LS-R-4.8

- LS-R-10.24

They both have the same enclosure, but with different standoff distances and FOV geometries.

The M&H LS-R laser sensors are best suited for inspection and reverse engineering applications.

You can use these laser sensors with PC-DMIS Offline to:

- Create and simulate Freeform, Linear Open, and Patch scans
- Handle metrology functions such as feature extraction from COP (no direct measurement)

PC-DMIS defines the scan paths which will then be converted to G-Code to move the machining center to scan the part. A separate software application initiates the laser to calibrate the probe and collect the data. When the G-Code executes on the machine, the data is sent to NC Server which opens the appropriate PC-DMIS measurement routine and executes the metrology functions, such as feature extraction from COP, using the collected laser data set.

M&H LS-R Laser Sensor: Licensing

To integrate the M&H LS-R sensor into PC-DMIS, you must have the correct license installed and configured.

- As a user, you must have the Work from Home (WFH) Offline CAD++ license.
- You can use the command line switch (/Laser:MTM) in a shortcut to a valid PC-DMIS executable.

> MTM stands for Machine Tool Management.

M&H LS-R Laser Sensor: Scanning

Linear Open, Patch, and Freeform scans

> Grid, Perimeter, and UNI scans are not currently available for this sensor.

Current support for scanning with the M&H LS-R sensor has some limitations which include:

- PC-DMIS disables unsupported scan types in the **Insert | Scan** menu options.
- PC-DMIS disables **Grid**, **Perimeter**, and **UNI** scan options from the **Scan** dialog box.

- PC-DMIS supports pointcloud simulation.

M&H LS-R Laser Sensor: Auto Features

The guidelines for Auto features using the M&H LS-R sensor are:

- Based on the active M&H sensor, PC-DMIS only supports extraction of Auto features from pointclouds.
- Direct measurements are not allowed.
- The Probe Toolbox only displays feature extraction parameters and COP object selection functionality.

M&H LS-R Laser Sensor: Probe Calibration

Because PC-DMIS does not interact directly with the M&H LS-R sensor and only offline operations are performed with the sensor, there is no need to calibrate the M&H LS-R sensor.

Step 1: Install and Launch PC-DMIS

Before you use your laser device, ensure that PC-DMIS has been properly installed on your computer.

To install PC-DMIS for your laser device:

1. Ensure that the machine that runs the laser sensor is properly set up and configured according to your machine's specifications. Follow the documentation that came with your laser sensor to properly connect the hardware.
2. Ensure that your license supports the laser option. This tells the installer to install the necessary laser components. If you don't have the necessary license or if it's not properly configured for laser, please contact the Hexagon Technical Support team.
3. Install PC-DMIS. To do this, see the release notes in the Readme.pdf file.
4. From the Windows **Start** menu, type **PC-DMIS 2026.2 Online** and then click **Open** to start PC-DMIS in Online mode.
5. Before you open a measurement routine, you can pre-configure PC-DMIS for a specific laser device with the Environment Configurator. For details, see the "Environment Configurator" section of the PC-DMIS Core documentation.
6. Open an existing measurement routine or create a new one. If you create a new measurement routine, PC-DMIS opens the **Probe Utilities** dialog box. In the next step, you use this dialog box to define your laser sensor.

The PC-DMIS installer manages the installation of drivers, and so on.

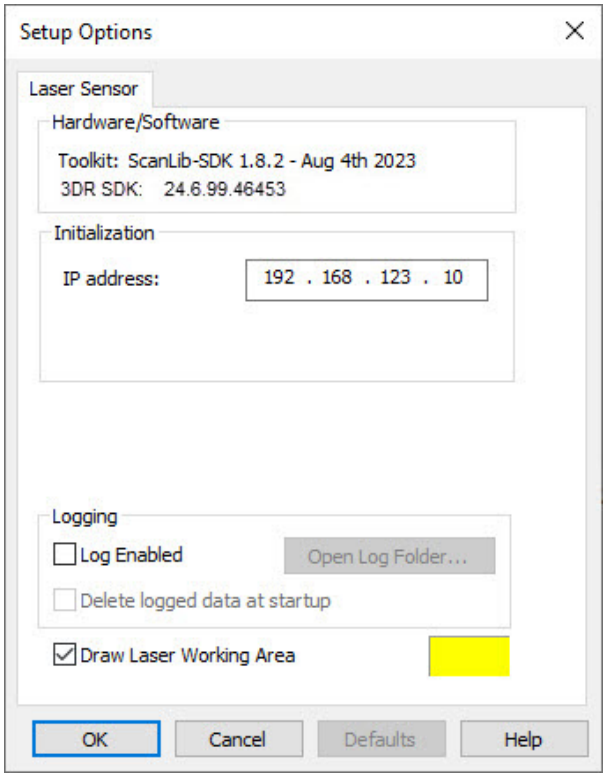
Setting Laser Sensor Parameters Before Opening a Measurement Routine

Sometimes you may want to change laser sensor parameters before opening a measurement routine.

For details on the options available on the **Laser Sensor** tab of the **Setup Options** dialog box, see the "Setup Options: Laser Sensor tab" topic in the PC-DMIS Core documentation.

To do this:

1. From the PC-DMIS Home screen, select the **Edit | Laser Sensor Setup** option from the menu. This opens the **Setup Options** dialog box to show the **Laser Sensor** tab.



Setup Options dialog box - Laser Sensor tab

2. From the **Laser Sensor** tab, select the **Draw Laser Working Area** check box.
3. Click the color patch to the right of the **Draw Laser Working Area** check box to open the color picker.
4. Select a pre-defined color or click the **Define Custom Color** button to define a different color.

5. Once you select the color to define the laser working area, click **OK** to close the color picker.

6. Click **OK** to close the **Setup Options** dialog box.

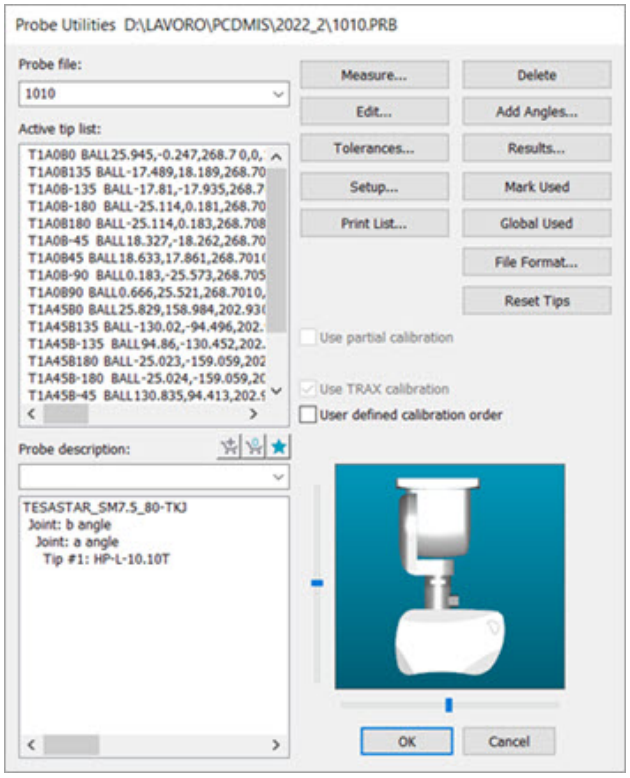
   The next time you scan a CAD model, PC-DMIS colors the scan working areas with the selected color.

The **Laser Sensor** tab is discussed in Step 3.

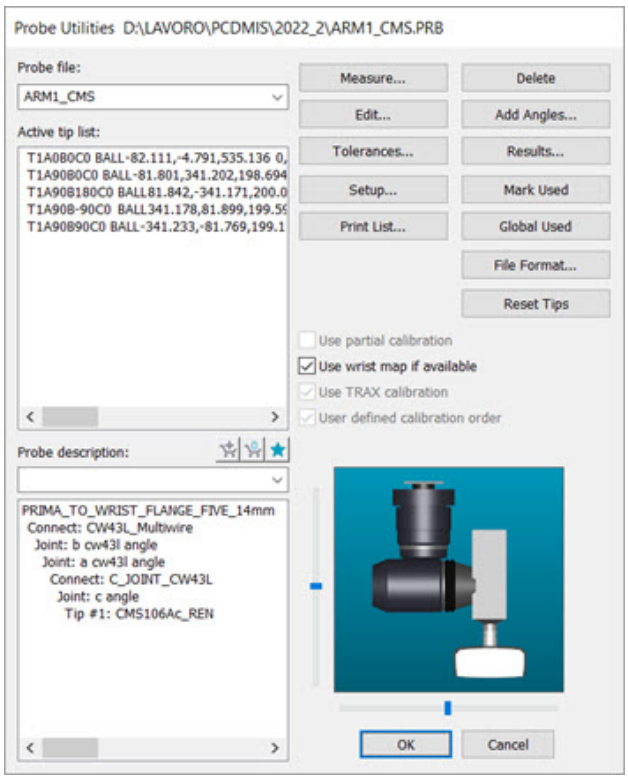
Step 2: Define the Laser Sensor

If you don't have a defined laser sensor, use the **Probe Utilities** dialog box to define it. This process creates a probe file.

1. If the Probe Utilities dialog box is not open, select the **Insert | Hardware Definition | Probe** menu option to open it. (This dialog box automatically opens whenever you create a new measurement routine.)



Probe Utilities dialog box for an indexable head



Probe Utilities dialog box for the CW43L

2. In the **Probe file** box, type a name that best describes your laser sensor.

3. From the list of components at the bottom, select the **No probe defined** text to highlight it.

4. From the **Probe Description** list, select the appropriate probe. You can mount the CWS or WLS sensor on a wrist with a TKJ connector, or on the OPTIV_FIXED for multi-sensor machines.

5. As needed, select additional components in the same manner for "empty connections" until you finish defining the probe. A defined probe shows a tip in the **Active Tip List**.

> Once you define the tip, the software no longer shows the probe image. This makes it so that the graphical image of the probe does not obstruct the view of the part during measurement. However, if you want to enable the display of probe components, double-click on the probe component to open the **Edit Probe Component** dialog box. Mark the **Draw this component** check box.

6. If you use a PH10, Tesa, or continuous-type wrists with a C joint, you need to verify that the joint angles are properly adjusted for visual purposes. Otherwise, PC-DMIS can't properly correlate the sensor's data to the machine position. If your probe is not rotated correctly about the joint, you can manually provide the extra rotation. To do this, right-click on the component and change the **Default rotation angle about connection** value to reflect the needed rotation.

The probe file does not define the orientation of the sensor about the joint; it only defines the probe vector.

For additional information on how to define probes, see the "Defining Hardware" section in the PC-DMIS Core documentation.

Step 3: Define Setup Options for the Laser Sensor

If PC-DMIS is configured for the HP-L laser sensor at startup, the system looks for the current-mounted probe. If it is *not* the laser HP-L sensor, and a probe changer is present, the system assumes that the sensor is in the probe changer and switches on the warm-up power state. This ensures that the sensor is warmed and ready for measuring.

1. If the **Probe Utilities** dialog box from the previous step is still open, close it.
2. Select **Edit | Preferences | Setup**, or press **F5**, to open the **Setup Options** dialog box.

   You can pre-define the preferences for your laser sensor before you start a measurement routine. To do this, from the PC-DMIS Home screen, select the **Edit | Laser Sensor Setup** option from the menu. This opens the **Setup Options** dialog box to show the **Laser Sensor** tab.

   For details on the options available on the **Laser Sensor** tab of the **Setup Options** dialog box, see the "Setup Options: Laser Sensor tab" topic in the PC-DMIS Core documentation.

   There is no **Laser Sensor** tab on the **Setup Options** dialog box for the CWS probe.

3. Select the **Laser Sensor** tab. The contents of this tab change based on the type of laser sensor that is defined in your PC-DMIS license.
   - HP-L Sensors
   - HP-L-10.10 Sensor
   - Using Zeiss Eagle Eye 2 with Zeiss I++ DME Server

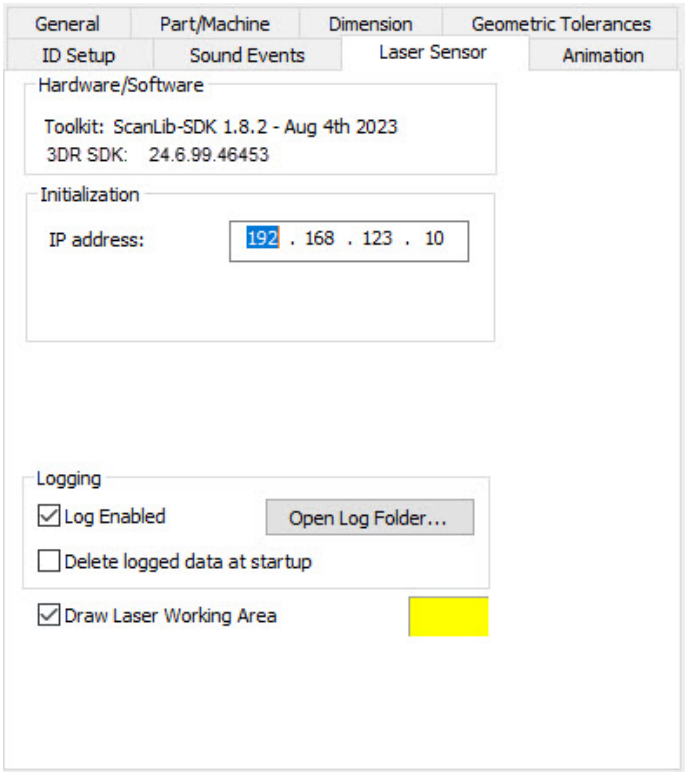
- Comparison of HP-L-5.8 and HP-L-10.6 Sensors

Changing the Color of the Laser Working Area

When you use laser scanners, you can change the color of the laser working area (or field of view) of the scanner. This helps to differentiate the area being scanned from the color of the CAD model in use, one or more of the CAD surfaces, or the screen background color while programming laser Auto features and scans.

To do this:

a. Open the **Setup Options** dialog box (**Edit | Preferences | Setup**) and select the **Laser Sensor** tab.



Setup Options dialog box - Laser Sensor tab

b. From the **Laser Sensor** tab, select the **Draw Laser Working Area** check box.

c. Click the color patch to the right of the **Draw Laser Working Area** check box to open the color picker.

d. Select a pre-defined color or click the **Define Custom Color** button to define a different color.

e. Once you select the color to define the laser working area, click **OK** to close the color picker.

f. Click **OK** to close the **Setup Options** dialog box.

The next time you scan a CAD model, PC-DMIS colors the scan working areas with the selected color.

You can also set laser sensor parameters before you execute a measurement routine. For details, see the "Setting Laser Sensor Parameters Before Opening a Measurement Routine" section of the "Step 1: Install and Launch PC-DMIS" topic in the PC-DMIS Laser documentation.

4. Follow the setup options instructions below for your laser sensor.

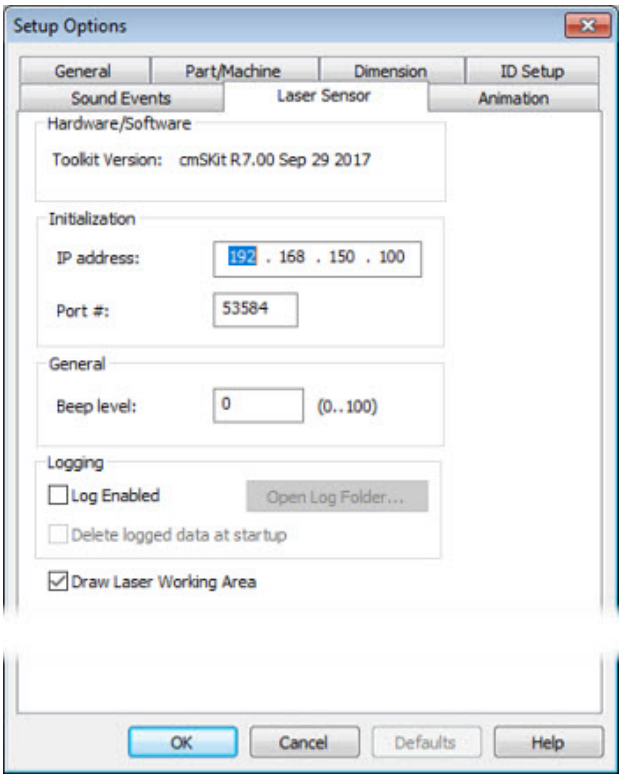
Settings Editor Entries for Laser Sensors

A PH10 wrist can automatically switch between a contact probe and a non-contact probe. These entries control that operation as well as the power on a laser sensor warm-up station:

- `PICSDifferentialSwitchBit`
- `WarmUpStationPowerBit`

HP-L Sensors

Open the **Setup Options** dialog box (**Edit | Preferences | Setup**) and click the **Laser Sensor** tab to show the options for your camera.



Setup Options dialog box - Laser Sensor tab example for the HP-L sensor

Hardware/Software Area

This area displays the current HP-L toolkit version.

Initialization Area

You can use the **IP address** and **Port #** boxes to define the IP address and the port number of the HP-L controller.

General Area

You can use the **Beep level** box to set the volume for beep sounds that come from the HP-L controller. It accepts any value from 0 to 100. A value of 0 turns the volume off completely.

Logging Area

Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Hexagon Technical Support can use these files to resolve issues that may involve your laser sensor.

> Logging stores debug information on your machine which may slow down normal operations. You should keep logging turned off for normal operations and only turn it on for the time that you need to capture debug information. When done, you should always turn it back off.

- **Log Enabled** - This check box enables or disables data sent to the log files.
- **Open Log Folder** - This button opens up the folder that contains your log files.

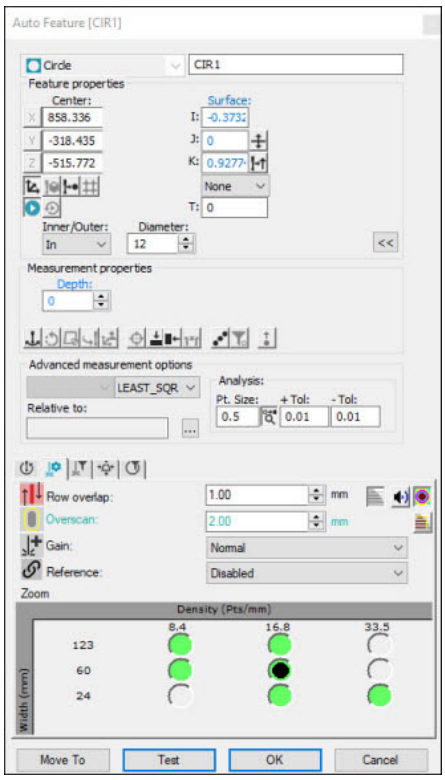
  > Typically, PC-DMIS stores these files in the "C:\ProgramData\Hexagon\PC-DMIS\2026.2\NCSensorsLogs\" folder.

- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you restart PC-DMIS.

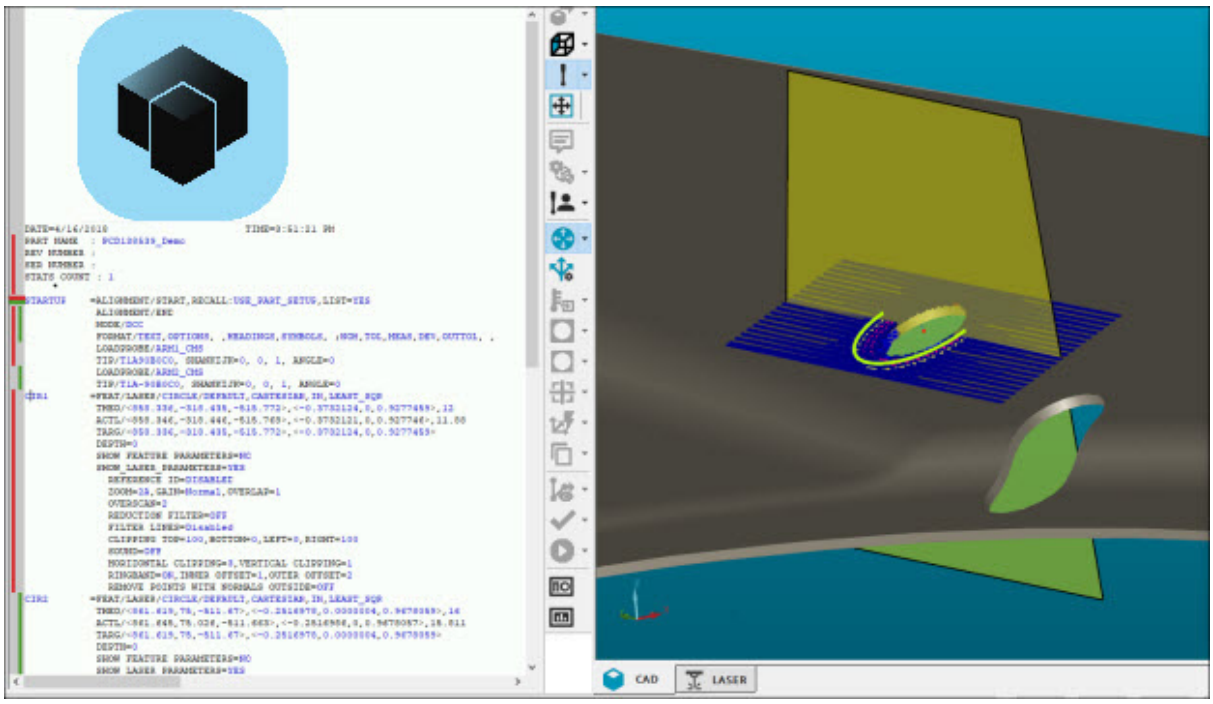
Draw Laser Working Area Check Box

If you select the **Draw Laser Working Area** check box, PC-DMIS uses the HP-L probe parameters to draw the trapezoid with the correct dimensions. This functionality helps with simulation in Offline mode and is available for laser Auto features and laser scans.

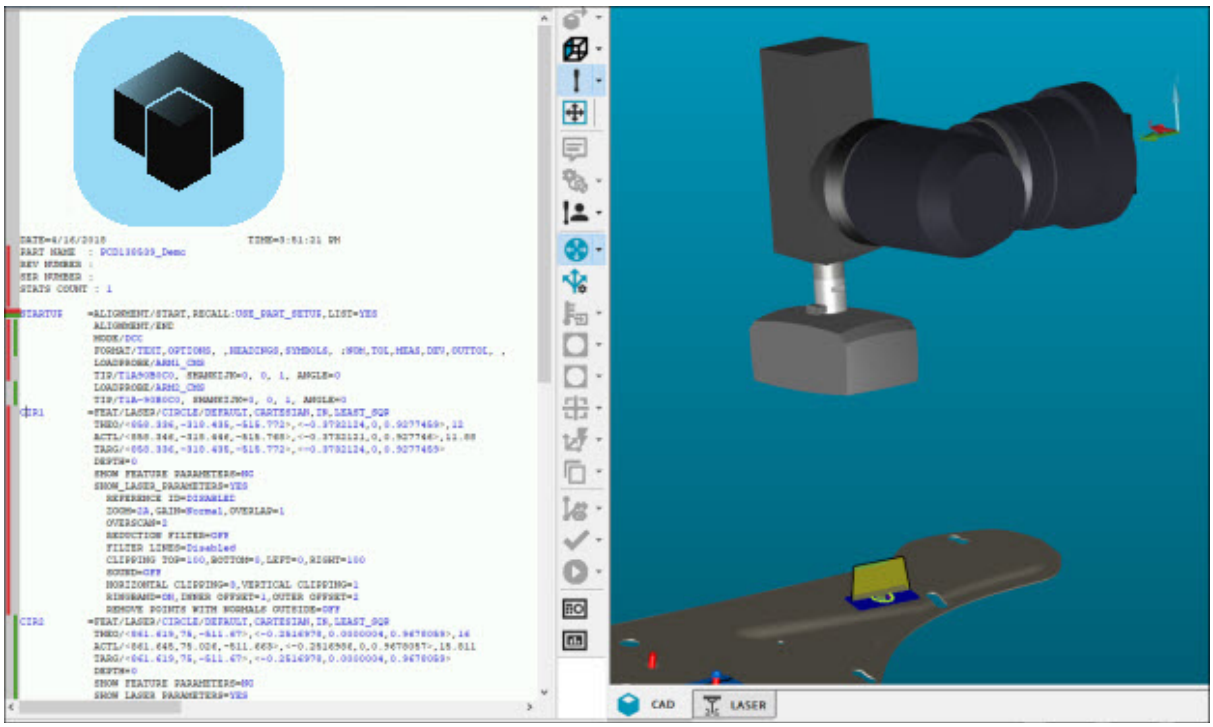
- For a laser Auto feature, PC-DMIS shows the trapezoid that represents the working area of the laser in the center of the feature. The trapezoid moves according to the simulation of the laser stripes. For an example, see the images below:



Example of Circle Auto Feature dialog box

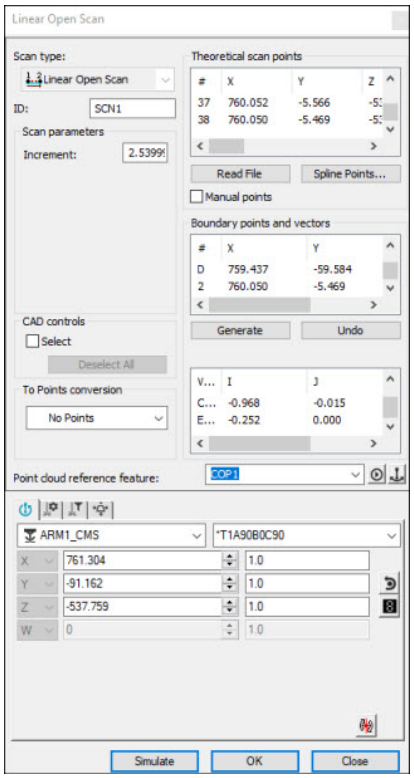


Example of Circle Auto feature

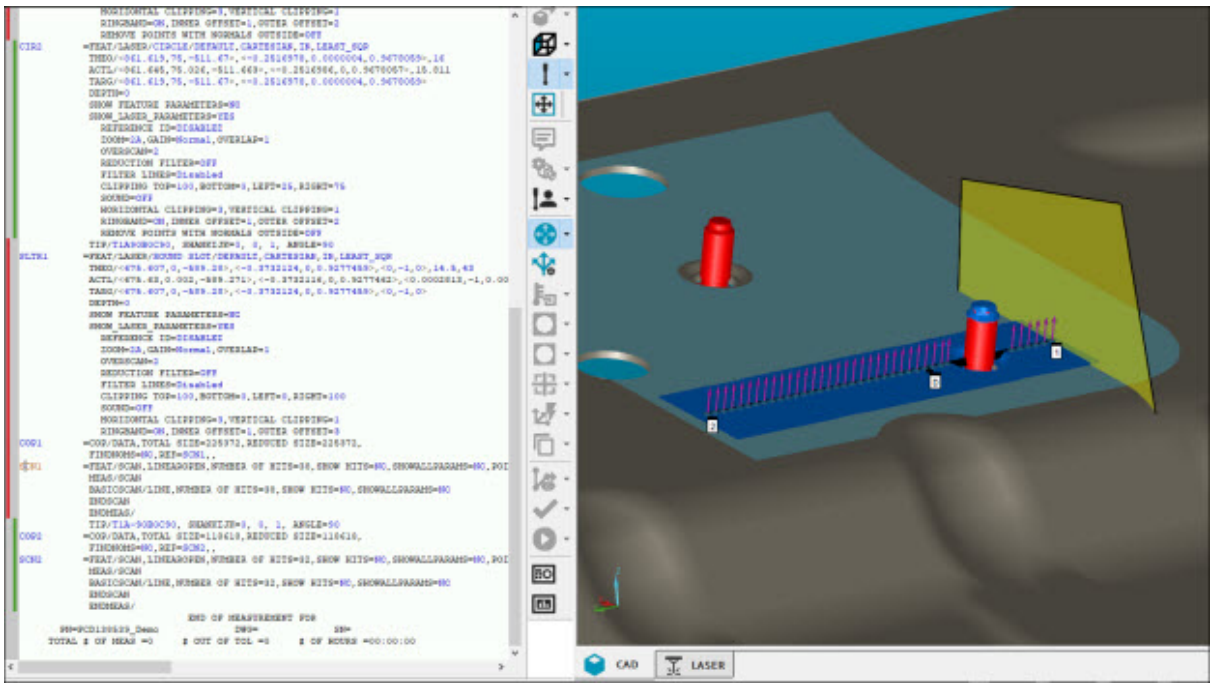


Example of Circle Auto feature

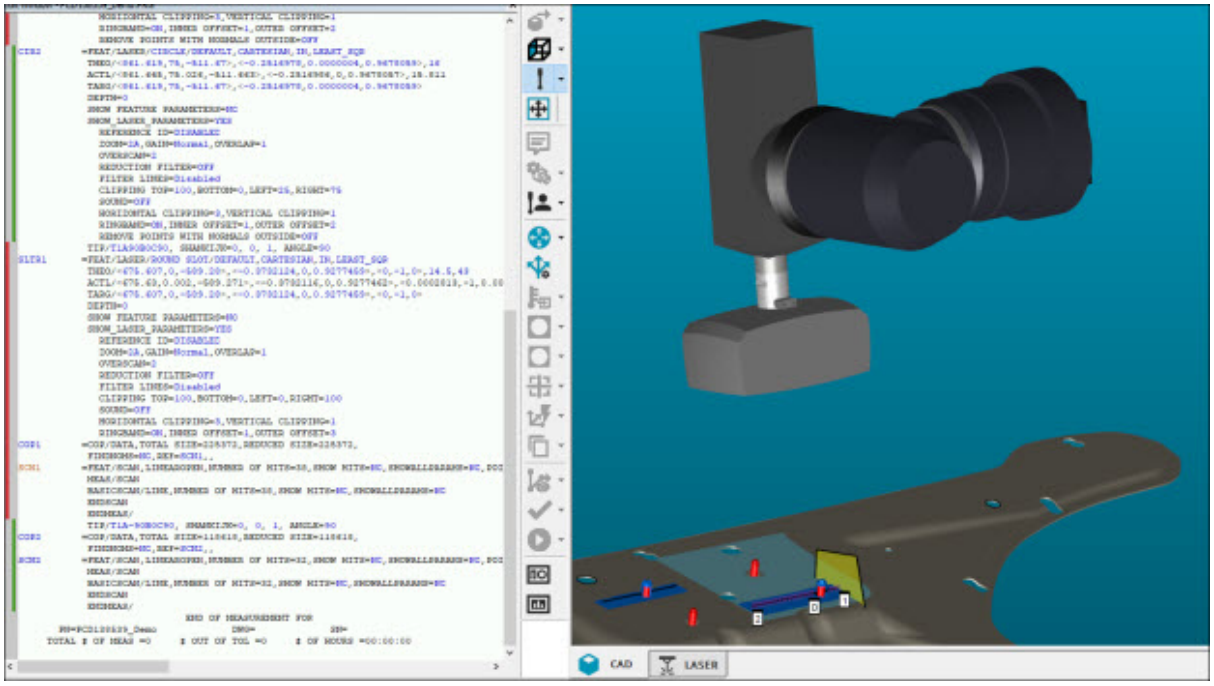
- For a laser scan, PC-DMIS shows the trapezoid that represents the working area of the laser as the start point. The trapezoid moves according to the simulation of the laser stripes. For an example, see the images below:



Example of Linear Open Scan dialog box



Example of linear open scan



Example of linear open scan

If you change the zoom settings (located on the Laser Scan Properties tab) and sensor-based clipping settings (located on the Laser Clipping Region Properties tab), PC-DMIS updates the trapezoid accordingly.

HP-L-10.10 Sensor

To use the HP-L-10.10 sensor, your computer must meet these specifications:

- Maximum number of Operations Per Second (MOPS) - Floating Point Math must be greater than or equal to 59
- Number of Physical cores (P-cores) must be greater than or equal to 8

You can find these values in your computer's processor specifications.

The HP-L-10.10 sensor is a faster, more flexible, and extremely accurate blue-light laser sensor that provides enhanced user guidance features.

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.



HP-L-10.10 laser sensor

A. ***Work Distance Indicator (WDI)*** *- This indicator provides you with a visual status of the HP-L-10.10 sensor, such as if it is in or out of range of the targeted scan area.*

   The indicator also provides you with visual feedback during warm-up and when the sensor is ready to use.

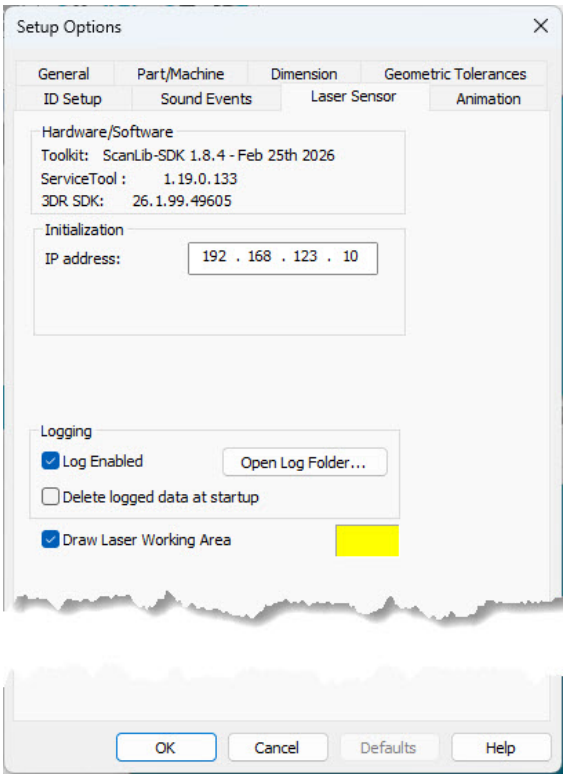
This table details the various WDI status indicator alerts:

WDI Status Color	Description
Flashing Green	Sensor is warming up
Static Green	Sensor is ready
Orange	Sensor is out of range
Flashing Red	Error - sensor is not ready

For more information on using Live View with your HP-L-10.10 sensor, see the "Using Live View with the HP-L-10.10 Sensor" topic in this documentation.

HP-L-10.10 Sensor Setup

Open the **Setup Options** dialog box (**Edit | Preferences | Setup**) and click the **Laser Sensor** tab to show the options for your camera.



Setup Options dialog box - Laser Sensor tab example for the HP-L-10.10 sensor

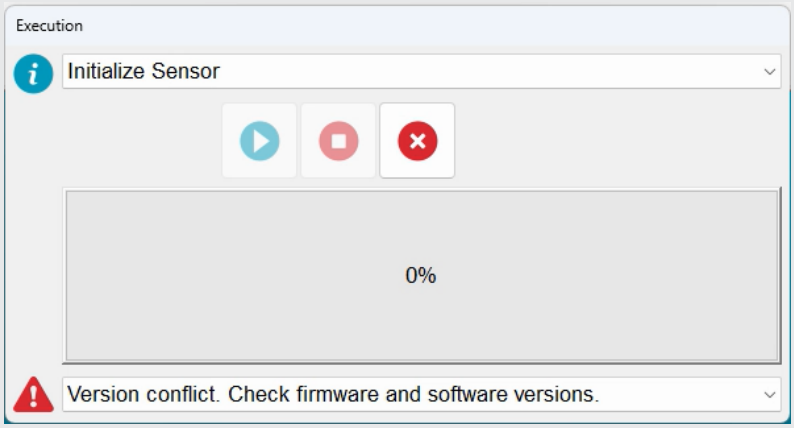
Hardware/Software Area

This area displays the current HP-L-10.10 Toolkit version, Laser Scanner Service Tool version, and 3DReshaper (3DR) SDK version.

Initialization Area

You can use the **IP address** box to define the IP address of the HP-L-10.10 controller.

When you initialize the HP-L-10.10 sensor, PC-DMIS checks that the sensor's firmware and software are compatible with your version of PC-DMIS. If PC-DMIS detects a version conflict, it does not initialize the sensor and shows the "Version conflict. Check firmware and software versions." message in the **Execution** dialog box:



This can happen when you update or downgrade PC-DMIS without updating the sensor's firmware and software to compatible versions. Make sure that your PC-DMIS version, sensor's firmware, and software are compatible before you continue. If you need help finding the correct versions for your system, contact your local Hexagon representative.

Logging Area

Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Hexagon Technical Support can use these files to resolve issues that may involve your laser sensor.

Logging stores debug information on your machine which may slow down normal operations. You should keep logging turned off for normal operations and only turn it on for the time that you need to capture debug information. When done, you should always turn it back off.

- **Log Enabled** - This check box enables or disables data sent to the log files.

- **Open Log Folder** - This button opens up the folder that contains your log files.

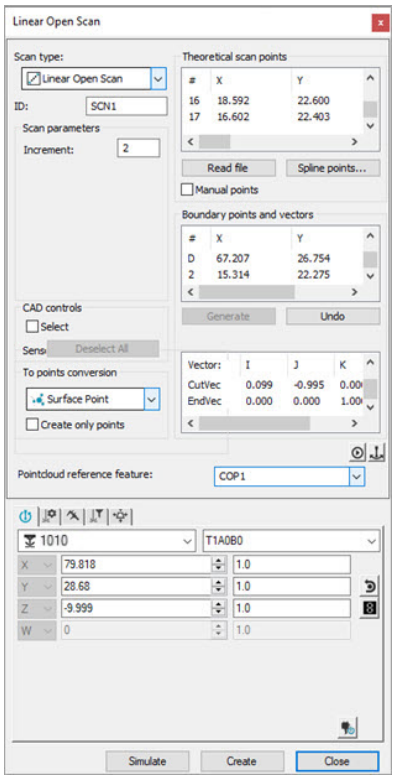
> Typically, PC-DMIS stores these files in the "C:\ProgramData\Hexagon\PC-DMIS\2026.2\NCSensorsLogs\" folder.

- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you restart PC-DMIS.

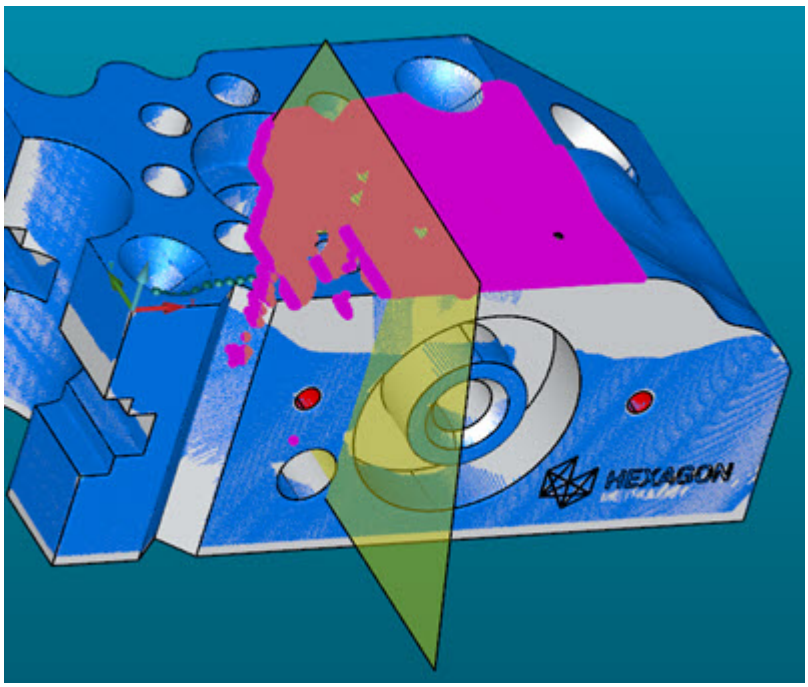
Draw Laser Working Area Check Box

If you select the **Draw Laser Working Area** check box, the HP-L-10.10 probe parameters draw the trapezoid with the correct dimensions. This functionality helps with simulation in Offline mode. This feature is available for laser Auto features and laser scans.

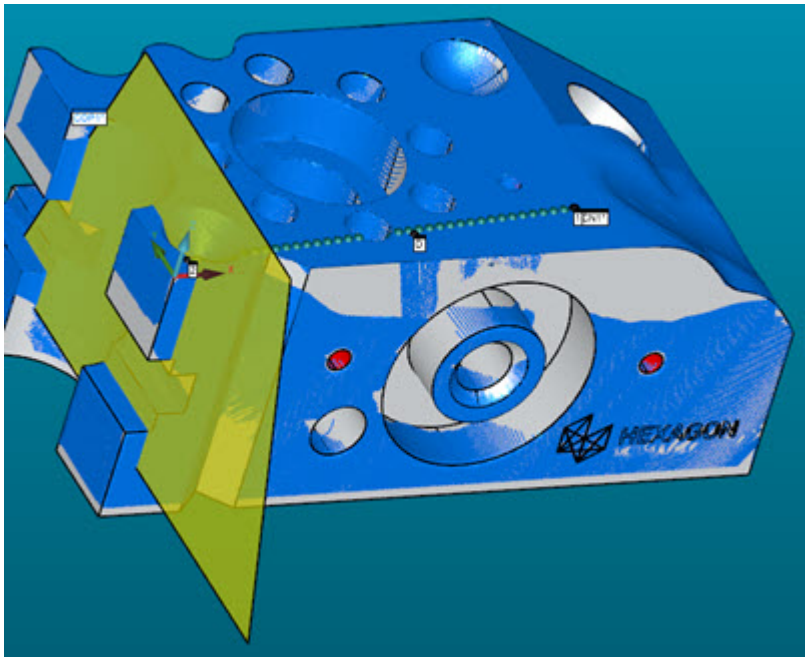
- For a laser scan, the trapezoid that represents the working area of the laser is shown as the start point. The trapezoid moves according to the simulation of the laser stripes. Here's an example of the trapezoid when you click the **Simulate** button:



Example of Linear Open Scan dialog box for the HP-L-10.10 sensor



Example of linear open scan using the HP-L-10.10 sensor, mid-scan

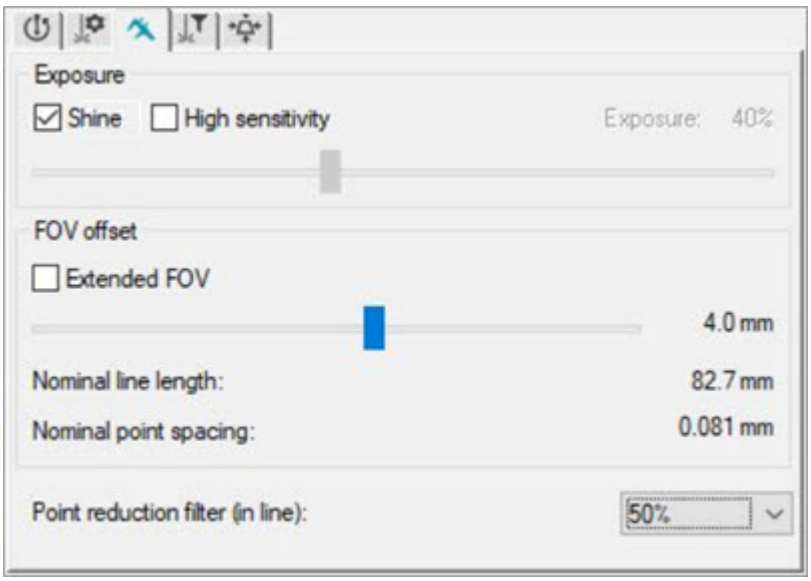


Example of linear open scan using the HP-L-10.10 sensor, scan completed

If you change the sensor-based clipping settings (located on the **Laser Clipping Region Properties** tab), PC-DMIS updates the trapezoid.

HP-L-10.10 Properties

You can adjust the scan properties for the HP-L-10.10 sensor from the **HP-L-10.10 Properties** tab on the Probe Toolbox.



Shine check box and **Exposure** slider (UD mode) - The HP-L-10.10 and HP-L-10.10 LITE sensors offer you two different acquisition modes (Auto and User-defined) to acquire a pointcloud with the highest quality.

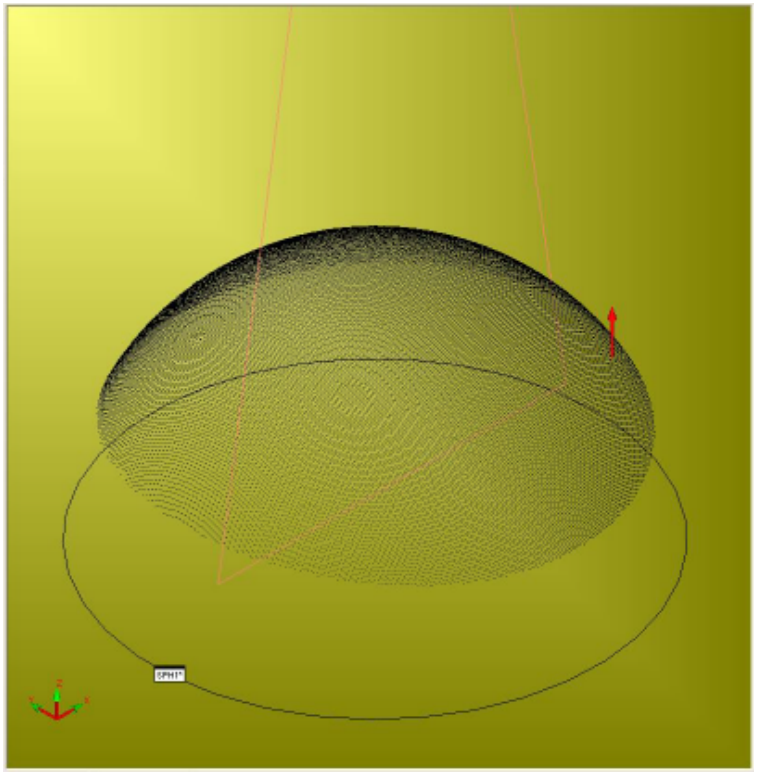
By default, both modes measure with a frequency of 300 Hz for the HP-L-10.10 sensor and 120 Hz for the HP-L-10.10 LITE sensor.

- When you select the **Shine** check box, you are in Auto mode. Auto mode has a high dynamic range. It allows you to measure different surfaces with different surface colors and characteristics without the need of adjusting any settings.
- User-defined (UD) mode is active when you do not select the **Shine** check box. In this mode, PC-DMIS enables the **Exposure** slider and disables the **High sensitivity** check box. The UD mode has a low dynamic range but has the best accuracy in terms of dispersion. In UD mode, you must adjust the exposure setting manually with the **Exposure** slider, or you can use the auto gain function which automatically determines the correct exposure setting for the surface to be measured. In UD Mode, it is difficult to measure different surfaces with different surface characteristics at the same time. You need to adjust the exposure level for each surface variation.

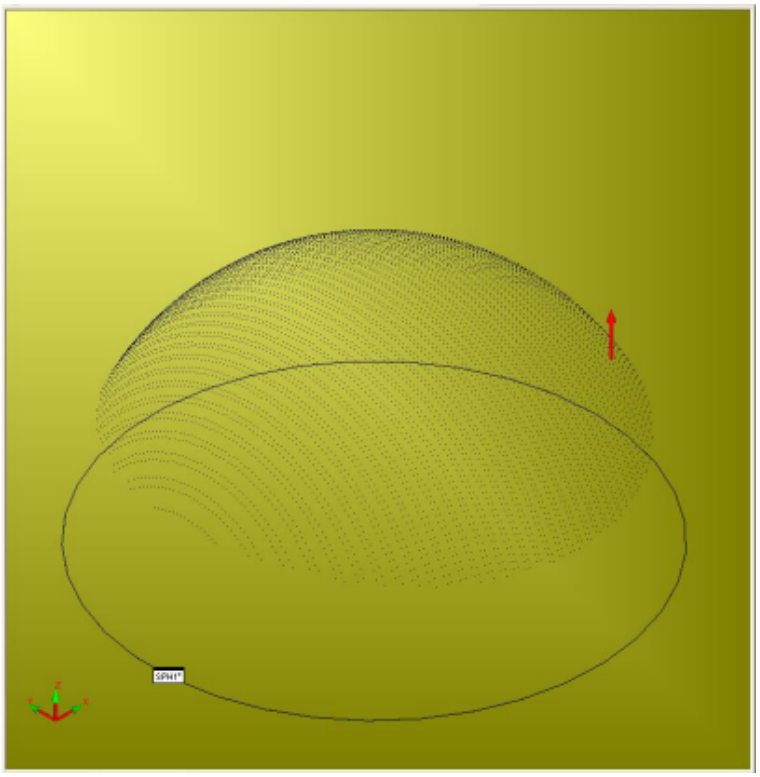
High sensitivity check box - Select this check box if you need to scan highly glossy blue or black parts that are not sufficiently reflective.

Point Reduction Filter list - This list determines whether or not PC-DMIS filters points along the scan line. You can select the desired percentage of total points to filter. If you select the **OFF** option, PC-DMIS acquires the complete data set without any filtering.

Example of Point Reduction Filter Set to OFF

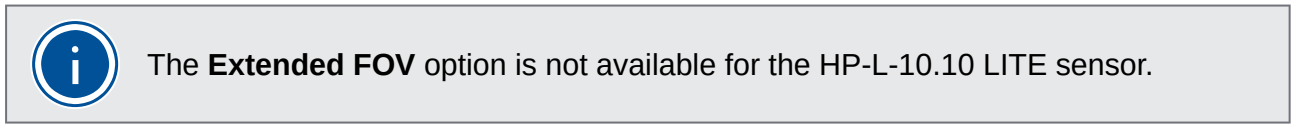


Example of Point Reduction Filter Set to 50%

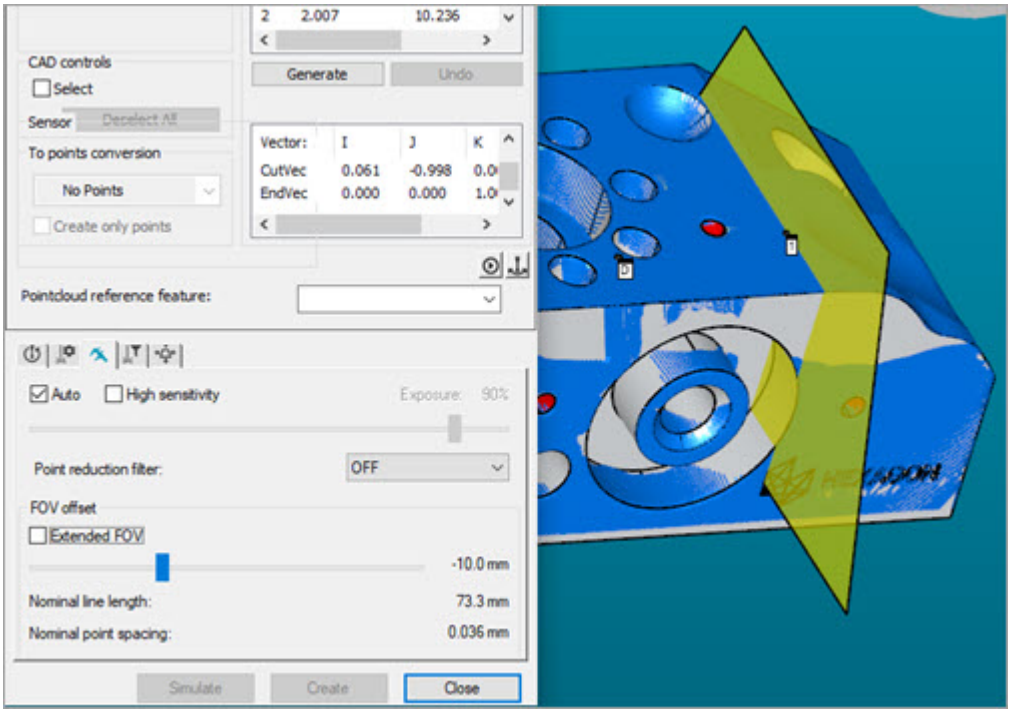


Extended FOV check box and slider - Select the **Extended FOV** check box to extend your field of view (FOV) measurement range an extra 30 mm. When you enable this option, PC-DMIS triggers a new scaling for the **Extended FOV** slider. This allows you to adjust the standoff from -30 to 60 mm. As soon as you move the slider to the Extended FOV location, the software highlights the background yellow. This notifies you that you can expect a

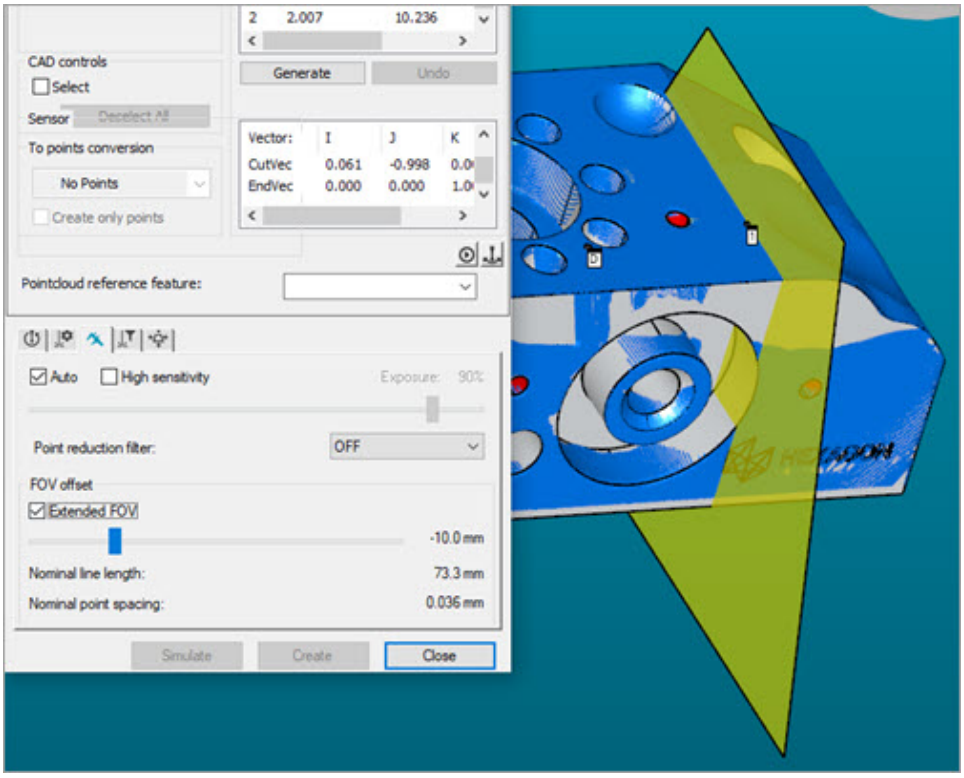
higher dispersion when the standard FOV is left. If you have laser Live View active with **Extended FOV** enabled, PC-DMIS adjusts the FOV accordingly.



The **Extended FOV** option is not available for the HP-L-10.10 LITE sensor.



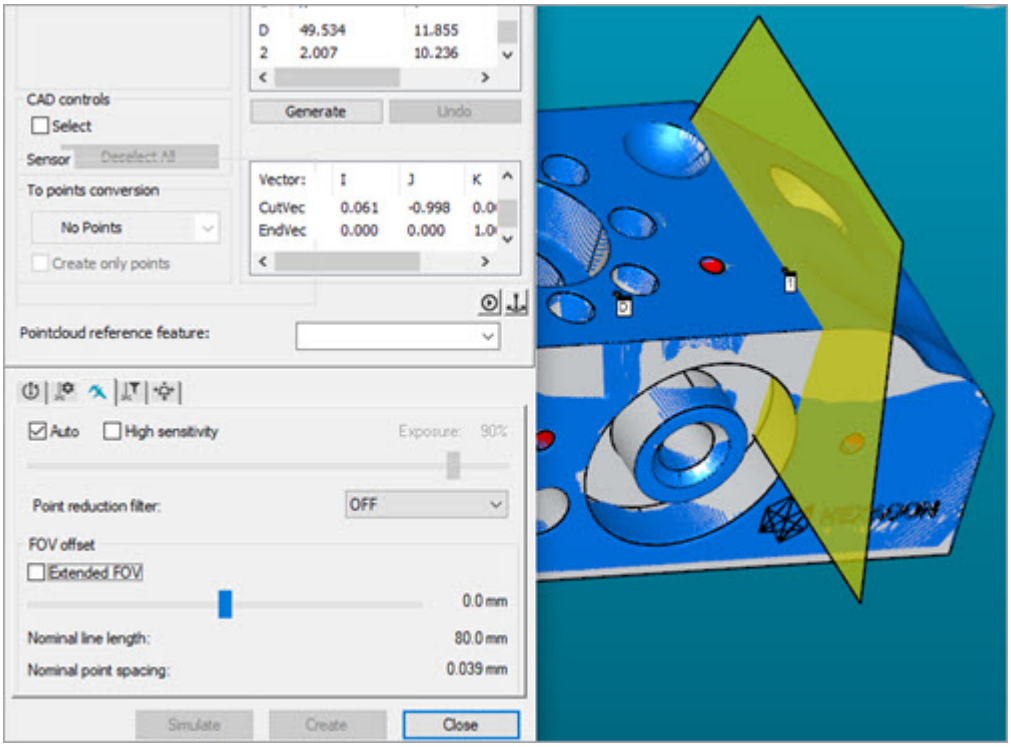
Example with no Extended FOV



Example with Extended FOV

> If you create a scan using the Standard FOV (the **Extended FOV** check box is not selected) and later edit that scan with the **Extended FOV** check box selected, the scan path remains the same. All subsequent executions of the scan are done using the Standard FOV.

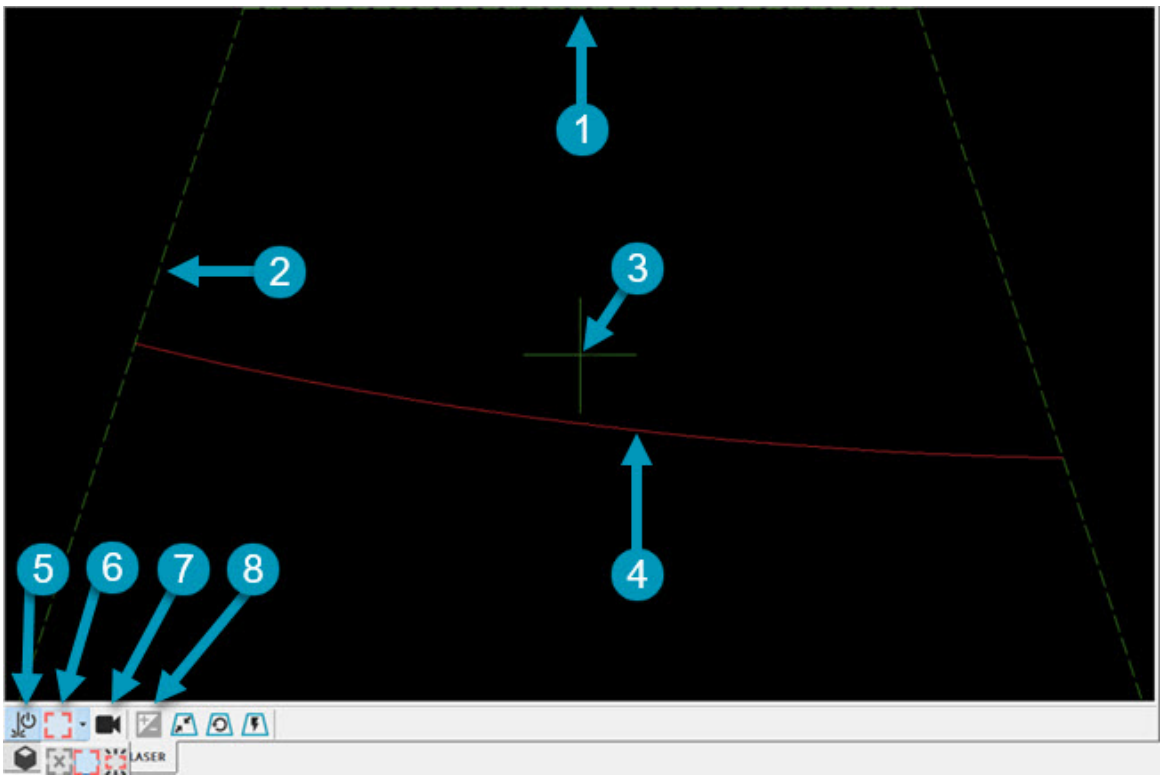
You can use the slider to change the center of the field of view for the scan. Here's an example of the same image above without the **Extended FOV** selected, but the **FOV offset** changed from -10.0 mm to 0 (zero):



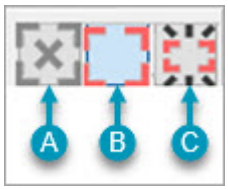
The **Nominal line length** and **Nominal point spacing** values dynamically change based on the position of the FOV slider.

Using Live View with the HP-L-10.10 Sensor

You can use the Live View to display a real-time view of the part to be scanned.



1 & 2 - The border of the Field of View (FOV). Identical to other HP-L type sensors, you can drag and drop these borders to adjust the clipping of the FOV.
3 - Mid-FOV indicator.
4 - Live display of the laser line.
5 - Laser ON/OFF button.
6 - Enable/Disable Visual Guide button.
Click the arrow button to show a drop-down toolbar with these options:



A - Off button: Click this button to turn the Visual Guide off.
B - On button: Click this button to place the Visual Guide in Standard mode where it stays on.
C - Dynamic button: Click this button to place the Visual Guide in Dynamic mode. The Visual Guide will flash based on whether you use the Standard or Extended FOV options, and the position of the laser scan line. For details, see the "The Overview Camera (OVC) and the Visual Guide" topic in this documentation.

7 - Enable/Disable button for the Overview Camera (OVC).
8 - Auto Gain button. You can use it to automatically determine the exposure time when you are in the User-defined (UD) acquisition mode.

The **Extended FOV** option is not available for the HP-L-10.10 LITE sensor.

To turn on Live View, click the **Laser ON/OFF** button  from the **Live View** toolbar.

The Overview Camera (OVC) and the Visual Guide



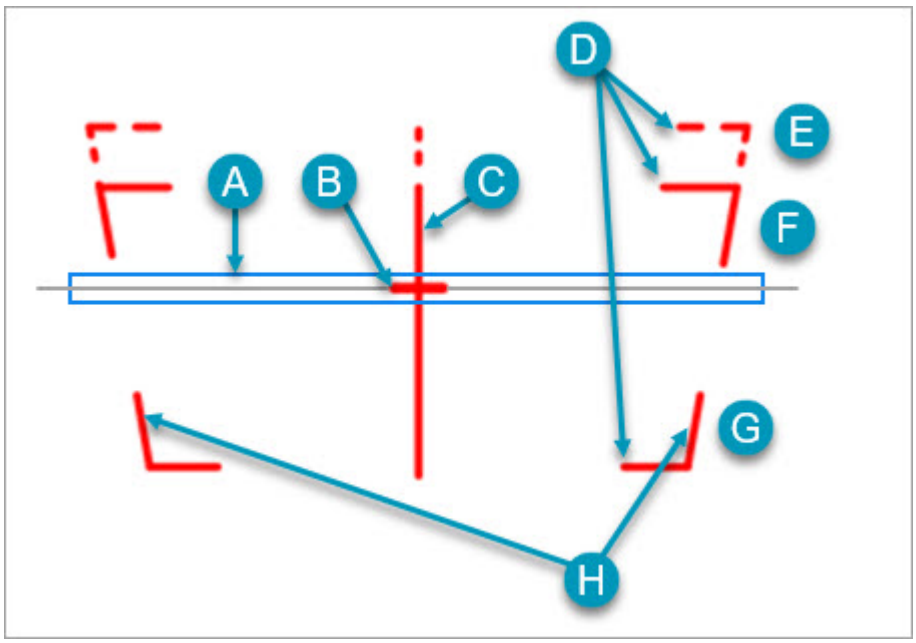
A. **Overview Camera Window** - The Overview Camera Window allows you to receive a live video stream of the part from the sensor. This is especially useful when positioning the sensor on large CMMs or in situations where an area or feature of interest is difficult to see.

   The Overview Camera Window also enhances the reporting capabilities by allowing you to take snapshots when you execute a measurement routine. You can then import those snapshots into your reports.

   With **Operator** comments, you can import these snapshots into your measurement routines to visually guide users when they execute measurement routines.

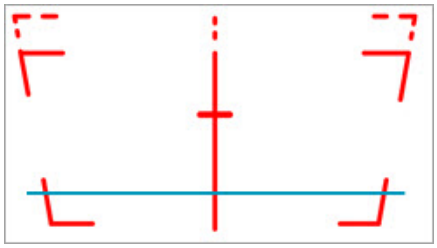
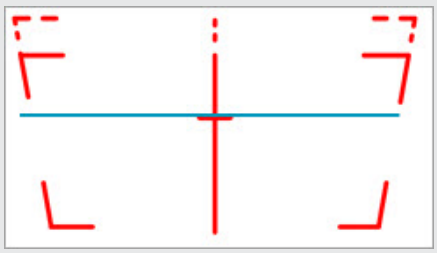
B. **Visual Guide Output** - The Visual Guide Output projects an overlay onto the scan area to show the region on your part or CAD model that the HP-L-10.10 sensor measures. This helps to pinpoint an area or a specific feature of interest when you move the sensor with a jog box or when you perform a measurement in CNC mode.

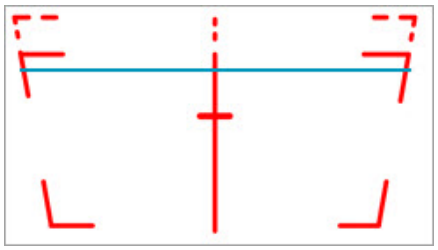
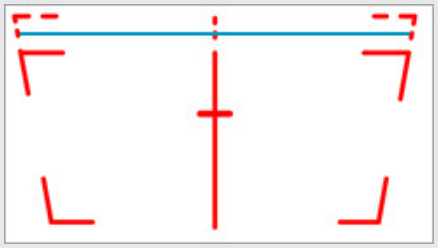
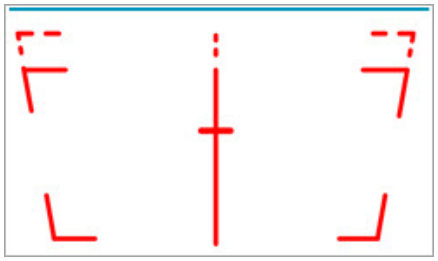
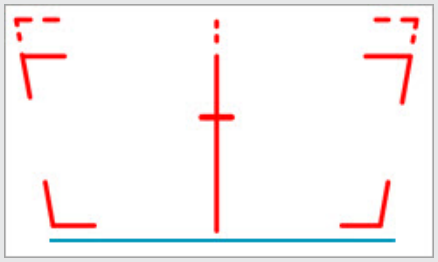
   The components of the Visual Guide are:



A - Laser scan line
B - Stand off indicator
C - Mid-line
D - Distance indicators
E - Extended distance (Stand Off: 90 mm + 60 mm)
F - Far distance (Stand Off: 90 mm + 30 mm)
G - Near distance (Stand Off: 90 mm - 30 mm)
H - Max measuring width indicator

The following table shows examples of how you can use the Work Distance Indicator (WDI), the Visual Guide and the position of the laser scan line to determine if you are in range or not.

Visual Guide Display	Description
	The laser sensor is close to the "near" distance. The Work Distance Indicator (WDI) shows a solid green light to indicate that you are in range.
	The laser sensor is at the "Stand Off" distance of 90 mm. The Work Distance Indicator (WDI) shows a solid green light to indicate that you are in range.

	The laser sensor is close to the "far" distance. The Work Distance Indicator (WDI) shows a solid green light to indicate that you are in range.
	The laser sensor is in the "Extended" distance. If the **Extended FOV** option is disabled, the WDI shows orange to indicate you are out of range. If you enable the **Extended FOV** option, the WDI switches to green to indicate you are now in range.
	The laser sensor is out of range. The WDI shows a solid orange color to indicate that you are out of range.
	The laser sensor is out of range. The WDI shows a solid orange color to indicate that you are out of range.

If you select the **On** button from the **Enable/Disable Visual Guide** drop-down toolbar in Live View, the Visual Guide is always displayed.

If you select the **Dynamic** button from the same toolbar:

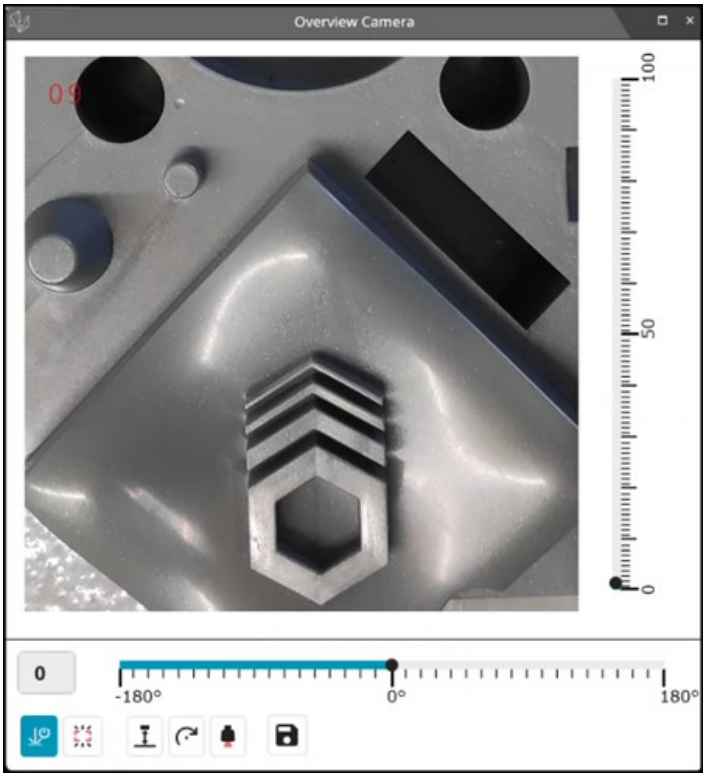
- The Visual Guide blinks if you do not have the **Extended FOV** option selected, and you position the laser sensor within the Extended FOV area (dashed lines) of the Visual Guide.

- The Visual Guide does not blink if you do not have the **Extended FOV** option selected, and you position the laser sensor in the Standard FOV area (solid lines) of the Visual Guide.
- The Visual Guide blinks if you have the **Extended FOV** option selected and you position the laser sensor anywhere outside of the Visual Guide, including the Extended FOV area.
- The Visual Guide does not blink if you have the **Extended FOV** option selected and you position the laser sensor anywhere within the Visual Guide, including the Extended FOV area.

The **Extended FOV** option is not available for the HP-L-10.10 LITE sensor.

The Overview Camera Dialog Box

The **Overview Camera** dialog box lets you see a live video stream of the part from the sensor.



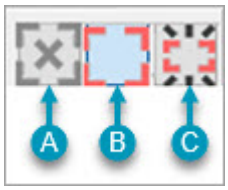
Overview Camera dialog box

You can use the box to set the image orientation or adjust the horizontal slider to set the orientation value from -180 degrees to 180 degrees. You can also use the vertical slider to adjust the brightness value from 0 percent to 100 percent.

PC-DMIS sends the brightness value to the sensor's controller only after you release the mouse button. This means that the brightness does not change dynamically as you move the slider, but only after you release the mouse button.

 **Laser start/stop** - Click this button to turn the laser on or off.

 **Visual guide on/off/dynamic** - Click this button to show a drop-down toolbar with these options:



Drop-down toolbar
*A (**Off**) - Click this button to turn the Visual Guide off.*
*B (**On**) - Click this button to place the Visual Guide in Standard mode, where it stays on.*
*C (**Dynamic**) - Click this button to place the Visual Guide in Dynamic mode. PC-DMIS flashes the Visual Guide based on whether you use the Standard or Extended FOV options and on the position of the laser scan line. For details, see the "The Overview Camera (OVC) and the Visual Guide" topic in the PC-DMIS Laser documentation.*

 **Show/Hide WDI** - Click this button to show or hide the **Work Distance Indicator**, which indicates the sensor's normal distance to the surface.

 **Show/Hide Angle** - Click this button to show or hide the applied rotation of the sensor to the image.

 **Show/Hide Probe Tip** - Click this button to show or hide the sensor on the image.

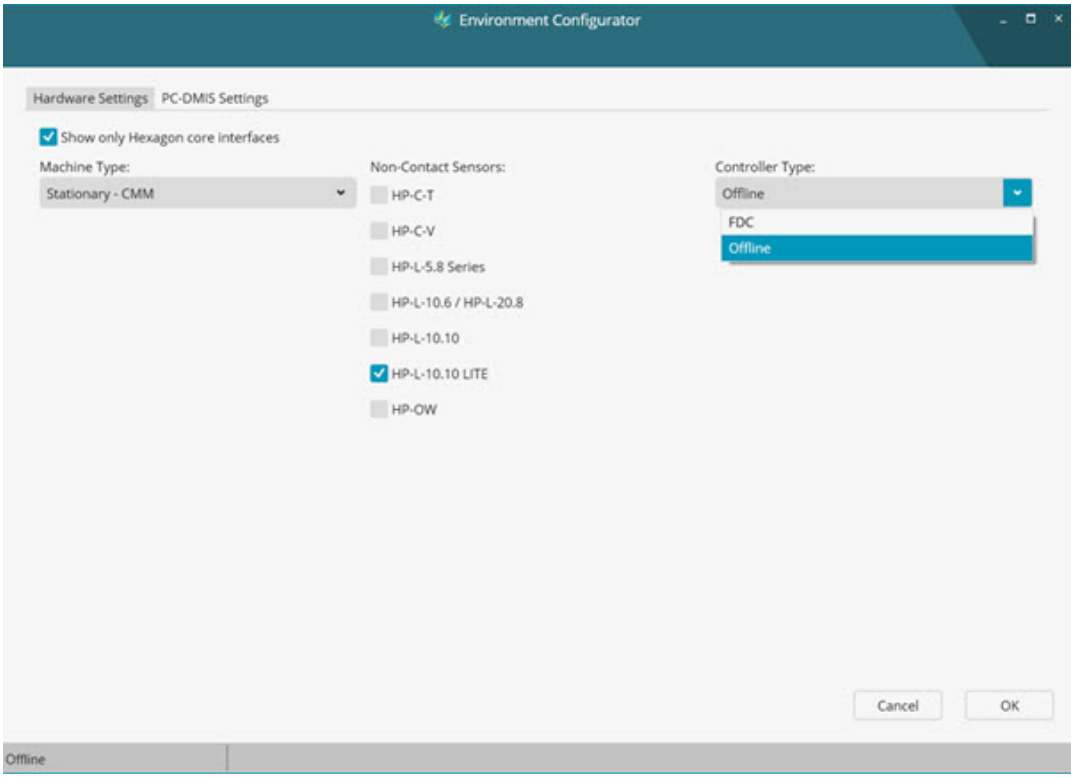
 **Save Image** - Click this button to open the **Save As** dialog box. You can use this dialog box to save captured images as .bmp files.

HP-L-10.10 LITE Sensor



HP-L-10.10 LITE sensor

The HP-L-10.10 LITE sensor is a lighter version of the HP-L-10.10 sensor. You select the probe from the Environment Configurator application.



Environment Configurator application with the HP-L-10.10 LITE sensor selected.

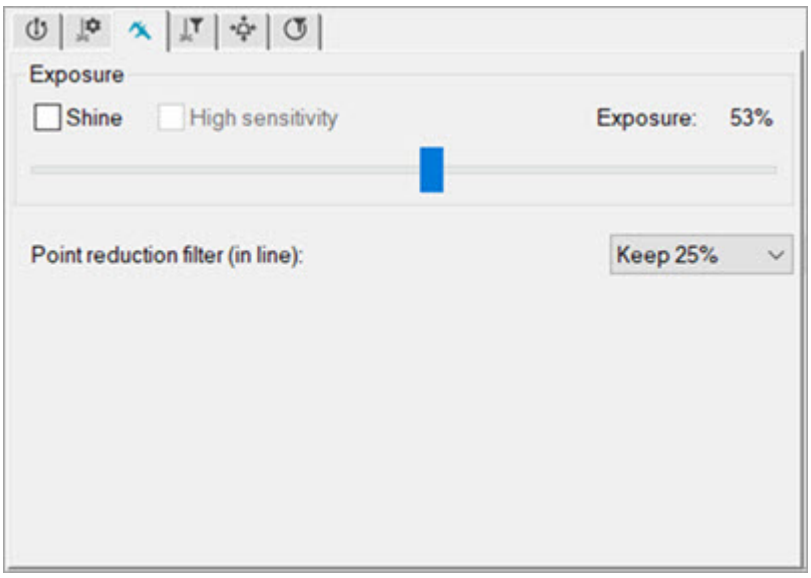
For details on the Environment Configurator application, see the "Environment Configurator" topic in the PC-DMIS Core documentation.

The HP-L-10.10 LITE sensor has a reduced set of functions compared to the HP-L-10.10 sensor as shown in this image of what an Auto Feature and a scan command look like for the HP-L-10.10 LITE sensor:



Example of an Auto Feature and a Scan command using the HP-L-10.10 LITE sensor

There are no changes to the Auto Feature or Scan dialog boxes for the HP-L-10.10 LITE sensor compared to the HP-L-10.10 sensor. Changes do occur with the third tab of the Probe Toolbox which is now called the **HP-L-10.10 Lite Scan Properties**.



Probe Toolbox HP-L-10.10 Lite Scan Properties tab with the Shine option not selected

The options on this tab are:

Shine check box and **Exposure** slider (UD mode) - The HP-L-10.10 and HP-L-10.10 LITE sensors offer you two different acquisition modes (Auto and User-defined) to acquire a pointcloud with the highest quality.

By default, both modes measure with a frequency of 300 Hz for the HP-L-10.10 sensor and 120 Hz for the HP-L-10.10 LITE sensor.

- When you select the **Shine** check box, you are in Auto mode. Auto mode has a high dynamic range. It allows you to measure different surfaces with different surface colors and characteristics without the need of adjusting any settings.
- User-defined (UD) mode is active when you do not select the **Shine** check box. In this mode, PC-DMIS enables the **Exposure** slider and disables the **High sensitivity** check box. The UD mode has a low dynamic range but has the best accuracy in terms of dispersion. In UD mode, you must adjust the exposure setting manually with the **Exposure** slider, or you can use the auto gain function which automatically determines the correct exposure setting for the surface to be measured. In UD Mode, it is difficult to measure different surfaces with different surface characteristics at the same time. You need to adjust the exposure level for each surface variation.

High sensitivity check box - Select this check box if you need to scan highly glossy blue or black parts that are not sufficiently reflective.

Point Reduction Filter list - This list determines whether or not PC-DMIS filters points along the scan line. You can select the desired percentage of total points to filter. If you select the **OFF** option, PC-DMIS acquires the complete data set without any filtering.

Live View

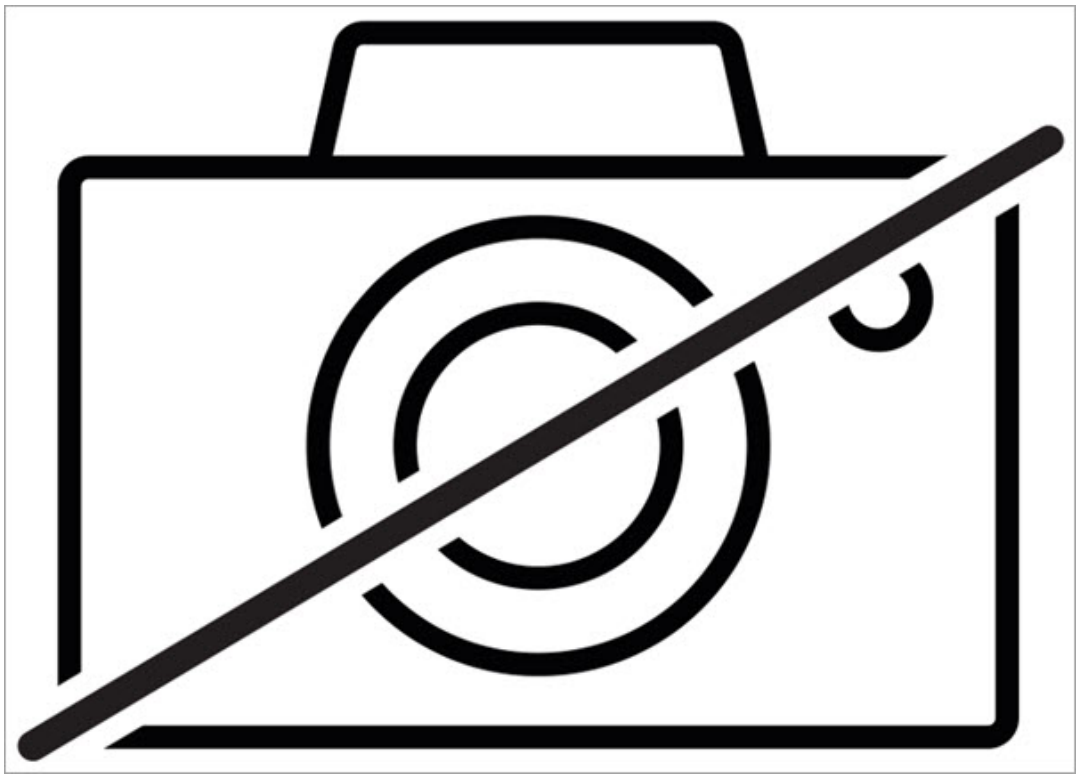
Unlike the HP-L-10.10 sensor, the HP-L-10.10 LITE version has reduced Live View functionality.



HP-L-10.10 LITE Live View toolbar

The limitations of the Live View toolbar for the HP-L-10.10 LITE sensor include:

- The Visual Guide is not available. PC-DMIS disables the button for this option.
- The Overview Camera is not available. The button is enabled but if you click it, PC-DMIS displays this image in the Live View display:



No Overview Camera

Using Zeiss Eagle Eye 2 with Zeiss I++ DME Server

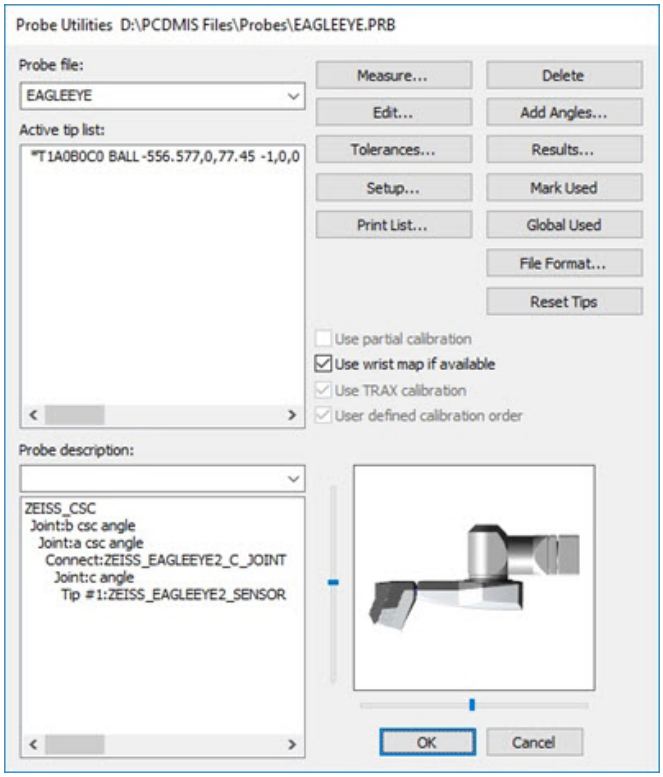
The steps below describe how to use the Zeiss Eagle Eye 2 with the Zeiss I++ DME server.

1. Set up the PC-DMIS I++ client. For details, see "I++ DME Client Interface" in the MIIM documentation.



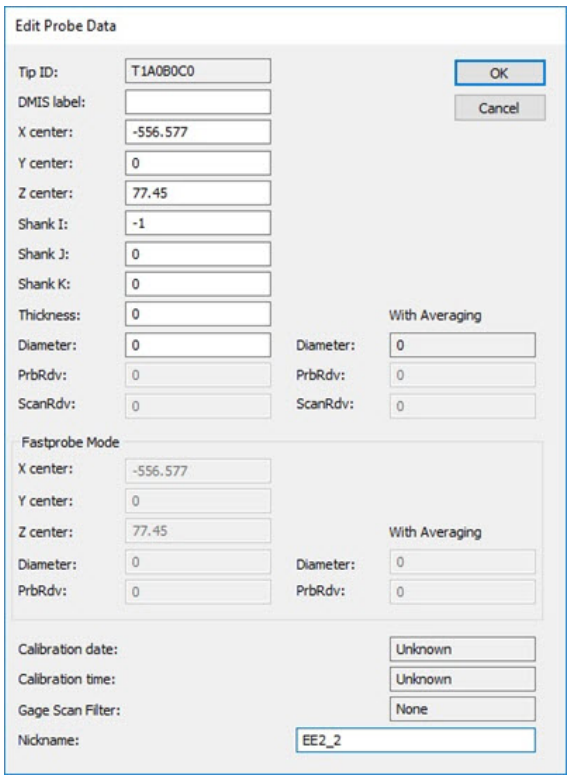
The qualification of the sensor is done inside the I++ DME server.

2. Use the `ZeissWrist` entry to enable the wrist in PC-DMIS. For details, see "ZeissWrist" in the "Option" section of the PC-DMIS Settings Editor documentation.
3. Define the probe assembly.



Probe Utilities dialog box

4. Select the **Use wrist map if available** check box.
5. Select the tip from the **Active tip list** and then click **Edit** to open the **Edit Probe Data** dialog box.



Edit Probe Data dialog box

6. Enter a name in the **Nickname** box for the A0B0C0 tip that corresponds to the probe name given in the I++ DME Server for the EagleEye probe.

7. Set up the probe orientation:

    a. Open the **Setup Options** dialog box (**Edit | Preferences | Setup**).

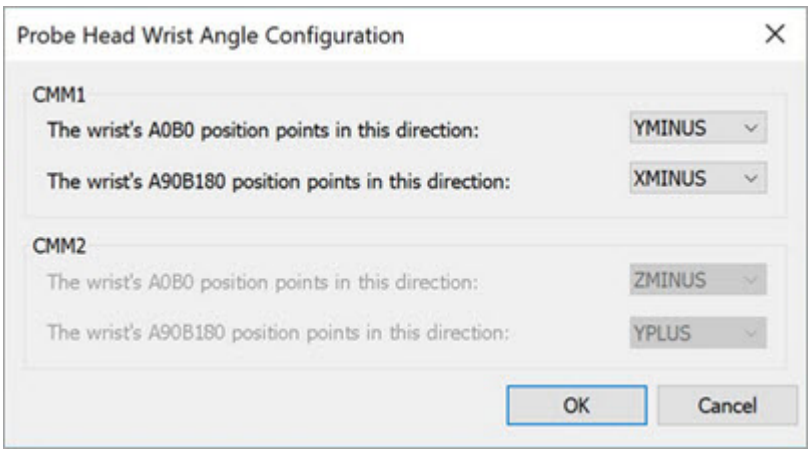
    b. Select the **Part/Machine** tab.

    c. Click the **Probe Head Orientation** button to open the **Probe Head Wrist Angle Configuration** dialog box.

    d. In the **CMM1** area, set these two options:

        - Select the **YMINUS** option from the **wrist's A0B0 position points in this direction** list.

        - Select the **XMINUS** option from the **The wrist's A90B180 position points in this direction** list.

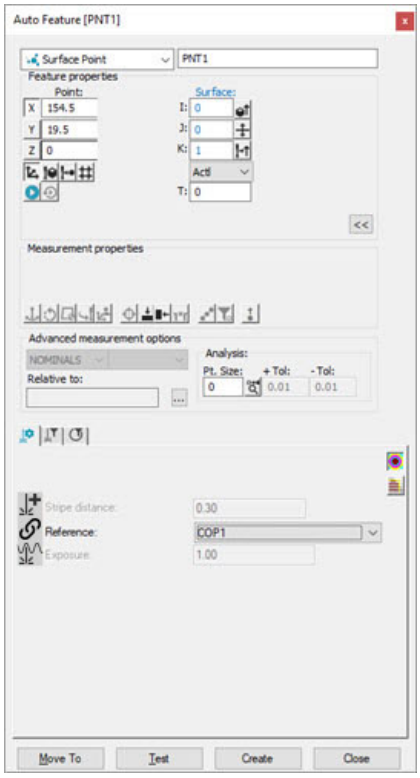


Probe Head Wrist Angle Configuration dialog box

Differences between the Zeiss Eagle Eye 2 and HP-L-10.6

- PC-DMIS does not use the **Laser Sensor** tab in the **Setup Options** dialog box.
- Changes to the **Laser Scan Properties** toolbox tab in the **Auto Feature** dialog box are:
    - For the Eagle Eye 2 measurement, the software hides the **Zoom** and **Gain** properties and adds the **Exposure** and **Stripe distance** properties.
    - **Stripe distance** is the distance between the laser stripes along the path line. Typically, you should use a value between 0.3 and 0.5, inclusive.

- The default value for the **Exposure** setting is 1.0. The valid values are 0.01 to 20, inclusive.



- Changes to the **Laser Scan Properties** toolbox tab on the **Scan Feature** dialog box are:

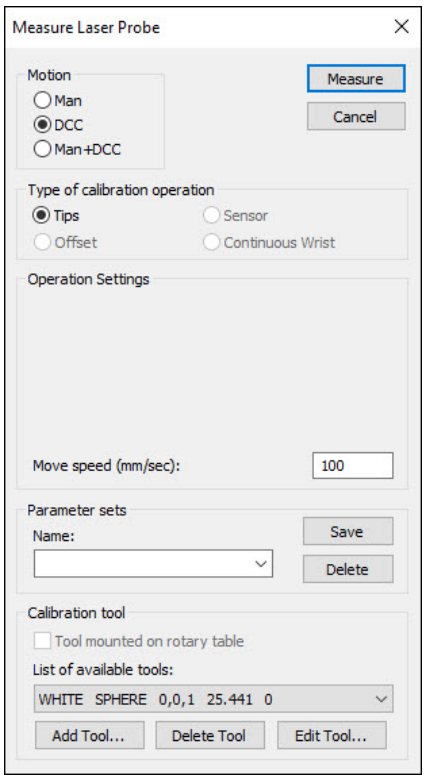
  - For the Eagle Eye 2 measurement, the software hides the **Zoom** and **Gain** properties and adds the **Exposure** and **Stripe distance** properties. The **Scan Feature** dialog box settings are the same as the settings described above for the **Auto Feature** dialog box.

Comparison of HP-L-5.8 and HP-L-10.6 Sensors

This topic describes the similarities and differences between the HP-L-5.8 sensor for CMM and the HP-L-10.6 sensor for DCC.

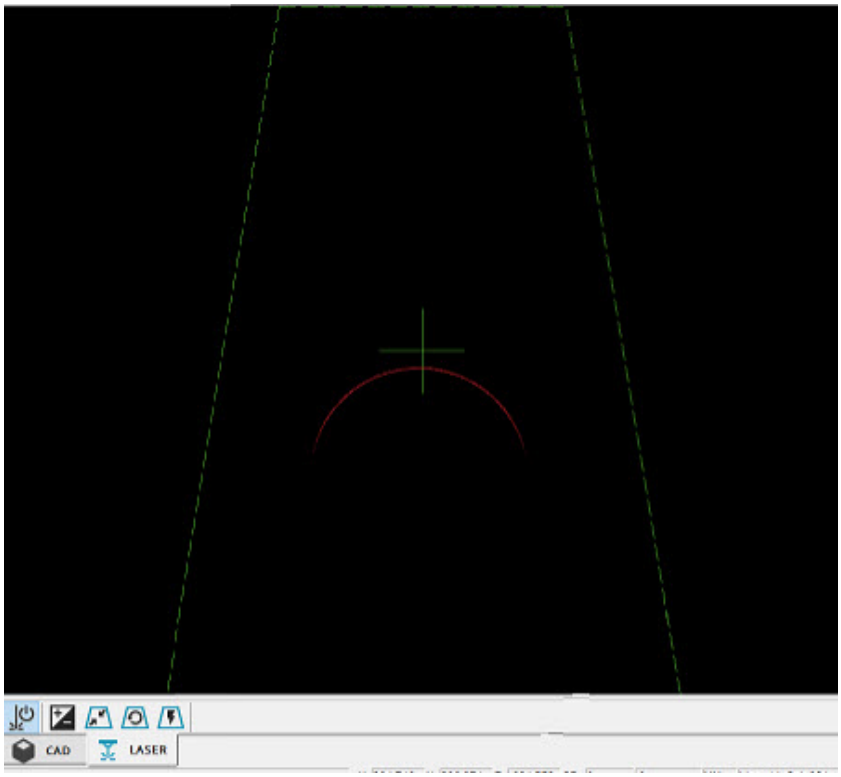
Similarities

- The values in the **Measure Laser Probe** dialog box (**Insert | Hardware Definition | Probe | Measure** button) are the same:



Measure Laser Probe dialog box

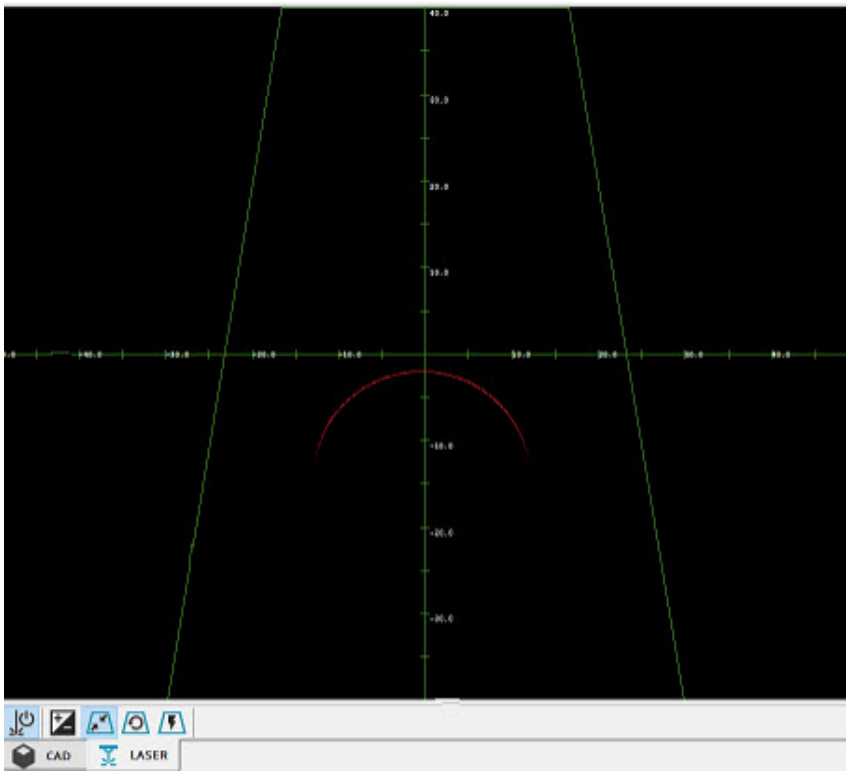
- The X, Y, and Z values on the **Position Probe** tab in the Probe Toolbox are the same.
- The **Laser** tab in the Laser View in the Graphic Display window is the same:



Graphic Display window - Laser tab

Differences

- The shape of the sensor is different.
- The related components in probe.dat are different.
- The standoff distance, and the sensor's field of view (that is, the sensor's geometry) are different:



Graphic Display window - Laser tab

- For the HP-L-5.8, the **AutoGain** button appears in the Laser View in the Graphic Display window. When the HP-L-5.8 sensor is in range on a part, you can select the button to learn the best gain setting and update the Probe Toolbox accordingly. You can also use this functionality while you set up laser Auto features and laser scan properties. For more information about how to set up these properties, see "Creating Auto Features with a Laser Sensor" and "Scanning Your Part Using a Laser Sensor" in the PC-DMIS Laser documentation.
- The default value for the **Increment 2** option (the increment distance between scan lines) in the **Scan parameters** area for a Patch Advanced Scan is 45 mm for the HP-L-5.8 (the HP-L-10.6 has a different default value).
- The differences on the **Laser Scan Properties** tab in the Probe Toolbox in the **Auto Feature** dialog box are as follows:
  - The HP-L-5.8 has only one scan zoom state, so the dimension of the field of view is fixed. (There are no green option buttons on the **Laser Scan Properties** tab as there are for the HP-L-10.6 and HP-L-20.8.)

- For the HP-L-5.8, there are five sensitivity modes (**1**, **2**, **3**, **4**, and **5**) in the **Gain** list on the **Laser Scan Properties** tab. When you select a mode, the image in the Laser View updates in real-time. You can also select the **Quality Filter** icon next to the **Gain** list to enable or disable the Quality Filter mode.

Step 4: Calibrate the Laser Sensor

The calibration process described in this step may vary depending on the options you select for laser sensor measurements and the type of installed interface. For detailed information on laser sensor calibration options, see the "Measure Laser Probe Options" topic.

Calibrating Portable HP-L Laser Sensors

Use this procedure to calibrate a portable laser HP-L sensor using a planar artifact:

1. From the **Probe Utilities** dialog box, click **Measure** to open the **Measure Laser Probe** dialog box. For information on this dialog box, see "Measure Laser Probe Options".
2. Select the appropriate sensor mode. The default is **Zoom2A**.
3. Place the planar artifact in a convenient location for the arm to measure.
4. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions.
5. The calibration procedure requires that you take 17 laser stripes on the planar artifact in different positions and orientations with respect to the planar artifact. To help you visualize where to take the stripe, the system draws a yellow target line on the **Laser** tab in the Graphic Display window.

Calibrating DCC HP-L Laser Sensors

> You must keep the feed knob on the jog box at 100% for the entire calibration process. If you lower the feed knob on the jog box to less than 100%, you reduce the accuracy of your system.

The calibration process described in this step may vary depending on the laser sensor options and the type of installed interface. For detailed information on calibration options, refer to the "Measure Laser Probe Options" topic.

Use this procedure to calibrate your laser sensor for the first time:

1. Select the menu option **Insert | Hardware Definition | Probe** to open the **Probe Utilities** dialog box.
2. From the **Active Tip List** box, select the tip that you defined in Step 2.

3. Click **Measure** to open the **Measure Laser Probe** dialog box (for information on this dialog box, see "Measure Laser Probe Options").

4. Define other calibration options: **Parameter Sets** and **Calibration Tool**.

> If you use a multi-sensor CMM with both a contact probe and a laser probe, you must first locate the calibration sphere using a calibrated contact probe before you calibrate the laser probe on the same sphere. This establishes the correct relationship between the contact and laser measurements for all probes after you complete the calibration process.

5. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions. The first several prompts that you see are identical to the setup procedure for touch-trigger probes.

> If you answer **Yes** to the message "Has the sphere moved", you need to bisect the qualification sphere. For information, see "Bisecting the Calibration Sphere". Once you do an Offset calibration, the software no longer asks you to bisect the sphere unless you answer **Yes** to the message "Has the sphere moved".

6. If needed, click **Add Angles** to define any other tip angles that you need to calibrate.

7. From the **Active Tip List** box, select the tips that you want to calibrate.

8. Click **Measure** to open the **Measure Laser Probe** dialog box. If you don't select any angles, the software asks if you want to calibrate all the tips.

9. Select the **Tips** option.

10. For **Calibration Tool**, select the same tool that you used earlier.

11. Click **Measure** to begin the tip calibration. Once calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.

> Certain sensor tip angles may cause the laser beam to fall on a portion of the calibration tool's stem. In some cases, the standard deviation for the sensor calibration of those tips exceeds the expected amount. In those cases, PC-DMIS displays a message to ask if you want to repeat the calibration of those tips. If you click **Yes**, the system uses the offsets and orientation determined by the first measurement rather than the theoretical values. This results in a clipping around the target that is more accurate during this re-calibration.

Calibrating CWS/WLS Sensors

You can calibrate the CWS tip offset on a sphere. Sphere tools with a less reflective surface work better than those with a highly reflective surface. Calibration is supported on fixed mount multi-sensor machines and on indexing wrists with a TKJ connector.

The calibration executes using the current temperature compensation.

The measurement range of most CWS probe heads is small. This can mean the manual point taken when the tool has moved or when using the Manual+DCC motion must be very close to the sphere pole or nearest point for the calibration to execute successfully.

During calibration execution, the machine automatically moves to the center of the CWS measurement range, or to the needed measurement range position for each point.

PC-DMIS does not support multiple wrist angle tip calibration in a single calibration operation. You must calibrate each tip separately.

When you calibrate a wrist angle tip for the first time and the tool has not moved, select Man+DCC. For all subsequent measurements of this tip, select DCC.



There are no automatic clearance moves before or after the calibration measurement sequence. Ensure there's sufficient clearance for any wrist rotation needed to position the wrist for the specified tip before starting the calibration. Ensure that there is sufficient probe clearance for the move from the probe's current position to the measurement start position.

The following steps outline the procedure to calibrate your laser sensor for the first time:

1. Select the **Insert | Hardware Definition | Probe** menu item.
2. In the **Probe Utilities** dialog box, define the CWS probe and tip.
3. Select **Measure** to open the **Calibrate Probe Offset** dialog box.
4. From the **Calibrate Probe Offset** dialog box, configure the settings and click **Calibrate**. PC-DMIS displays a message to verify if the qualification tool has moved or if the machine's zero point has changed.
    - If you select **Yes**, PC-DMIS displays the **Execution** dialog box and prompts you to take a manual point. The point should be at the top or at the nearest point of the sphere from the perspective of the probe and probe vector.
    - If you select **No**, PC-DMIS displays the **Execution** dialog box and begins DCC measurement.
5. After the calibration measurement completes, click **Results** in the **Probe Utilities** dialog box to see the detailed results.

Mapping Infinite Wrist DCC HP-L Laser Sensors

The hardware configuration of an HP-L laser sensor and an infinitely indexable wrist, such as the CW43L, has the ability to qualify infinite tip orientations through a Laser Wrist Map (LWM).

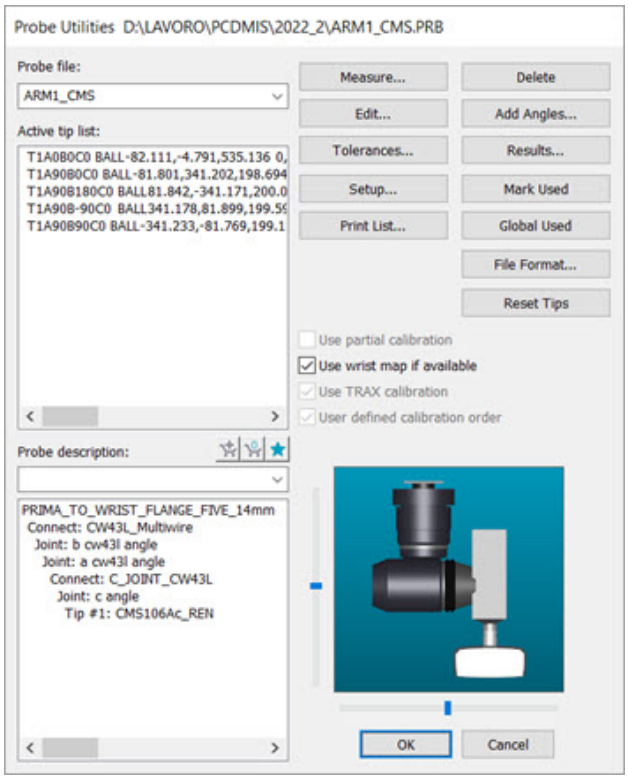
Once you create the LWM for a specific sensor, if you add new tips to the sensor, they are automatically qualified and you can begin measuring.

> **!** You need to recreate the LWM each time a component of the wrist changes (for example, when the CJoint changes). Also, refer to your hardware and vendor information for the appropriate times to map a wrist since this can change based on the construction of the device and the manufacturer's recommendations.

The following steps outline the procedure for mapping infinite wrist DCC HP-L laser sensors:

1. Define the sensor:
    a. In the **Probe Utilities** dialog box, create a sensor as indicated below:
        - Infinitely indexable wrist, such as the CW43L
        - CJoint
        - HP-L laser sensor

    For example:

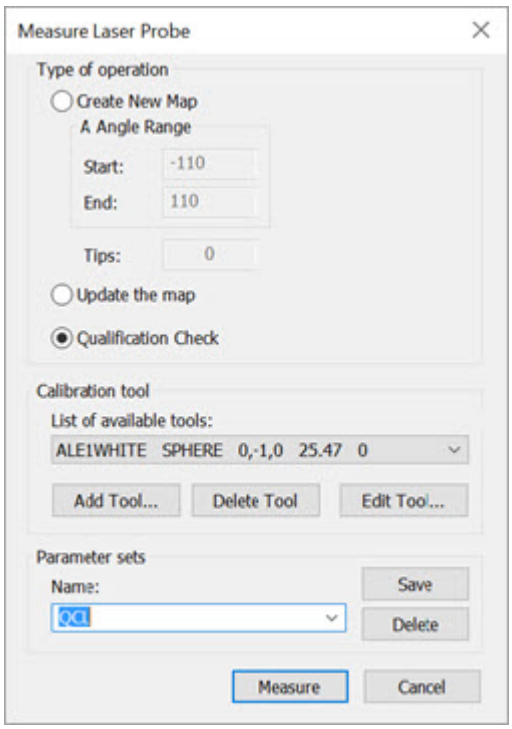


Sample Probe Utilities dialog box with an HP-L laser sensor and a continuous wrist

b. Select the **Use wrist map if available** check box.

c. Click **Measure** to display the **Measure Probe** dialog box.

For example:



Measure Laser Probe dialog box for the CW43L sensor

2. Create the map:

a. From the **Measure Probe** dialog box, select the **Create New Map** option.

b. For **A Angle Range**, type the desired **Start** and **End** values according to the available wrist.

> The B and C angles are always mapped within the full physical range (typically, -180 to +180 degrees).

The **Tips** box displays the total number of tips measured to create the map.

c. Click **Measure**.

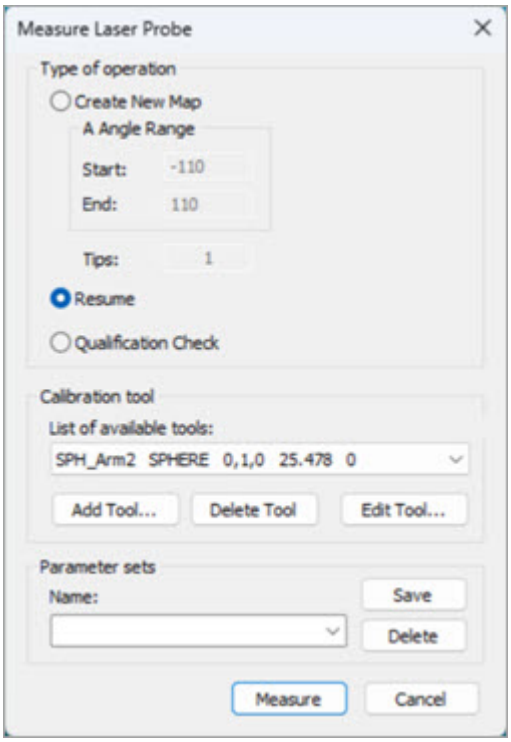
- PC-DMIS measures five sensor orientations around the sphere tool.
- PC-DMIS measures all of the tips in the mapping grid.

If math errors occur during the creation of a new map, the mapping continues.

When PC-DMIS completes the map creation, and if the map validation detects missing nodes during the mapping process, for example caused by math error occurrences, there are two methods to complete the mapping for the missing data:

- **Method 1**: From the **Measure Laser Probe** dialog box, select the **Resume** option to show the number of missing tips. To open the **Measure Laser Probe** dialog box, click the **Measure** button on the **Probe Utilities** dialog box.

  With the **Resume** option selected, click the **Measure** button on the **Measure Laser Probe** dialog box to complete the mapping of the tips with the missing map data.



Measure Laser Probe dialog box with the Resume option selected

- **Method 2**: From `Autocalibrate` commands in the Edit window, it is now possible to complete the creation of a new wrist mapping even when some math errors occur during the mapping process. This only applies if the math errors are sporadic and not systematic. To do this, execute the following procedure in sequence:

  a. Run the initial `Autocalibrate` command with the **Create New Map** option enabled in the parameter set.

  b. Run all subsequent `Autocalibrate` commands with the **Resume** option enabled in the parameter set.

Updating an Existing Map

Once you create the map, you can recover the correct qualification for all of the tips whenever a geometrical or thermal parameter of the Sensor - Wrist system changes. For

example, after the sensor experiences a physical collision, or when the room temperature changes.

To recover the correct qualification:

1. From the **Measure Probe** dialog box, select the **Update the map** option.
2. Click **Measure**. PC-DMIS starts to re-measure the same five sensor orientations around the sphere tool as it measured during the map creation process.

Performing a Qualification Check

Use this option to re-measure the tip orientations that you specify within the selected probe file. PC-DMIS performs a comparison to the previously measured data for these tip orientations. You can then use this comparison to determine if a complete map update is needed. This is an audit-only procedure within the selected probe file and does not update the tip offsets.

Defining Parameter Sets for Mapped Wrists

You can define a parameter set to create or update a wrist map, as well as to run a Qualification Check. You can also use the `AUTOCALIBRATE` command within a measurement routine to perform the same operations.

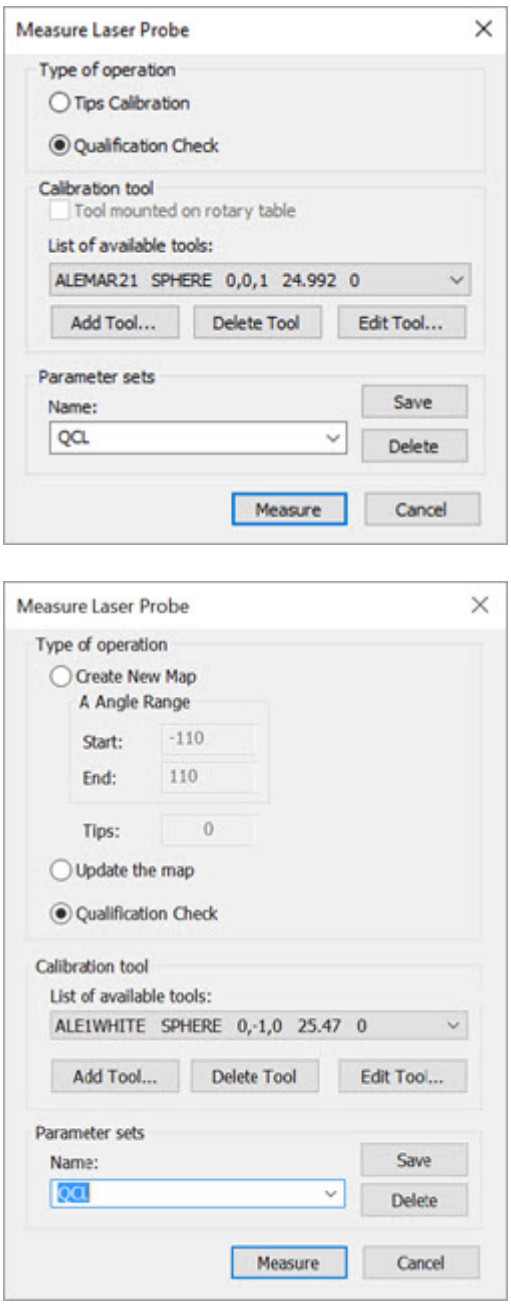
To define a parameter set:

1. In the **Measure Probe** dialog box, select and enter the desired values.
2. In the **Probe Utilities** dialog box, select the desired tips to check.
3. In the **Name** box, type a name for the parameter set.
4. Click **Save**.
5. To close the dialog box, click **Cancel**.

For more information about parameter sets and how to use the `AUTOCALIBRATE` command, see "Dual Arms with Wrists Calibration Example" in the PC-DMIS Core documentation.

Measure Laser Probe Options

The options on the **Measure Laser Probe** dialog box determine the procedure that the software uses for the laser sensor calibration. To access this dialog box, open the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), and then click **Measure**.



Measure Laser Probe dialog box for an indexable head (top) and the CW43L (bottom)

Change the options as needed or as directed in "Step 4: Calibrate the Laser Sensor".

Types of Calibration Operations

The options in this section are available based on the laser sensor. **Tips** works for all probes.

- **Tips Calibration** - Use this option to calibrate the tips for your laser sensor.
- **Qualification Check** - Use this option to re-measure the tip orientations that you specify within the selected probe file. PC-DMIS performs a comparison to the previously measured data for these tip orientations. You can then use this

comparison to determine if a complete calibration is needed. This is an audit-only procedure within the selected probe file and does not update the tip offsets.

If you use a mapped CW43L continuous wrist, these options become available:

- **Create New Map** - This option creates a new complete wrist map.

  > This is an operation that should be done when your machine was initially set up. It should only be re-done when service intervention occurs. Depending on your machine's configuration and speed, this operation can take up to 8 hours to complete.

- **Update the Map** - This option updates an existing map. You can use this option when a probe crash occurs that does not need service operation support, or when you get larger than expected Qualification Check return errors.

Parameter sets

Parameter sets allow you to create, save, and use saved sets for your laser sensor. PC-DMIS saves this information with the probe file, which includes the settings for your laser sensor.

To create your own named parameter sets:

1. Modify any parameters on the **Measure Laser Probe** dialog box.
2. From the **Parameter Sets** area, in the **Name** box, type a name for the new parameter set, and click **Save**. To delete a saved parameter set, select it, and click **Delete**.

Calibration Tool

Select the appropriate calibration tool. If this is your first calibration, you need to click **Add Tool** to first define the calibration tool. For specific information on defining a qualification tool, see the "Defining Hardware" chapter of the PC-DMIS Core documentation.

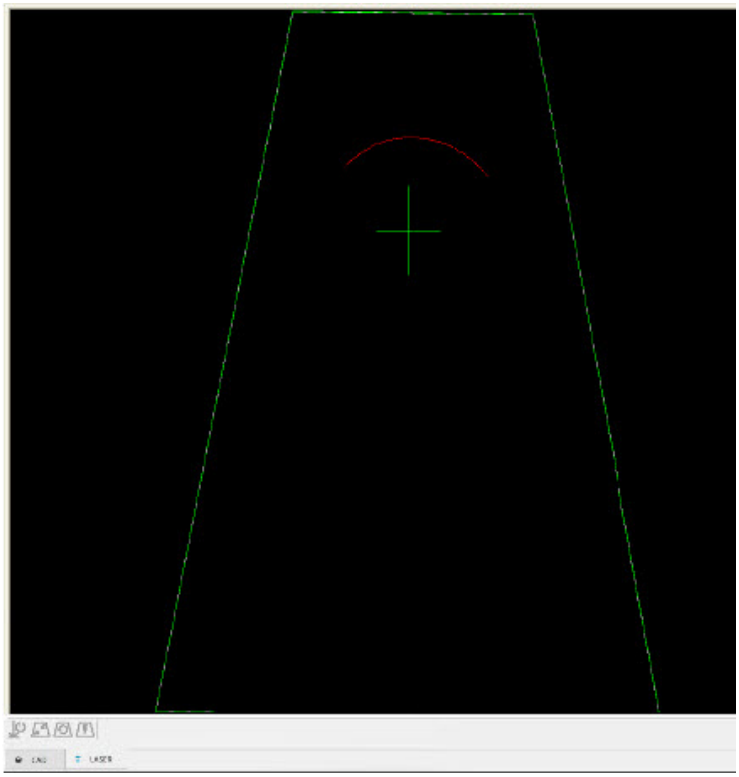
> Make sure you use the spherical qualification tool that comes with your laser sensor. The surface characteristics of this tool are designed for optimal scanning results. If you use a tool made by another manufacturer, it may produce inaccurate data.

Manually Bisecting the Calibration Sphere

If you moved the qualification sphere or do not know its location, you must manually bisect it before you start the qualification. During this process, PC-DMIS prompts you when to move the machine. The prompt instructs you to bisect the calibration sphere and position the sphere in the center of the keystone.

To manually bisect the sphere:

1. Leave the PC-DMIS Message open.
2. Switch to the **Laser** tab on the main Graphic Display window.
3. Click the **Start/Stop** button. This turns on the laser. A flashing red arc appears in the graphic area of the **Laser** tab and a green crosshair. The red arc is where the laser hits the calibration sphere.
4. Center the cross hair inside the circular region formed by the arc by moving the machine with the jog box. The red arc moves as you move the machine. If you imagine that the flashing arc indicates the edge of a circle, the center point of this imaginary circle should optically align with the center of the cross hair.



Aligning the arc

5. Once you have aligned the arc, click the **On/Off** button again. This turns off the laser.
6. Click **OK** in the PC-DMIS Message to accept the change you made of aligning the arc. PC-DMIS stays in Execute mode, and the laser sensor moves through a series of defined positions used to calibrate your tip.
7. At each position, the laser beam hits the sphere in a stripe and the laser sensor collects the data from that stripe. The collected data and the corresponding machine position determine the sensor's mounting orientation on the machine.
8. Once execution finishes, PC-DMIS returns to Learn mode and shows the **Probe Utilities** dialog box.

HP-L Automatic Self-Centering of Tool Sphere

The HP-L laser sensor provides automatic self-centering (bisecting) of the calibration tool sphere during calibration. This happens if you click **Yes** when PC-DMIS displays the message "Has the sphere moved?". From the Graphic Display window, click the **Laser** tab. You can drive the laser sensor to the center of the sphere.

You have two possibilities at this point:

- You can manually bisect the sphere and bring it in the center of the keystone and then click **OK** to start the laser calibration.
- Display a portion of the calibration sphere in the Laser View and then click the **Probe Enable** button to automatically center the sphere. When completed, press the **OK** button to complete the laser calibration.

When PC-DMIS determines that the calibration sphere has been moved, PC-DMIS displays a message to confirm whether you are ready for the calibration.

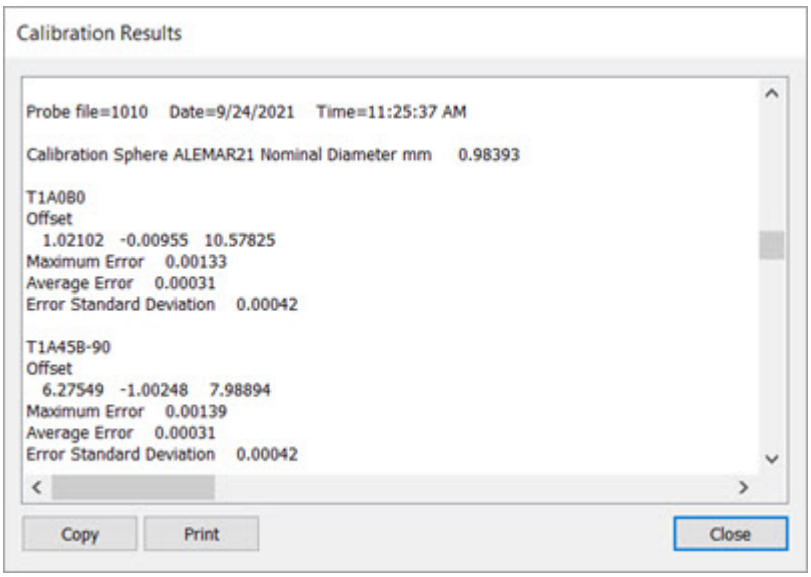
Follow the instructions as described in the message box. Click the **OK** button when done.

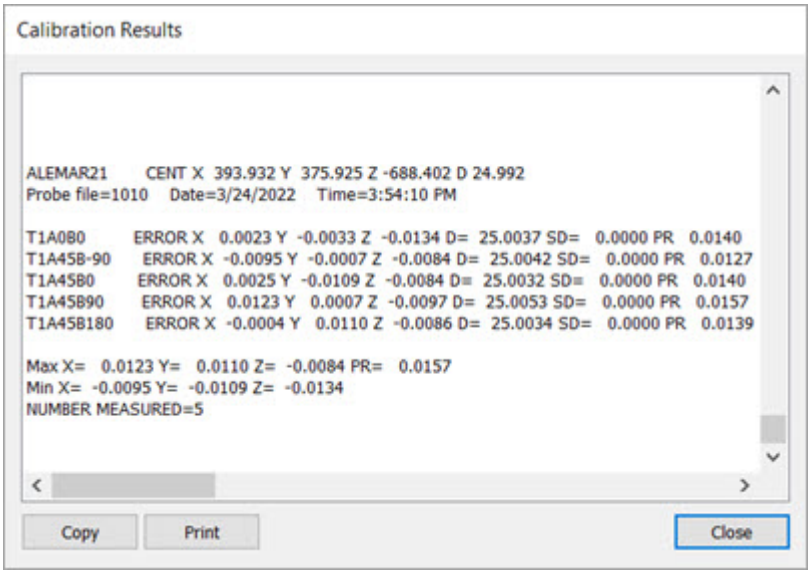


For convenience, during the auto-centering procedure, the software shows the laser sensor alignment stripe in yellow.

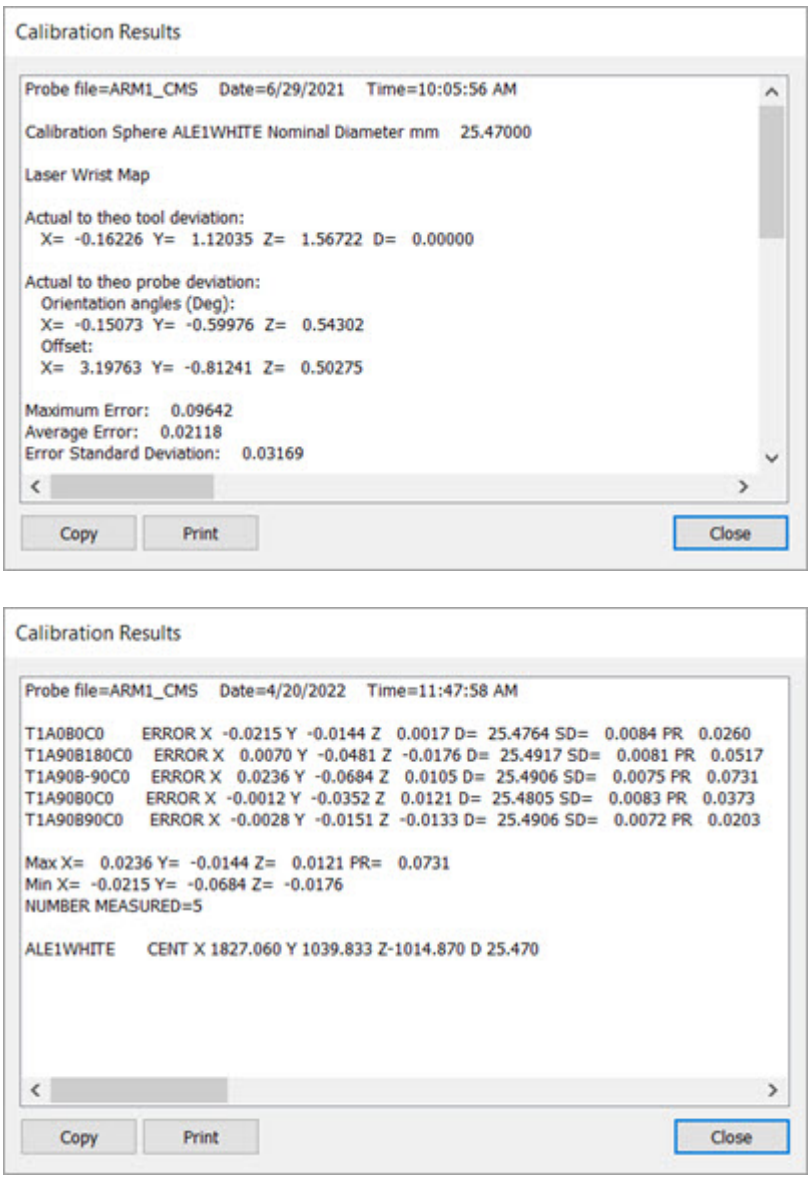
Step 5: Check the Calibration Results

From the **Probe Utilities** dialog box, click the **Results** button to show the **Calibration Results** dialog box.





Calibration Results dialog boxes for indexable head after tip calibration (top) and after qualification check (bottom)



Calibration Results dialog boxes for continuous wrist after wrist mapping (top) and after qualification check (bottom)

PC-DMIS records several results from the calibration in this dialog box. Look at the maximum, average, and standard deviation values.

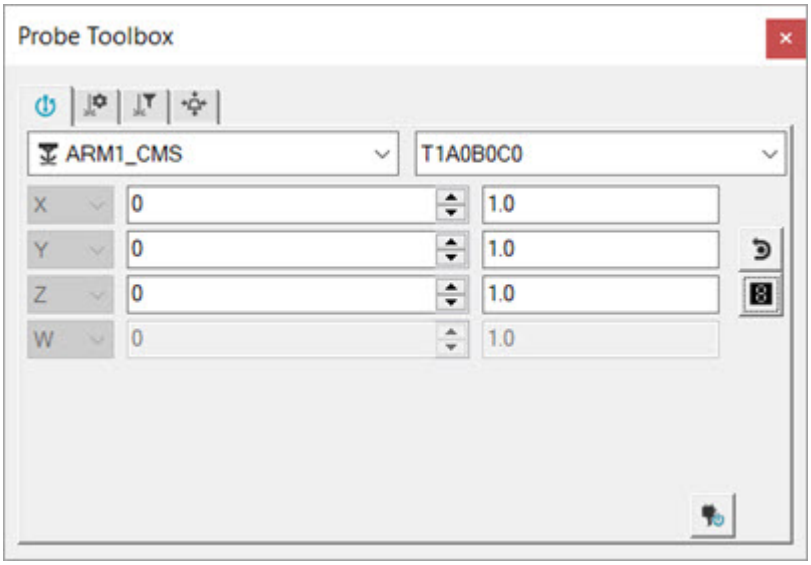
The maximum should be somewhere between 20 and 100 microns. The average and the standard deviation should be around 20 microns.

If the values look correct, click the **OK** button to close the **Calibration Results** dialog box. You have these options:

- To paste the report into a different application (such as Microsoft Word, Notepad, or other application), click **Copy**, open the desired application, and press Ctrl + V to paste it.
- To send the report to a printer, click **Print**.

This finishes the setup and calibration process for your laser sensor. You can now use all the laser-related functionality.

Using the Probe Toolbox in PC-DMIS Laser



Probe Toolbox - Position Probe tab selected

The available options on each **Probe Toolbox** tab are dependent on the sensor configured for your system. Because of this, the individual **Probe Toolbox** images and the associated descriptions in this section of the PC-DMIS Laser documentation may not match what you see in PC-DMIS.

The **View | Other Windows | Probe Toolbox** menu option displays the Probe Toolbox. The **Probe Toolbox** contains various laser sensor parameters that you can use to acquire the data points that a measurement routine needs.

Your PC-DMIS license must contain the laser option, and you must be working with a supported laser sensor to access the laser-related tabs in the **Probe Toolbox**.

The **Probe Toolbox** contains the laser parameters on these tabs:

For Portable Configurations

- Laser Scan Properties **+!**
- Feature Extraction **!**

For CMM Configurations

- Position Probe
- Laser Scan Properties
- Laser Filtering Properties

Laser Clipping Region Properties

Feature Extraction

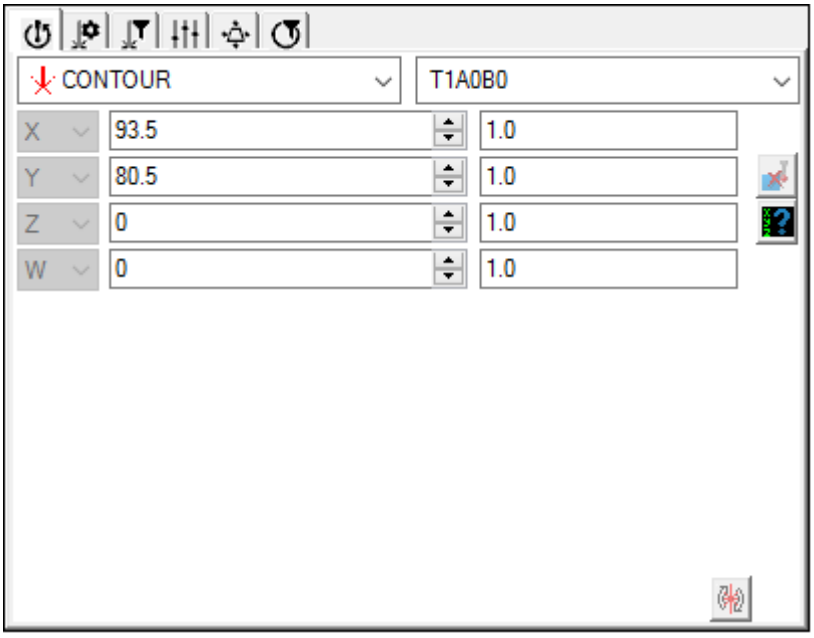
Laser AF Multiple Creation

The list above shows all possible **Probe Toolbox** tabs. The tabs that are available depend on the sensor that you have on your system. If the capabilities for a tab do not apply to your specific sensor, then that tab is unavailable.

+ For HP-L probes, these tabs are visible when you close the **Auto Feature** dialog box.

! For HP-L probes, these tabs are visible when you open the **Auto Feature** dialog box.

Laser Probe Toolbox: Position Probe tab



Probe Toolbox - Position Probe tab

The **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) allows you to select the current probe file and tip and define the current probe location in the active alignment coordinates. You can double click the X, Y, and Z values to edit them.

Warning: When you edit the current probe location, the machine moves to the new coordinate without notice. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

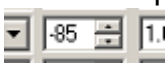
If you don't see any information in the **Probes** and **Probe Tips** lists of the Probe Toolbox, you need to first define a probe. For information on how to define a probe, see the "Defining Hardware" chapter in the PC-DMIS Core documentation.



While you can use this tab with all probe types (contact, laser, and optical), this document only covers PC-DMIS Laser related items. For information about the toolbox as it relates to probes in general, see "Using the Probe Toolbox" in the "Using Other Windows, Editors, and Tools" chapter in the PC-DMIS Core documentation.

To Position Your Laser Sensor

You can use the **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) to position your laser sensor. This tab contains sets of values in two columns.

Left Column: The X, Y, Z values. They show the laser sensor's current position. You can click up and down arrows to change the value in the **XYZ Probe Position** box  for an axis. This moves your laser sensor in real-time by the increment value on the right.

Right Column: The increment values. These specify how much to increase or decrease the XYZ Probe Position box for each axis when you click the up and down arrows on the left column.

Alternately, you can type the XYZ values in the left column, and press Enter to move your laser sensor to a predefined position.

Controls for Position Probe Tab

The toggle buttons on the **Position Probe** tab of the Probe Toolbox (**View | Other Windows | Probe Toolbox**) are:

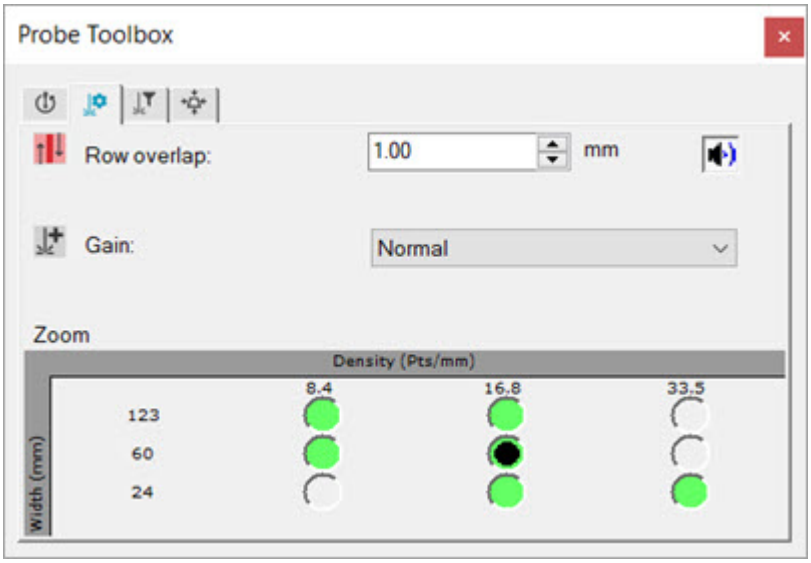
 **Probe Readouts** - This toggle button shows or hides the Probe Readouts window. You can easily resize or relocate this window. Most of the information on the Probe Readouts window is the same for all probe types. For details, see "Using the Probe Readouts Window" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

 **Laser On/Off** - This toggle button turns the laser on or off. It is only available for laser probes.

 **Initialize Probe** - This button starts or initializes the laser. You can't do anything with the laser until it is initialized. This takes about 15 seconds.

The **Laser On/Off** button and the **Initialize Probe** button are no longer present on the **Position Probe** tab for HP-L-10.10, HP-L-10.10 LITE, and DSL-100 (Maestro) sensors. You can use the toolbar in the bottom left of the Live View window to toggle the laser for those sensors. For more information, see "Using Live View with the HP-L-10.10 Sensor".

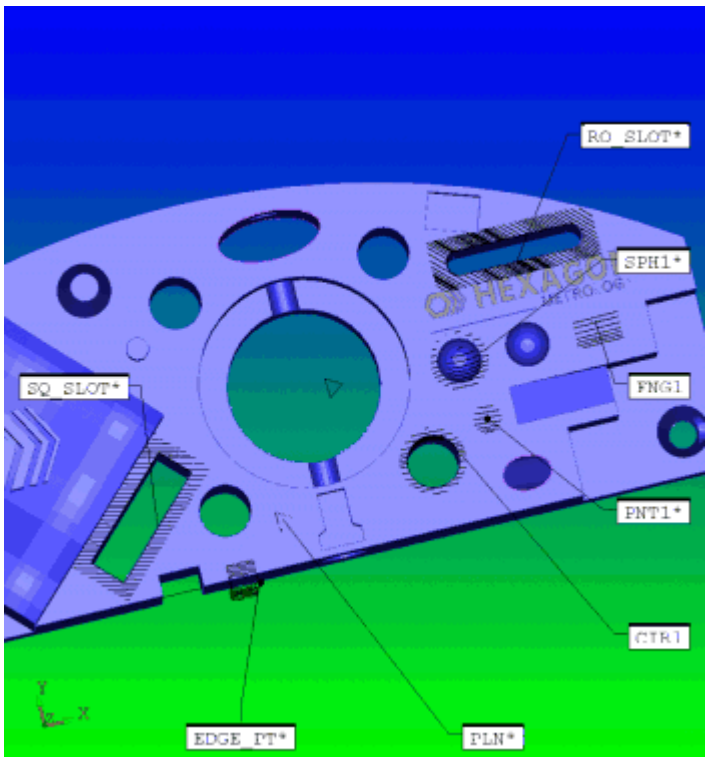
Laser Probe Toolbox: Laser Scan Properties tab



Probe Toolbox - Laser Scan Properties tab

The **Laser Scan Properties** tab defines how PC-DMIS acquires data from the scan, and whether or not scan lines and feature visualizations appear in the Graphic Display window.

Show/Hide Stripes - This button toggles the display of the laser stripes on the part model. Click this button to allow the laser scan stripes to appear in real time in the Graphic Display window. PC-DMIS limits how the stripes appear in the Graphic Display window to the distance of the feature nominal plus the **Overscan** value. The software uses the **Overscan** value to control the amount of clipping and the visibility of the stripe. The graphic below gives an example of how these stripes appear.

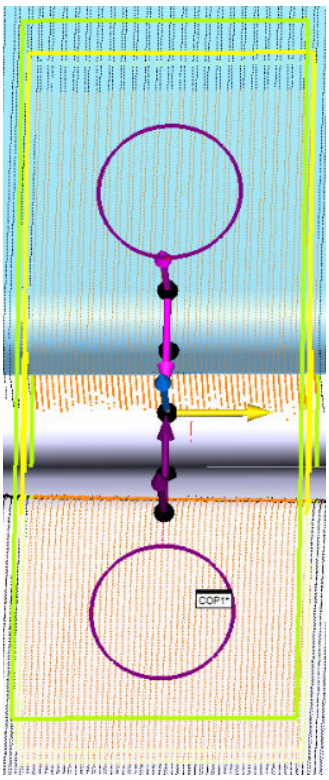


Scan features showing stripes

Sound ON/OFF - This button turns the sound on or off. See "Using Sound Events".

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 those points that the software passes to the feature extractor engine based on the current settings.



Showing segregated points inside a sample Flush and Gap feature

Initialize Probe - This button starts or initializes the laser which takes about 15 seconds and is only available for Portable configurations. You can't do anything with the laser until you initialize it.

AutoZoom ON/OFF - This button turns the laser AutoZoom functionality on or off. Whenever you start the scan, AutoZoom dynamically pans, zooms, rotates, and sizes the view containing the laser data in the Graphic Display window to show the incoming data.

You can override the limitations with the up and down arrows, or enter a value into any of the boxes. However, your machine rejects invalid values and forces them to a valid number.

Sensor Frequency

This parameter controls the internal sensor frequency of the probe. The value that appears is the sensor pulses per second. For sensors with variable frequency capabilities, the higher the frequency, the more data you get. It is important to understand that more data is not always better. With variable frequency scanners, you should use a frequency in the middle of the supported range. This is a good balance between speed and accuracy.

Row Overlap

If the feature or patch scan is larger than the width of the scan line, multiple passes of the probe are taken. In that case, this parameter controls how far each pass overlaps the previous pass. The default value is 1.0 mm.

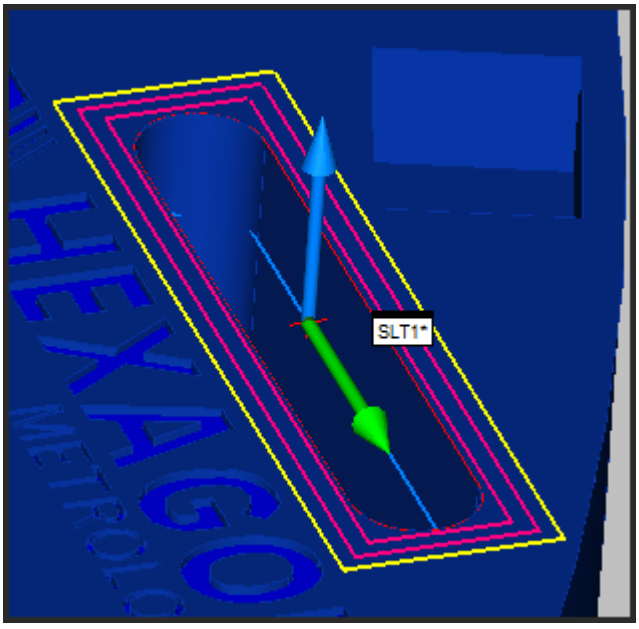
Overscan

The **Overscan** parameter extends the scan beyond the boundaries of the feature you measure.

For DCC systems, this parameter controls how far beyond the nominal feature's dimensions the probe scans along both the major and minor axis of the feature. The default value is 2.0 mm. If you measure features whose actual location may vary significantly from their theoretical values, you need to increase this value to ensure that PC-DMIS measures the entire feature.

The **Overscan** value does not perform the clipping of the data. The **Feature Based Clipping** area in the **Feature Extraction** tab handles clipping. For information, see the "Feature Based Clipping Parameters" topic.

- For a DCC laser Cylinder or Cone feature, the **Overscan** value should be a negative value.
- For a laser Stud feature (see the laser Cylinder for stud information), the **Overscan** value should be a positive number.



Sample Slot Auto feature that shows the overscan in yellow

Exposure

This parameter controls the exposure of the sensor. The default value of 150 works well for most parts, but for parts that absorb a lot of light (such as a black anodized surface), you may need to increase the value.

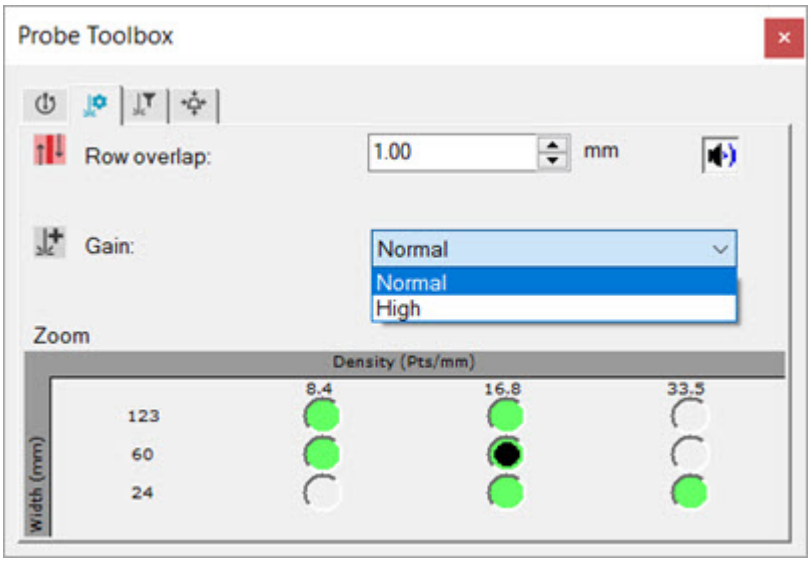
Pointcloud

This parameter defines the COP command from which PC-DMIS extracts the Auto feature. If you select "disabled", then PC-DMIS internally stores the data from the scan. You can use the **Operation | Laser Autofeatures** from the menu to delete the internal data if needed. See "Clearing Auto Feature Scan Data".



The "disabled" option is only used with DCC laser scans.

Gain (for HP-L Sensors)



Probe Toolbox - Laser Scan Properties tab showing the Gain options list

HP-L sensors provide you with an additional list called **Gain** in the **Laser Scan Properties** tab of the Probe Toolbox.

- HP-L-10.6 supports **NORMAL** and **HIGH**.
- HP-L-20.8 supports **NORMAL**, **HIGH**, and **XHIGH**.
- HP-L-5.8 supports **1**, **2**, **3**, **4**, and **5**.

This list lets you choose among these sensitivity modes:

Sensitivity Modes

NORMAL sensitivity - You should use this default sensor mode on most normal parts. This mode sets the `QUALITY FILTER` toggle field in the Edit window in Command Mode to `ON` , so that the Edit window shows the associated fields. This sensitivity mode also hides the **Quality Filter** icon.

HIGH sensitivity - The **HIGH** sensitivity mode becomes available for selection if you run PC-DMIS in Online mode. You should only use the **HIGH** sensitivity mode if you are scanning a part comprised of a troublesome material where the **NORMAL** sensitivity mode returns poor data. For example, a part that absorbs too much light because it has glossy, dark, or black surfaces may require this type of mode. However, note that scanning a normal part in **HIGH** sensitivity mode may yield noisy data.

XHIGH (extra high) sensitivity - **XHIGH** is similar to **HIGH**. It provides an option for scanning materials that may be even more troublesome than those that can be handled using the **HIGH** option. If you don't get good results using **HIGH**, you can try using the **XHIGH** option. However, as with the **HIGH** option, if you scan a normal part in **XHIGH** mode, it may return even more noisy data.

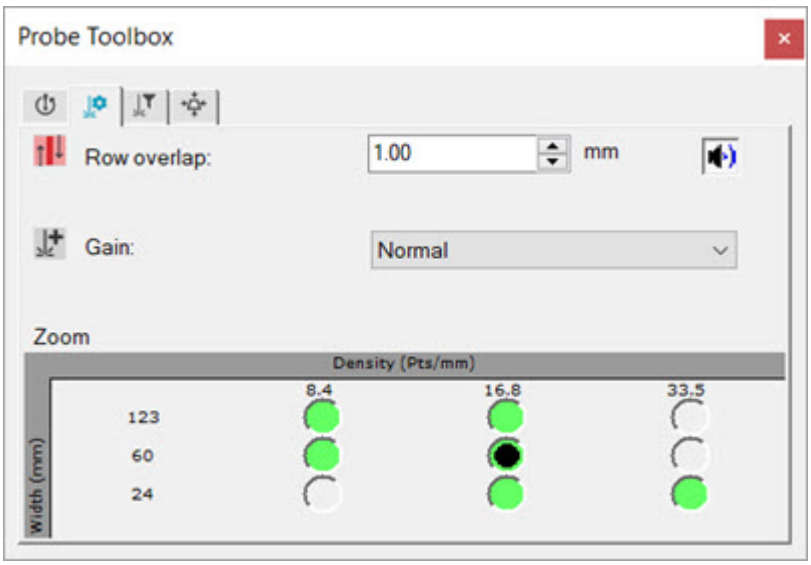
In the **HIGH** and **XHIGH** modes, a **Quality Filter** icon appears next to the **Gain** list:

Quality Filter  - If you enable this mode, PC-DMIS filters low quality points including double reflections, poor quality data on edges, and outliers. If enabled, it sets the `QUALITY FILTER` toggle field to ON in the Edit window in Command Mode to show the associated fields in the Edit window.

1, **2**, **3**, **4**, and **5** sensitivities - These sensitivities are available for the HP-L-5.8 sensor.

Scan Zoom States (for HP-L-10.6 and HP-L-20.8 Sensors)

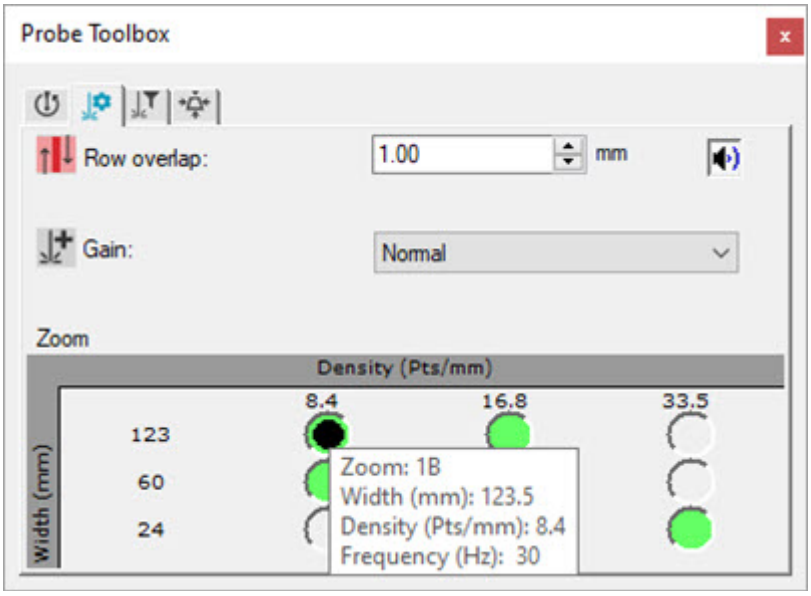
HP-L sensors provide you with an additional area called **Zoom**, located at the bottom of the **Laser Scan Properties** tab of the Probe Toolbox. This area tells the sensor to work in predefined zoom states, with each state comprised of a specific combination of sensor frequency, data density, and Field of View (FOV) width.



Probe Toolbox - Laser Scan Properties tab with the Sample Zoom area

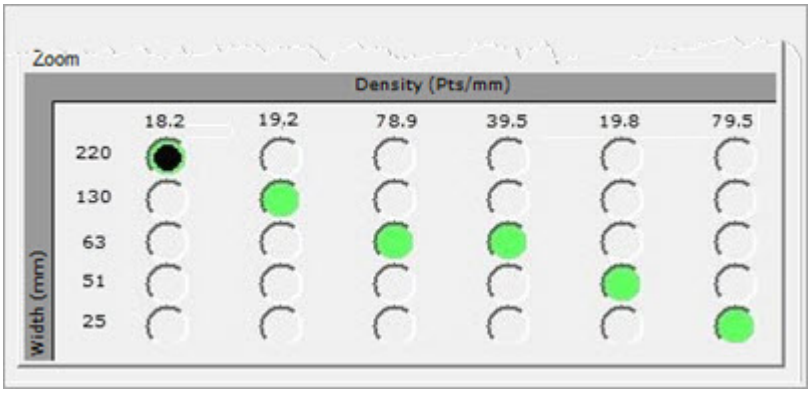
The **Zoom** area displays a grid of option buttons arranged in columns and rows. Across the top, the "columns" show the data density. Along the side, the "rows" list the FOV width. You can only select proper combinations which are the option buttons with a green background. The software grays out the improper combinations.

You can hover your mouse pointer over any valid option button to display a yellow tooltip that details the option's scan mode information.



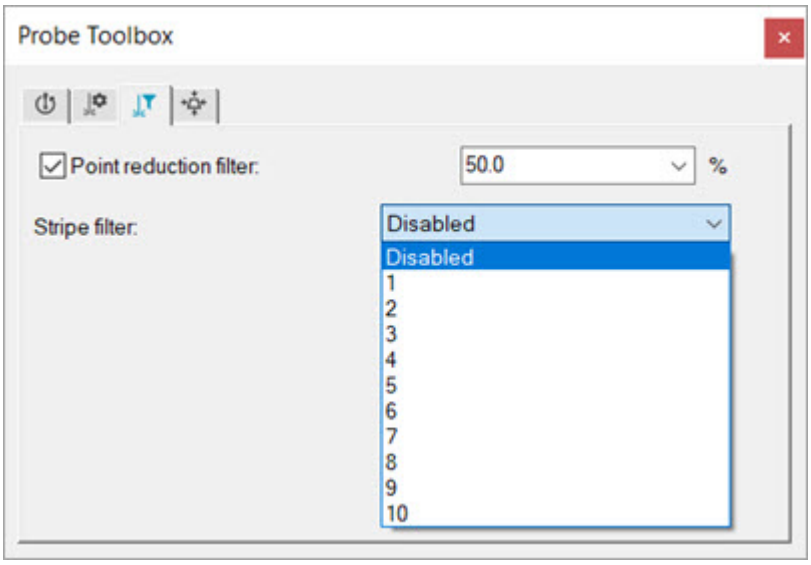
Probe Toolbox - Laser Scan Properties tab showing tooltip in the Sample Zoom area

Available Scan Zoom States for HP-L-20.8



Example of possible scan states for the HP-L-20.8 sensor

Laser Probe Toolbox: Laser Filtering Properties tab



Probe Toolbox - Laser Filtering Properties tab

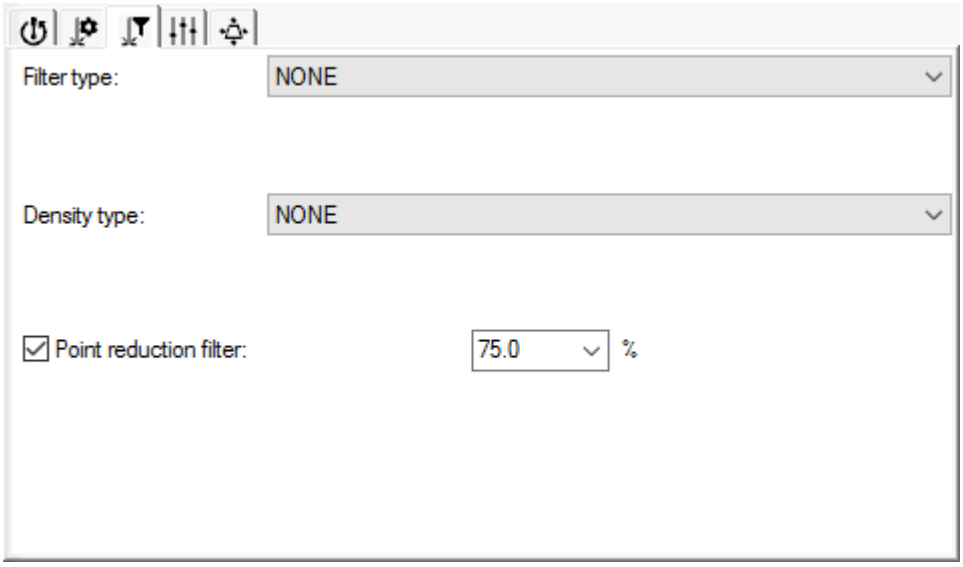
The **Filtering** tab is useful when you want to filter the data as PC-DMIS collects it.

The following filtering options are available from the list.

Filter Type: Only Available for HP-L Sensors

- Point Reduction
- Stripe

Filter Type: None

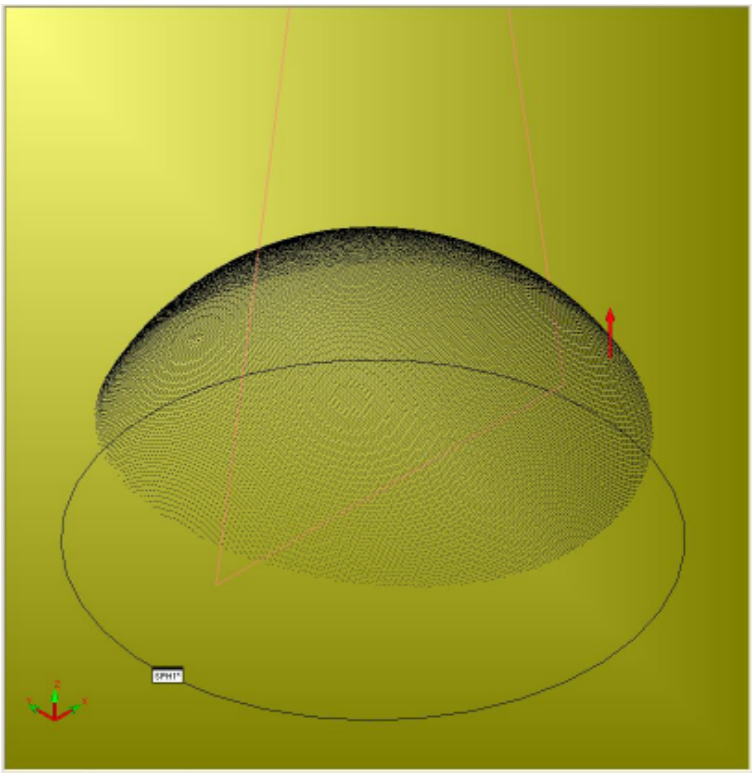


None Filter type

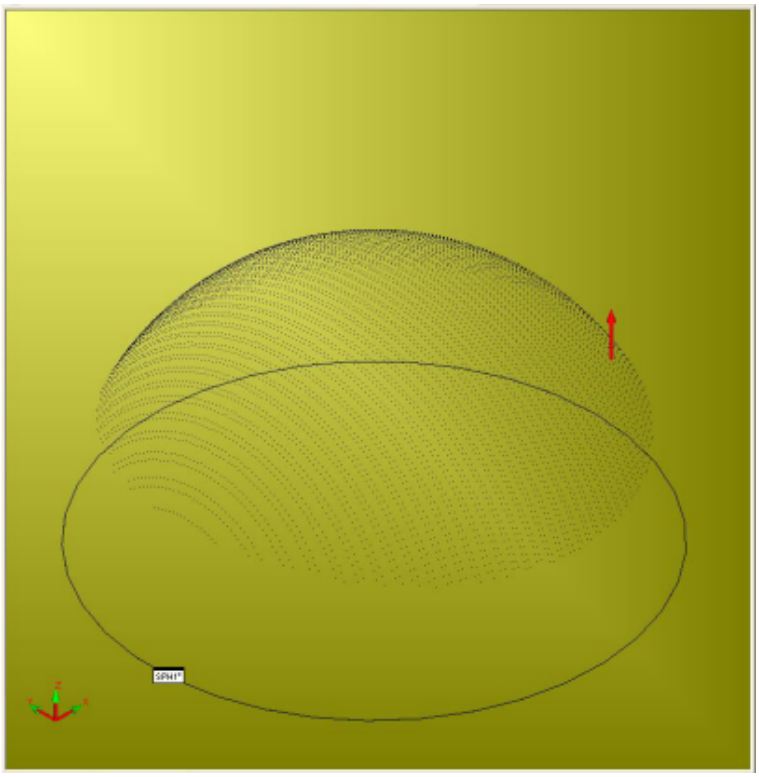
No initial filter is made. However, you do have the option to filter by point reduction.

Point Reduction Filter - This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, PC-DMIS acquires the complete data set without any filtering.

Example of Point Filtering Disabled



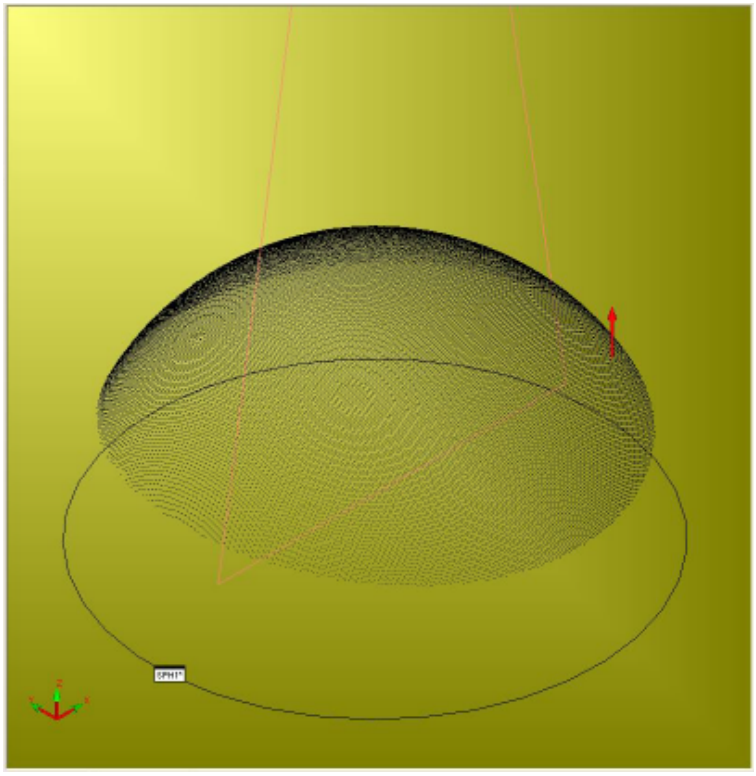
Example of Point Filtering of 50%



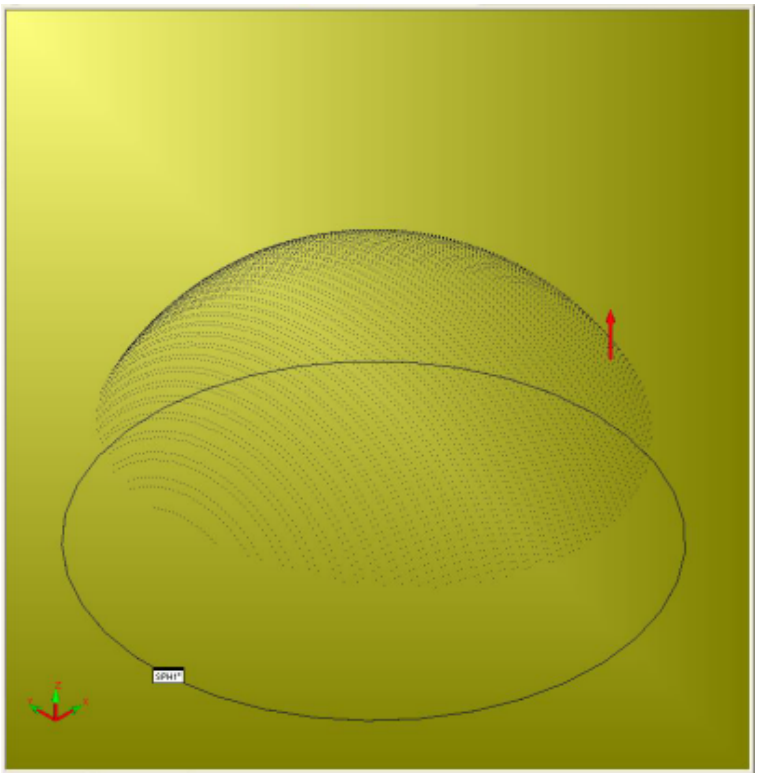
Filter Type: Point Reduction

Point Reduction Filter - This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, PC-DMIS acquires the complete data set without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%



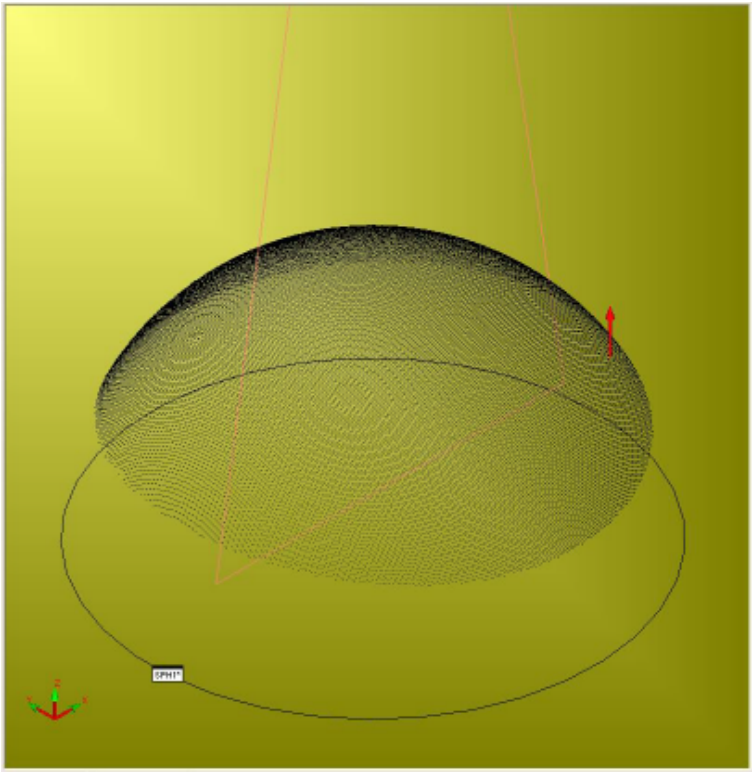
Filter Type: Stripe



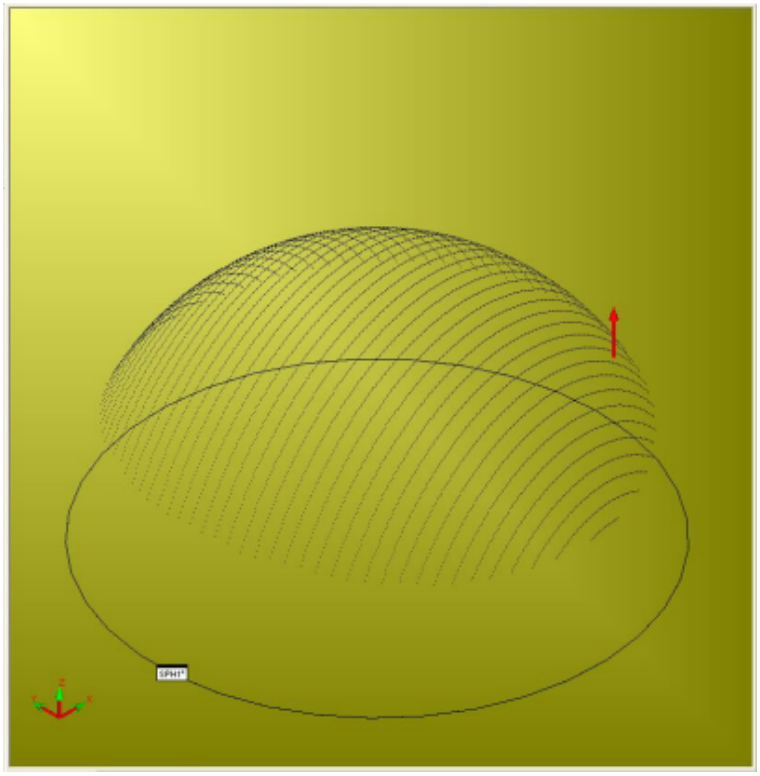
This type is only available for HP-L sensors.

The **Stripe Filter** list lets you filter scan lines along the scanning direction. You can select a number from a scale of 1 to 10 (1 represents minimum filtering while 10 represents maximum filtering). If disabled, PC-DMIS acquires the complete data set without any filtering.

Example of Stripe Filtering Disabled

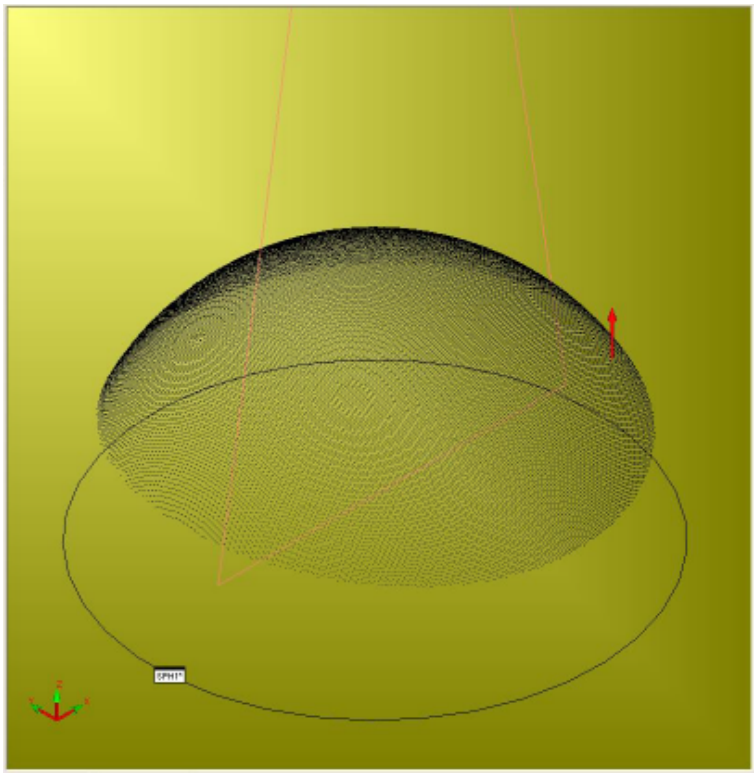


Example of Stripe Filtering of 5

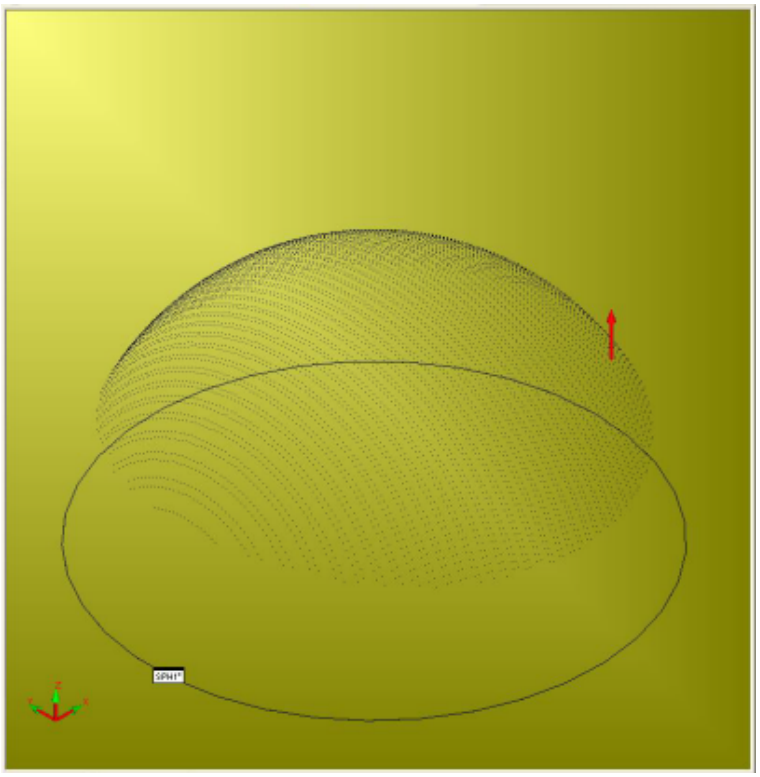


Point Reduction Filter - This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, PC-DMIS acquires the complete data set without any filtering.

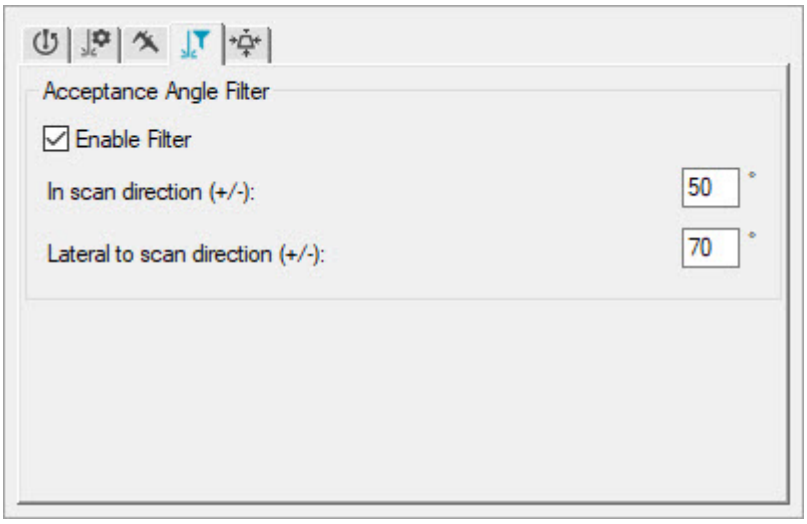
Example of Point Filtering Disabled



Example of Point Filtering of 50%



Laser Probe Toolbox: Acceptance Angle Filter tab



Probe Toolbox - Acceptance Angle Filter tab

The **Acceptance Angle Filter** tab allows you to set the **In scan direction** and the **Lateral to scan direction** properties.

To do this, select the **Enable Filter** check box and type a value in these boxes:

> **In scan direction (+/-)** - This value defines the acceptable angle of incidence in the direction of the scan. The default value is 50 degrees. The maximum value is 90 degrees.

Lateral to scan direction (+/-) - This value defines the acceptable angle of incidence lateral to the direction of the scan. The default value is 70 degrees. The maximum value is 90 degrees.

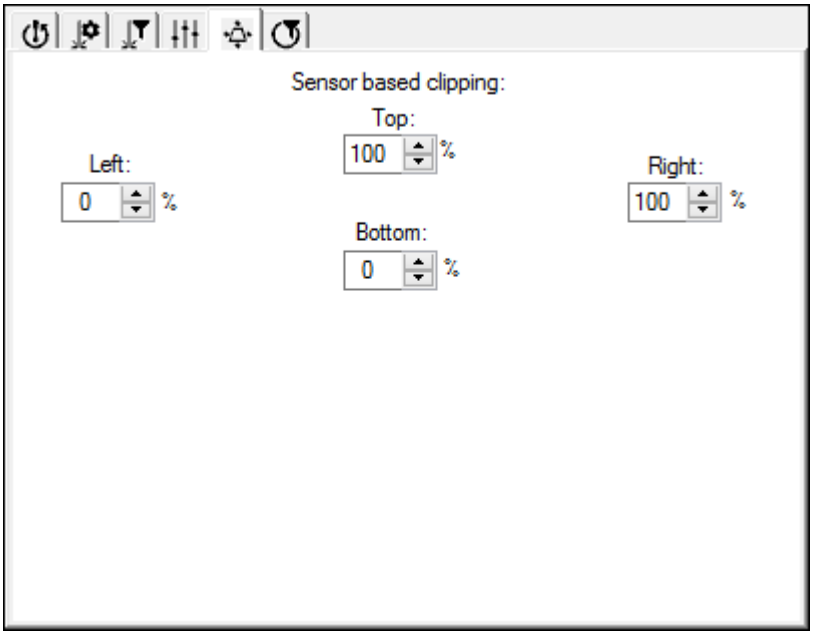
If you do not select the **Enable Filter** check box, the filter is disabled.

The purpose of these filters is to limit the angle of incidence and to prevent PC-DMIS from collecting points that have a large angle of incidence. You can expect a higher dispersion when points are collected with a large angle of incidence. The filters remove outliers which may occur if the angle of incidence is getting too big.

Recommendations:

- Use the default filter values to collect the highest quality pointcloud data.
- If coverage is not good enough, because of complicated geometries or the difficulty of reaching an area or specific feature, first increase the angle values.
- If coverage is still not sufficient, disable the **Acceptance Angle Filter** option.

Laser Probe Toolbox: Laser Clipping Region Properties tab



Laser Clipping Region Properties tab

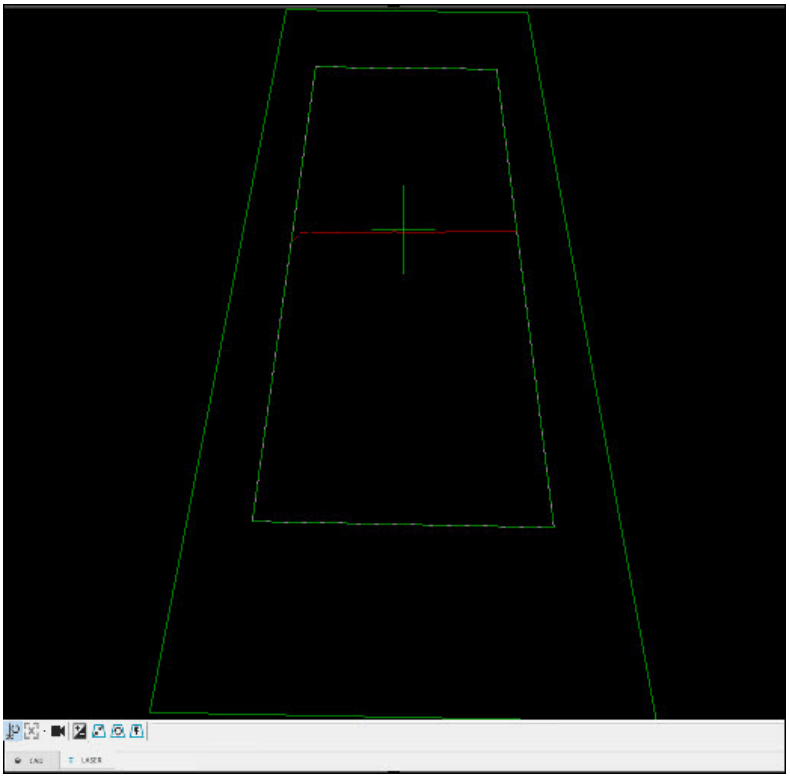
The **Laser Clipping Region Properties** tab allows you to set parameters to discard data outside a specified region, within the sensor's field of view. This feature lets you keep only pertinent data.

Keystone - The large green trapezoid in the Laser View (see below) represents the sensor's maximum field of view. The clipping region is within this field of view.

Sensor Based Clipping region - The smaller green trapezoid within the sensor's field of view.

The **Top**, **Left**, **Right**, and **Bottom** boxes can be set with values from 0 to 100 percent that allow control over the clipping region. This lets you discard unnecessary data.

When the **Bottom** and **Left** values are at 0% and the **Top** and **Right** values are at 100%, the sensor keeps all of the data collected because the clipping region is the same as the maximum field of view.



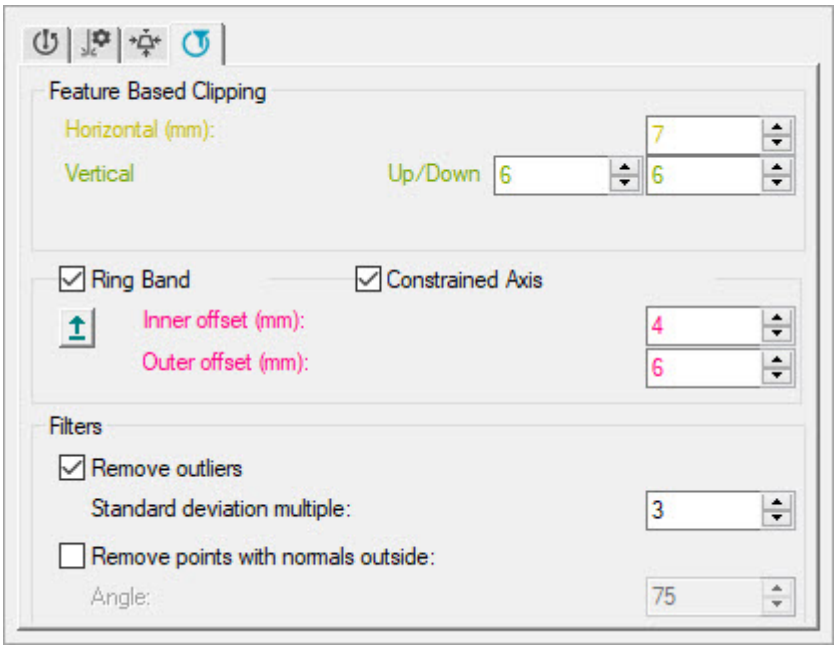
Example of clipping data using Top 85, Bottom 85, Left 15, and Right 15

For example, you can use the clipping area to measure a hole. Since you would not want data from an adjacent hole to interfere with feature computation, you can control the clipped area to discard unwanted data.

Dragging Lines

Another way to set the desired percentage is to use your mouse pointer to drag the clipping region lines of the smaller green trapezoid.

Laser Probe Toolbox: Feature Extraction tab



Feature Extraction tab

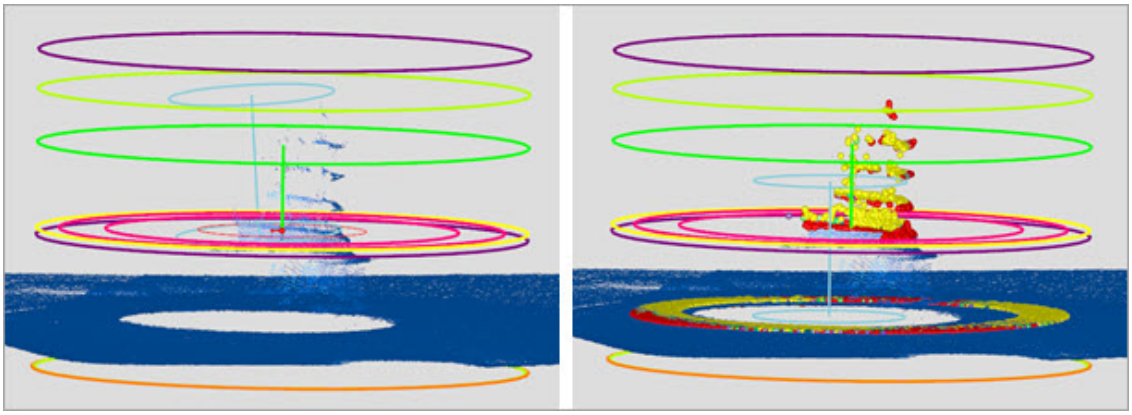
You can use the **Feature Extraction** tab to specify ring band and feature-based clipping parameters as well as to remove outliers on supported features.

> The **Feature Extraction** tab is only available for laser sensors. Note that the above image shows all possible options and are shown only for descriptive purposes. The feature type you select is only one criteria PC-DMIS uses to determine the options to display on the **Feature Extraction** tab.

Depending on the feature type, the following feature extraction parameters are available:

- Feature Based Clipping parameters - This is available for all laser Auto features.
- Ring Band parameters - This is only available for Circle, Cone, Cylinder, Polygon, Round Slot, and Square Slot Auto features.
    - The Probe Toolbox **Feature Extraction** tab has an additional feature called **Constrained Axis** that you can use to constrain the Cylinder feature to the vector of the plane surrounding the cylinder. The plane is defined by the Ring Band area. The **Constrained Axis** option is only available for the Cylinder feature and is only available when you select the **Ring Band** option.

      You can see the before and after effects of the **Constrained Axis** option in the images below. Even with a sparse set of data as shown below, this option calculates a more stable cylinder feature constrained to the plane's vector.



Examples with the Constrained Axis not selected (left) and selected (right)

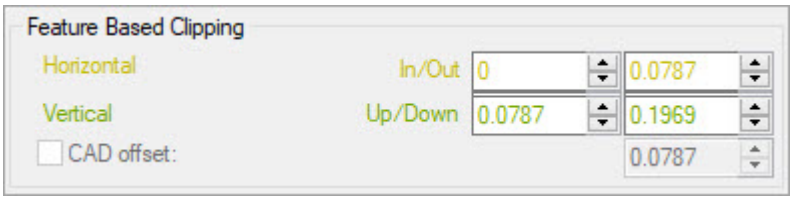
- Filters:
  - Remove outliers parameter - This is only available for High Point, Surface Point, Plane, Circle, Cone, Cylinder, Polygon, Round Slot, Square Slot, and Sphere Auto features.
  - Remove points with normals outside parameter - This is available for all laser Auto features.

    If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area of the Probe Toolbox **Feature Extraction** tab, select the **Remove points with normals outside** check box and type the value in the Feature Extraction **Angle** box.

    The Feature Extraction **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.

Feature Based Clipping Parameters



Feature Based Clipping area for non-plane Auto features

Note that not all options shown are available for all feature types.

PC-DMIS can clip laser data in both the horizontal and vertical directions when you type a distance in the **Horizontal** box and, when available, in the **Vertical** box. This distance clips all of the laser data outside of the defined distance and excludes that data when it extracts the feature.

Alternatively, for 3D features, you can clip data within an offset boundary around all of the CAD elements on a surface. This is also termed CAD segregation. For information on this offset boundary, see "CAD Offset" below.

For the Cone Auto feature:

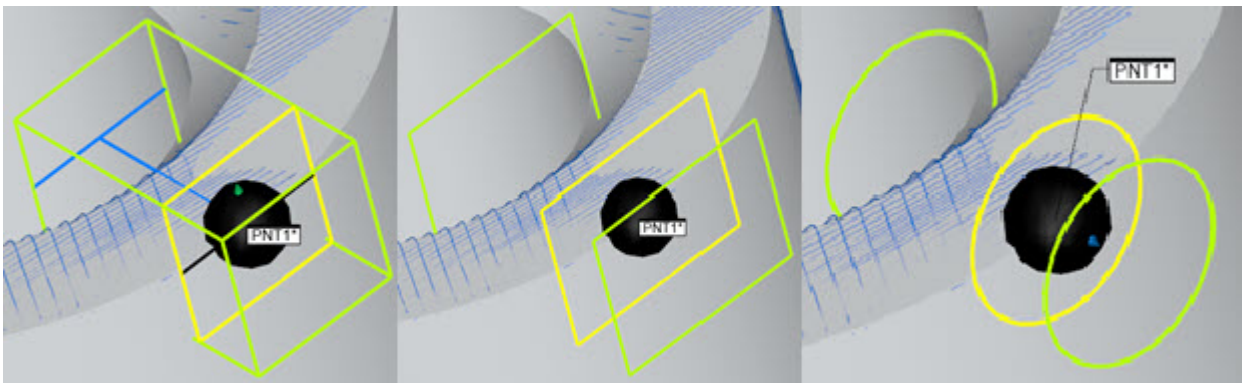
- The value for the **Horizontal** option defines how much larger the circular boundary (within which the feature points lie) is than the theoretical diameter.
- The value for the **Vertical** option defines how much longer the cylindrical boundary (within which the feature points lie) is than the theoretical length.

Horizontal and Vertical Up/Down Clipping - Point Features



All laser Auto features support standard horizontal clipping. All laser Auto features except for the laser Auto Sphere feature support Vertical Up/Down clipping.

Examples of what each of the laser auto feature point types look like with the same Horizontal and Vertical Up/Down settings:



Laser Auto Point with the same Feature Based Clipping settings: Edge Point (left), High Point (middle), Surface Point (right)

Horizontal Clipping, Vertical Up/Down Clipping, and CAD offset

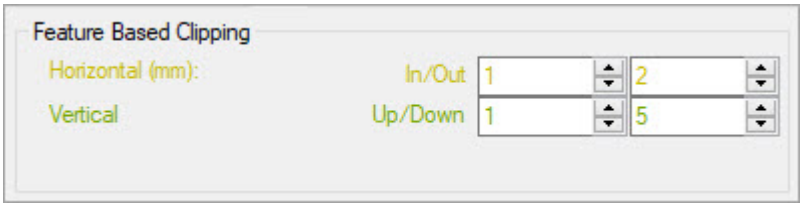


All of the Auto features support standard Horizontal clipping. These features support Vertical Up/Down clipping and CAD offset:

- Cone
- Cylinder
- Plane

PC-DMIS shows the clipping distances defined in the feature based clipping rings as colored rings. The horizontal clipping appears as a yellow ring, and the vertical clipping appears as a light-green ring.

Horizontal In/Out Clipping



For inner features only, you can apply the Horizontal In clipping, which appears as an innermost yellow ring, and the Horizontal Out clipping which appears as an outermost yellow ring.

These laser Auto features support Horizontal In/Out clipping:

- Circle
- Polygon
- Round Slot

- Square Slot

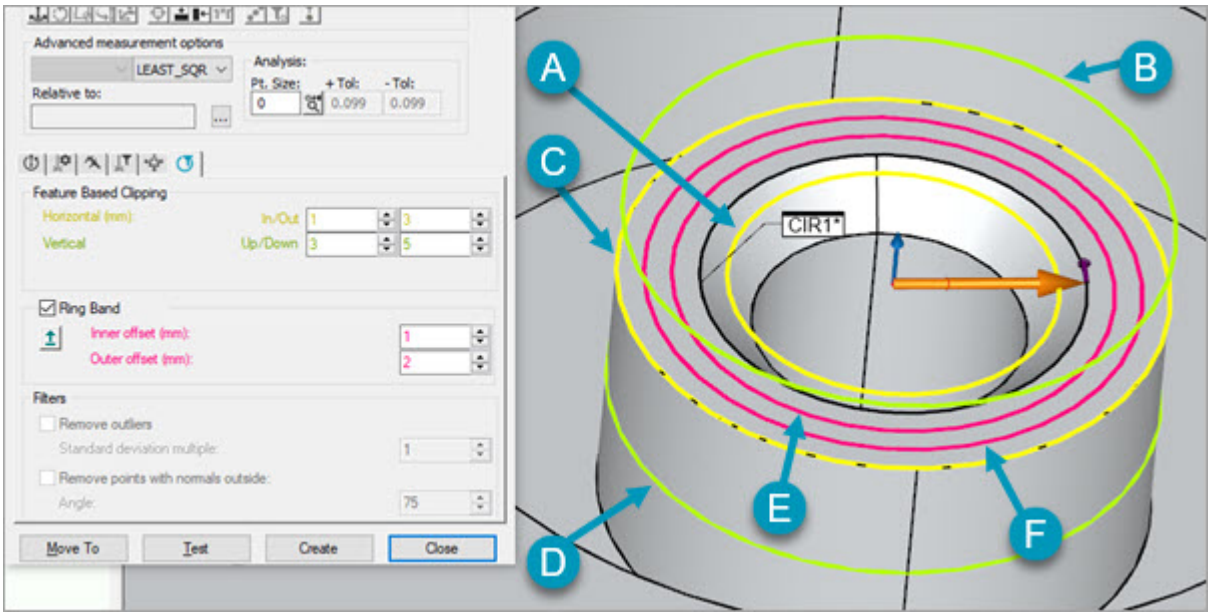
For Horizontal Out clipping, you must type a value greater than the value of the Ringband outer offset value. See the inner 2D laser Auto Circle feature example below.

The nominal size of each feature is defined as:

- Diameter for Circles
- Inner Diameter for Polygons
- The shortest value between the Width and Length for Round Slots or Square Slots

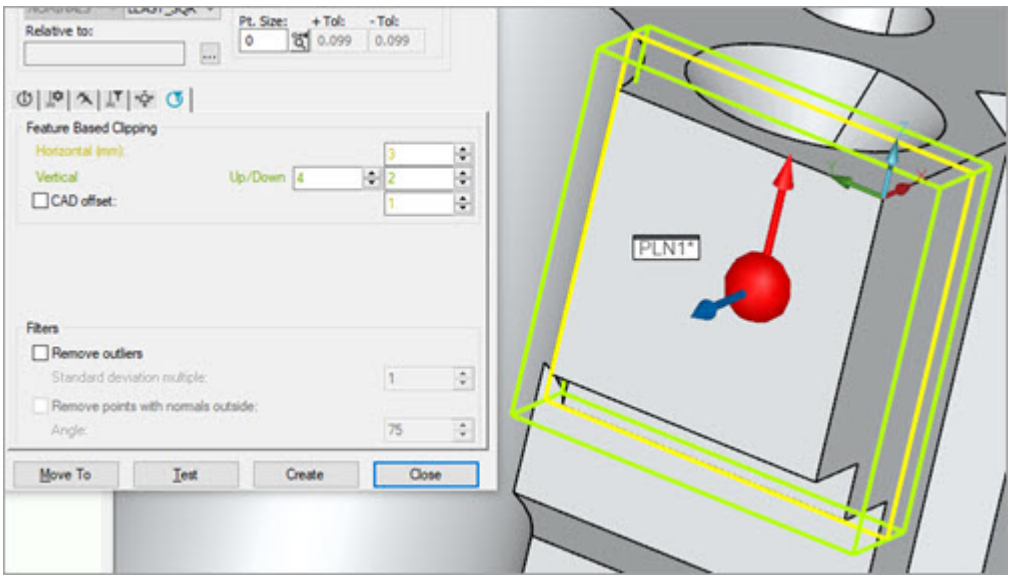
The **Horizontal Inner Clipping** parameter is available in the **Feature Based Clipping** area of the **Feature Extraction** tab. You can also use the `Inner_Horizontal_Clipping` entry in the Settings Editor to set this value. For details, see the "Inner_Horizontal_Clipping" topic in the PC-DMIS Settings Editor documentation.

Clipping Examples



Example of an Inner Auto Laser Circle feature:

A - Horizontal Inner Clipping Ring
B - Up Vertical Clipping Ring
C - Horizontal Outer Clipping Ring
D - Down Vertical Clipping Ring
E - Inner Offset Ring Band
F - Outer Offset Ring Band



Example of a Laser Auto Plane feature with Horizontal and Vertical Up/Down clipping enabled

Typical Edit window command for a laser Circle Auto feature:

```
CIR1       =FEAT/LASER/CIRCLE/DEFAULT,CARTESIAN,IN,LEAST_SQR
             THEO/>40,30,0<,>0,0,1<,30
             ACTL/>40,30,0<,>0,0,1<,30
             TARG/>40,30,0<,>0,0,1<
             ANGLE
 VEC=>1;,0,0<              DEPTH=1,START
 ANG=0,END ANG=360              DIRECTION=CCW
             SHOW
 FEATURE PARAMETERS=NO              SHOW_LASER_PARAMETERS=YES

                 REFERENCE
 ID=COP1                 INNER
 HORIZONTAL CLIPPING=2,OUTER HORIZONTAL CLIPPING=3
               UP
 VERTICAL CLIPPING=1,DOWN VERTICAL CLIPPING=4
               RINGBAND=ON,INNER
 OFFSET=1,OUTER OFFSET=2                OUTLIER_REMOVAL=OFF
               REMOVE
 POINTS WITH NORMALS OUTSIDE=ON,40
```

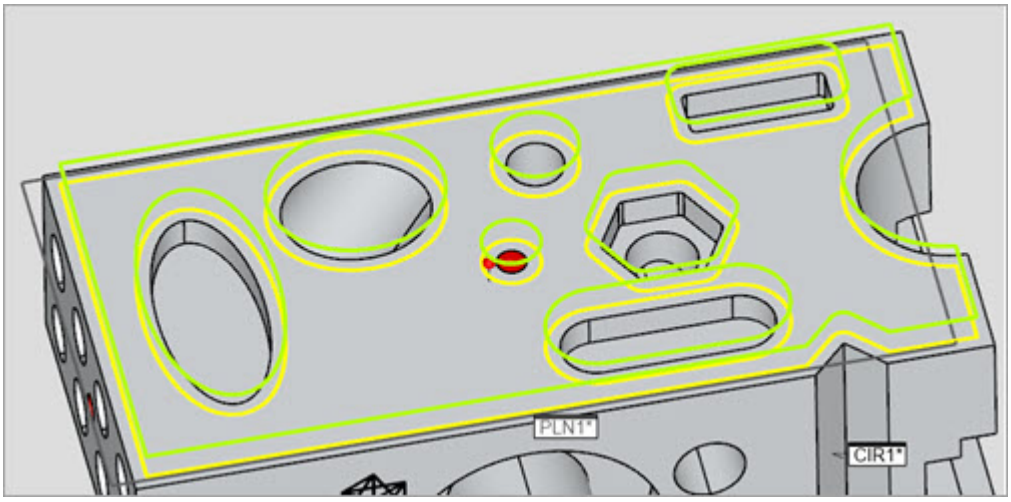
CAD Offset



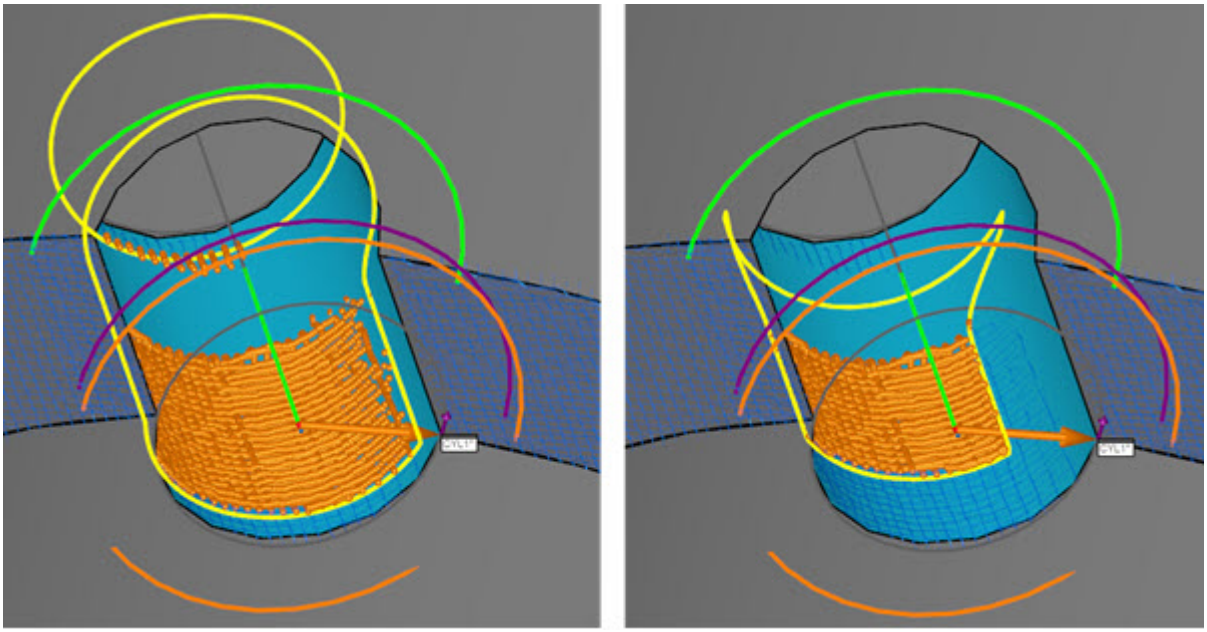
Feature Based Clipping area for Laser Auto Plane feature

You can enable the **CAD offset** option for all 3D Auto features and all 3D Constructed Extracted features (Plane, Cone, Cylinder, and Sphere). The **CAD offset** option provides a way for PC-DMIS to "shrink away" from the selected CAD face boundaries and eliminate points that are within the offset distance to the edges of the feature.

When you select this check box, PC-DMIS creates a yellow offset boundary around each feature on the CAD model's surface.



Example of a Laser Auto Plane feature with CAD-based clipping enabled



Example of a Cylinder Auto feature with CAD offset of 1 (left) and CAD offset of 3 (right)

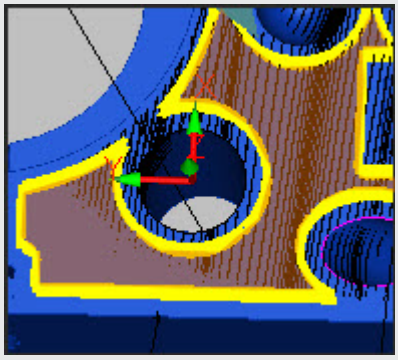
To view the segregated points, you must have the **Show/Hide segregated points** button enabled on the **Laser Scan Properties** tab of the Probe Toolbox. For details, see the "Laser Probe Toolbox: Laser Scan Properties tab" topic in the PC-DMIS Laser documentation.

When you extract a Laser Cylinder feature, the **CAD offset** option is available as long as you do not use the **Ring Band** option.

PC-DMIS draws the boundary around the selected surface to show the area of the considered points. You must select a CAD face when you use the **CAD offset** option.

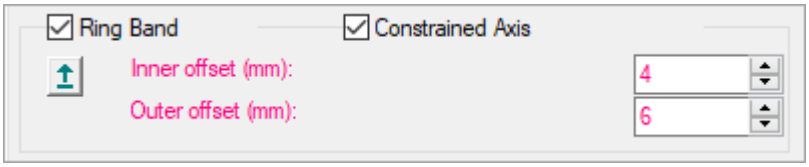
PC-DMIS clips the laser data that falls inside of an offset boundary for all of the features on the selected CAD model surface. PC-DMIS uses the data outside of the offset boundary to solve the plane.

For example, consider the image below. The image shows a section of a sample part. The section indicates the data that PC-DMIS would use to create the Plane Auto feature:



Note that the translucent orange overlay on the image is for clarification only; it is not visible in PC-DMIS.

Ring Band Parameters



Feature Extraction - Ring Band

You can use the **Ring Band** area to calculate the feature's projection plane and normal vector. PC-DMIS projects the feature data up into the ring band's plane. You can use the following **Ring Band** controls to accomplish feature extraction for circles, cones, cylinders, polygons, round slots, and square slots:

Ring Band

When you select this option, the software enables these **Ring Band** options:

Plane Selection



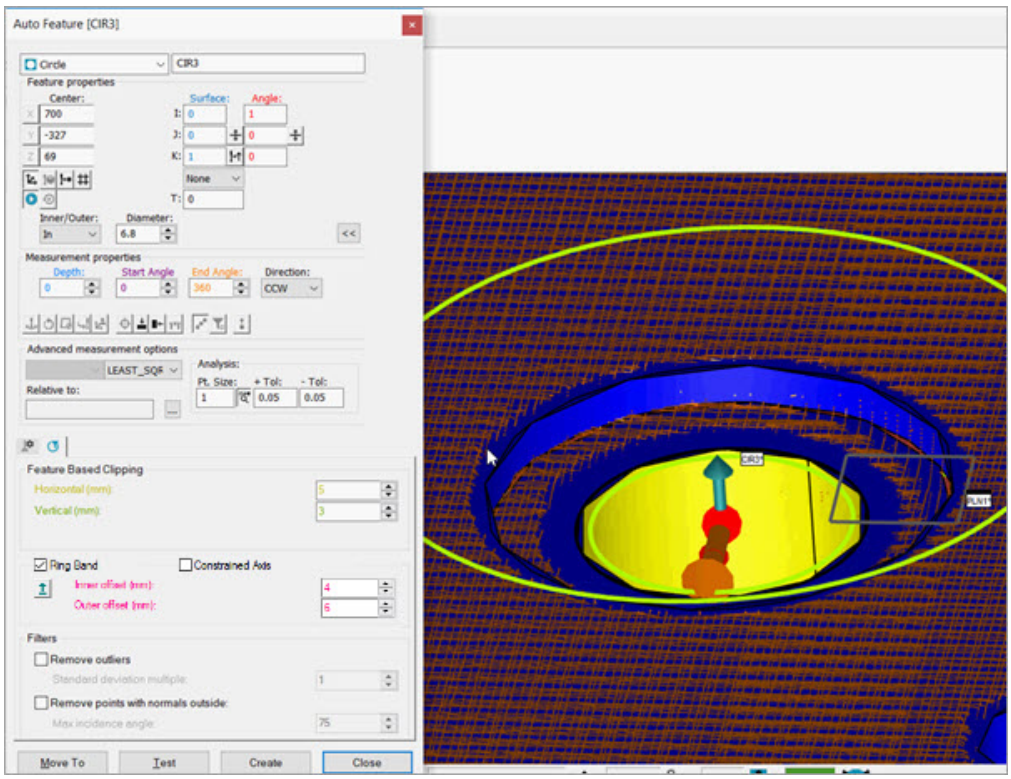
Prior to PC-DMIS version 2020 R1, when you select a cylinder on the CAD model with the **Ring Band** option, the software sets the Ring Band on a plane. The plane is a virtual plane located in the lower end of the cylinder feature based on the design of the CAD model.

The problem is that the plane is not real, and it is not correct to define a cylinder by piercing a virtual plane to get its intersection point. The reason is that, depending on the part you are inspecting, there may be multiple components between the cylinder and the plane.

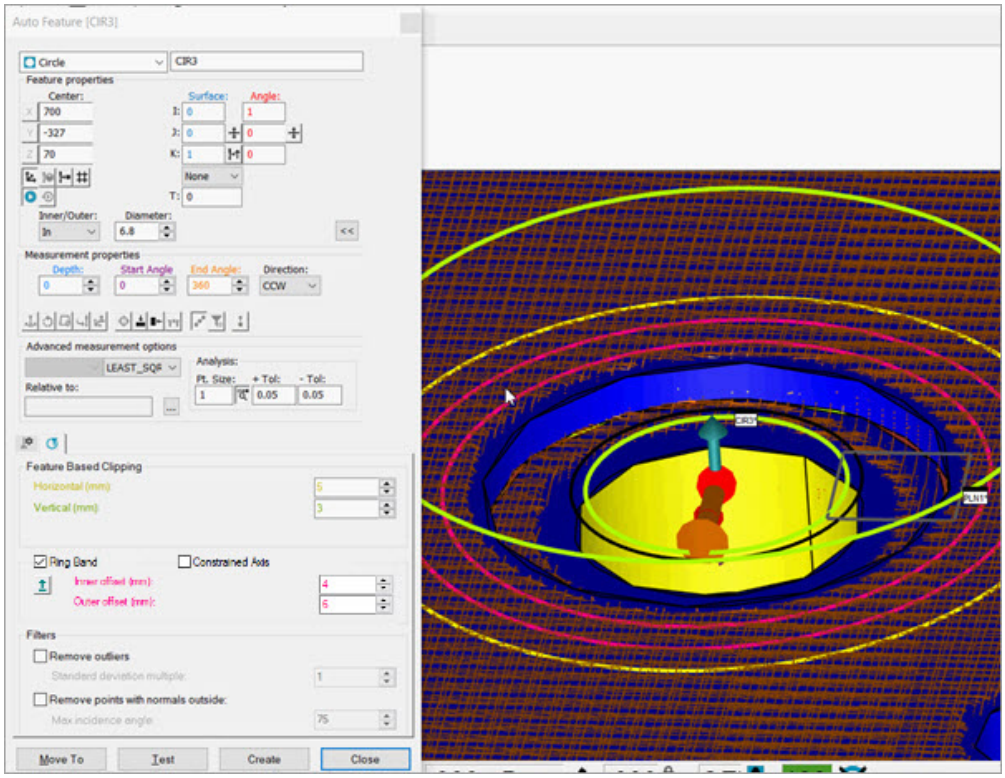
As a result, the nominal value of the intersection point between the cylinder and the plane is off in the Z axis by some value. You could get a correct measured value if you set the **Vertical Clipping** so that the real plane gets included in the **Ring Band** area, and therefore measured at the correct Z location. The problem is with the reported deviation, since the nominal value is wrong.

When you select the **Ring Band** check box, you enable the **Plane Selection On/Off** button. You can find this button in the **Ring Band** section of the **Feature Extraction** tab of the Probe Toolbox for laser. Click the button and select the desired plane in the CAD model to set the ring band controls on that plane, and to have the software properly update the nominals.

When you program a 2D feature (Circle, Polygon, Round Slot and Square Slot), you can select the plane that PC-DMIS programs and extracts the feature from. To do this, click the **Plane Selection On/Off** button and, in the Graphic Display window, select the plane. The Z nominal for the 2D feature moves to that plane. PC-DMIS calculates the selected plane at the defined **Depth** location from the selected plane. It then projects the feature onto this plane.



Example prior to selecting the Ring Band plane to extract 2D feature: Z nominal = 69



Example after selecting the Ring Band plane to extract 2D feature: Z nominal = 70

The **Depth** parameter setting is crucial for the calculation. The reason is that the feature extractor uses this value to search for the points to calculate the 2D feature. The calculation uses the **Depth** parameter value relative to the selected plane.

It is your responsibility to ensure that there are points at that depth in the pointcloud where you want to extract the feature.

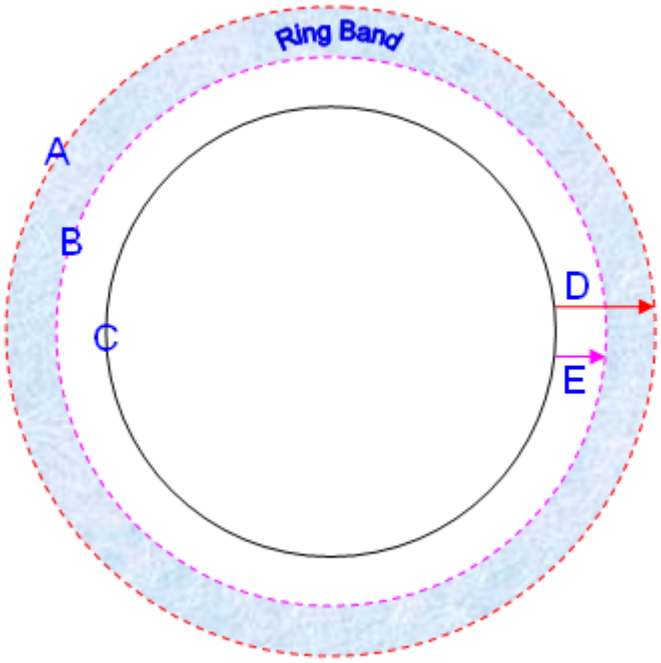
Inner offset and Outer offset

PC-DMIS uses these default values for Auto Circle, Auto Polygon, Auto Round Slot, and Auto Square Slot if Ring Band is disabled:

- **Inner offset** = 0.4 multiplied by the theoretical diameter value
- **Outer offset** = **Inner offset** value + 3 mm

Inner Offset - This value provides the offset from the theoretical feature radius or form for the *inner* edge of the ring band. This value is expressed in measurement routine units and must be greater than or equal to zero (a value of zero means an inner edge of the ring band coincides with the feature nominal.) See the image below.

Outer Offset - This value provides the offset from the theoretical feature radius or form for the *outer* edge of the ring band. This value is expressed in measurement routine units and must be greater than the **Inner Offset** value. See the image below.



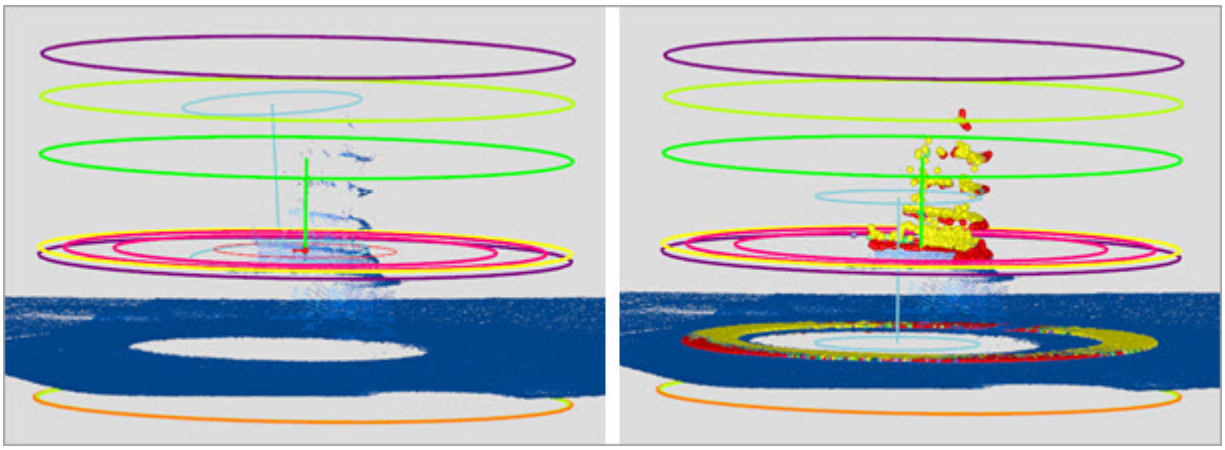
(A) Ring Band outer edge
(B) Ring Band inner edge
(C) Feature theoretical value

(D) Outer offset
(E) Inner offset

Constrained Axis

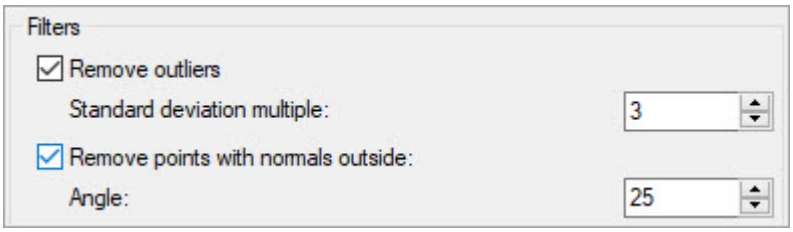
The Probe Toolbox **Feature Extraction** tab has an additional feature called **Constrained Axis** that you can use to constrain the Cylinder feature to the vector of the plane surrounding the cylinder. The plane is defined by the Ring Band area. The **Constrained Axis** option is only available for the Cylinder feature and is only available when you select the **Ring Band** option.

You can see the before and after effects of the **Constrained Axis** option in the images below. Even with a sparse set of data as shown below, this option calculates a more stable cylinder feature constrained to the plane's vector.



Examples with the Constrained Axis not selected (left) and selected (right)

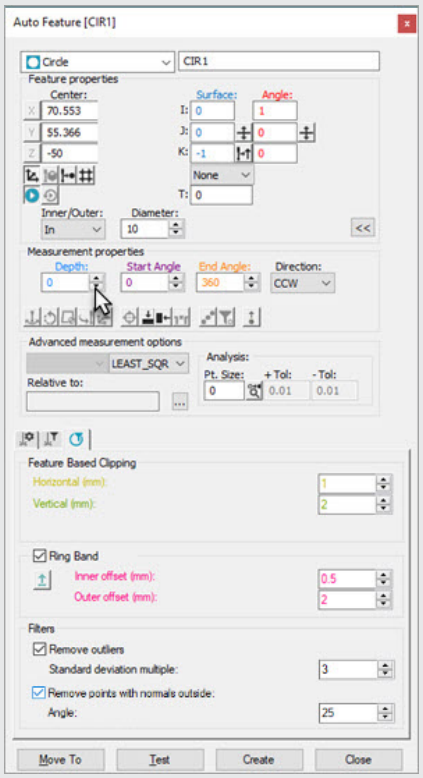
Filters



Feature Extraction - Filters area

Remove outliers - When you select this check box, it excludes outliers from the feature based on the value of the **Standard deviation multiple** option. The **Remove outliers** check box applies only to these Auto features: Cone, Cylinder, Flush and Gap, Plane, Sphere, and Surface Point.

For 2D laser Auto features (Circle, Polygon, Round Slot and Square Slot), the **Remove outliers** filter is only available if the **Depth** value in the **Measurement properties** area of the **Auto Feature** dialog box is set to 0 (zero).

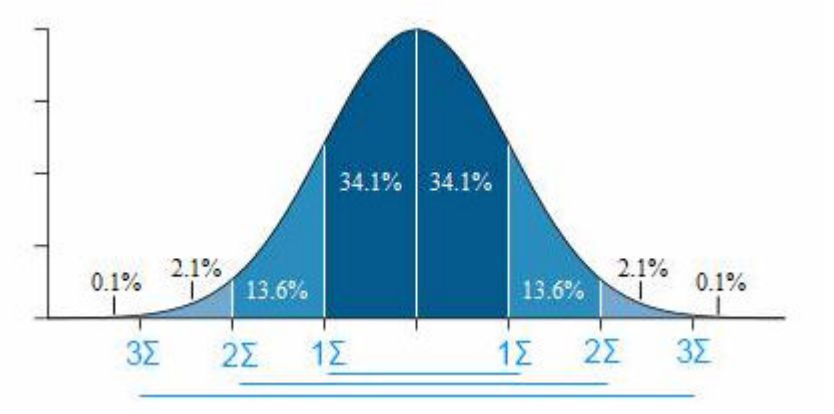


Example setting Depth to 0 (zero) to enable Remove outliers filter option

- The feature extractor evaluates the feature internally two or more times on the first attempt to get the standard deviation based on all points.
- In successive attempts, it re-evaluates the feature using only the points that are in the range of the outlier multiplied by the Σ (sigma). The sigma is the range, in the Gaussian distribution of the deviations, where the 68.2% of the best points used for fitting the feature lie.

  Standard deviation multiple - The value for this option defines the selectivity of the filter. It can be a generic real number that is greater than 0. If **m** is the selected value, it means that all the scan points that deviate from the extracted cone are greater than **m** x **Actual standard deviation** (that is, the standard deviation of the measured points with respect to the calculated feature) and are cut off from the calculation. Therefore, the lower the value of **m**, the more selective the filter.

In the first evaluation, the standard deviation is evaluated on all points. In a normal distribution, this could be represented as follows:



This means that the best points are in the interval from 0 to 1Σ. For example, if you wanted to get points only in that range, you would need to specify an outlier value from 0 to 1. Worse solutions would be obtained if you used higher outlier values.

Remove points with normals outside

If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area of the Probe Toolbox **Feature Extraction** tab, select the **Remove points with normals outside** check box and type the value in the Feature Extraction **Angle** box.

The Feature Extraction **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.

When you enable this setting, PC-DMIS compares the estimated normal of each scanned point within the clipping zone to the feature theoretical normal (or CAD surface for 3D features).

The **Remove points with normals outside** setting is only available for laser Circle, Cone, Cylinder, Edge Point, Flush and Gap, Plane, Polygon, Round Slot, Sphere, Square Slot, and Surface Point Auto features. The Edge Point and Flush and Gap features use the 2D filter method.

When measuring the laser feature, this filter excludes scanned points which are on the opposite side of the part or on adjacent surfaces. The smaller the Feature Extraction **Angle** value, the more points PC-DMIS excludes.

The effect of the Feature Extraction **Angle** filter is evident when you enable the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab of the laser **Auto Feature** dialog box.

3D Features Using the Feature Extraction Angle Filter

If you want to filter out any points that are outside of a maximum incidence angle, from the **Filters** area of the Probe Toolbox **Feature Extraction** tab, select the **Remove points with normals outside** check box and type the value in the Feature Extraction **Angle** box.

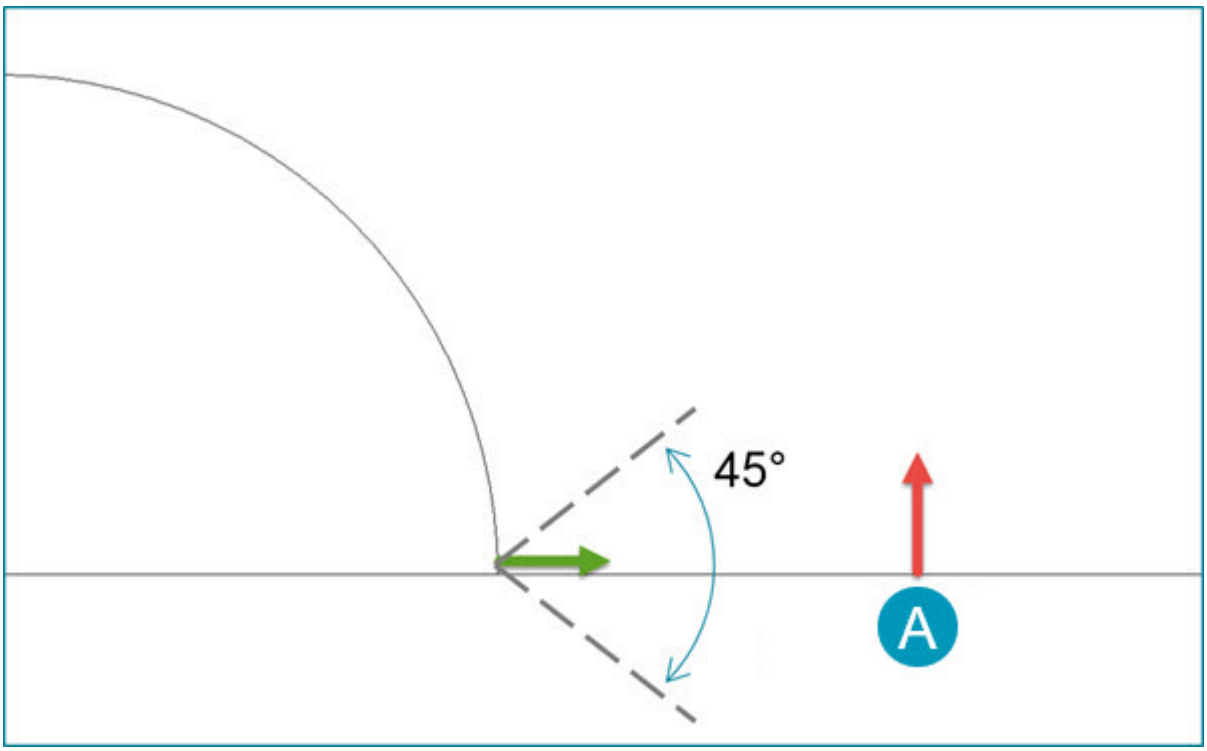
The Feature Extraction **Angle** filter was initially designed to compare the incidence direction of the laser (which is roughly the laser sensor orientation) to the estimated normal of the segregated points. For 3D laser features (Laser Auto Cone, Cylinder, Plane, Sphere, Surface Point and Constructed 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.

Laser Auto features have a Horizontal and Vertical clipping zone. PC-DMIS initially evaluates all scanned points within the clipping zone.

For 3D features (Surface Point, Plane, Cylinder, Cone, and Sphere), this setting compares the estimated normal of each scanned point to the feature theoretical normal or the vector of the CAD surface if a CAD model is used.

PC-DMIS excludes points with a vector that fall outside of this angle when measuring the feature.



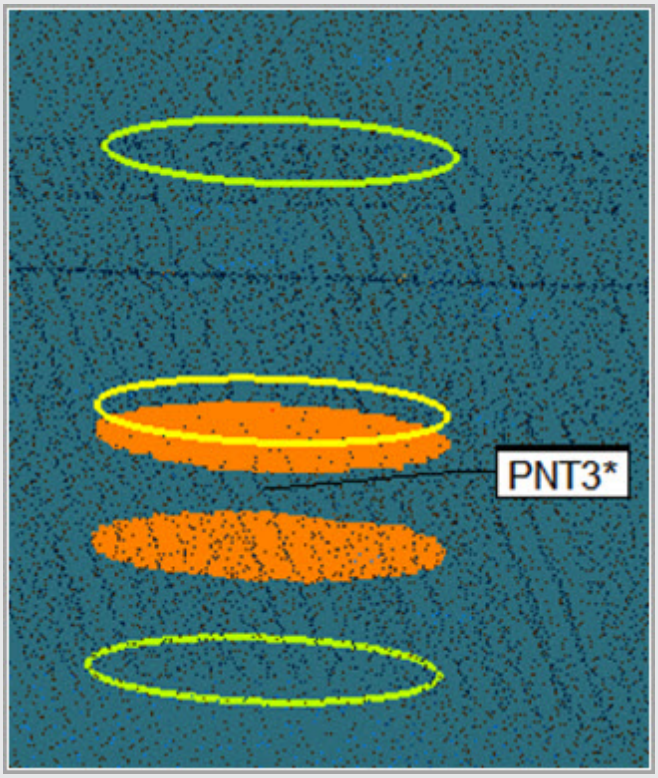
(A) - Plane (adjacent surface)

Example of the Feature Extraction **Angle** applied to a 3D laser Auto Point feature

Suppose you created a laser Auto Surface Point from a thin sheetmetal part that was scanned on both sides.

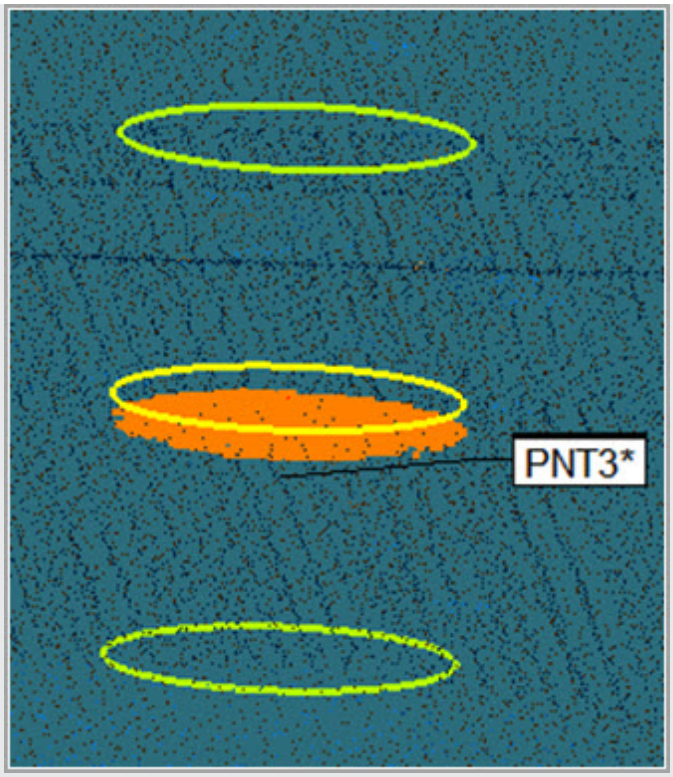
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case are larger than the sheetmetal thickness.

In this image, the scan uses no **Max incidence angle**:



Since the normals of the scanned points are not taken into consideration, the extracted point uses data from both sides of the part.

In this image, the scan uses a **Max incidence angle** of 60 degrees:

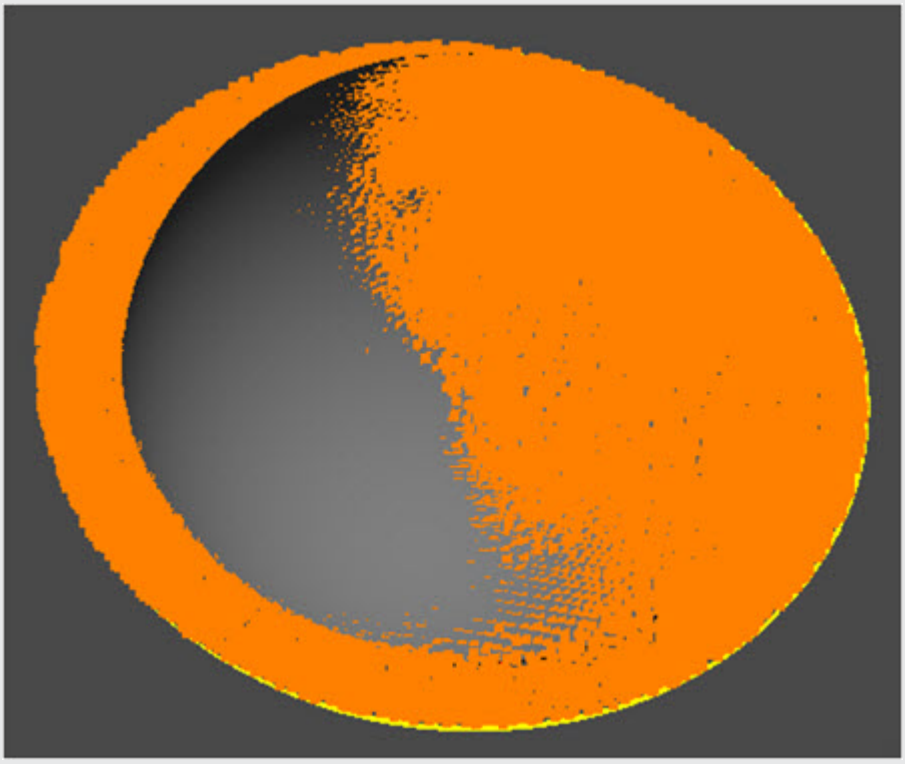


The software compares the estimated normal of each point in the clipping zone to the laser auto surface point theoretical normal. PC-DMIS does not use points that fall outside this angle for the feature calculation.

Example of the Feature Extraction **Angle** applied to a 3D laser Auto Sphere feature

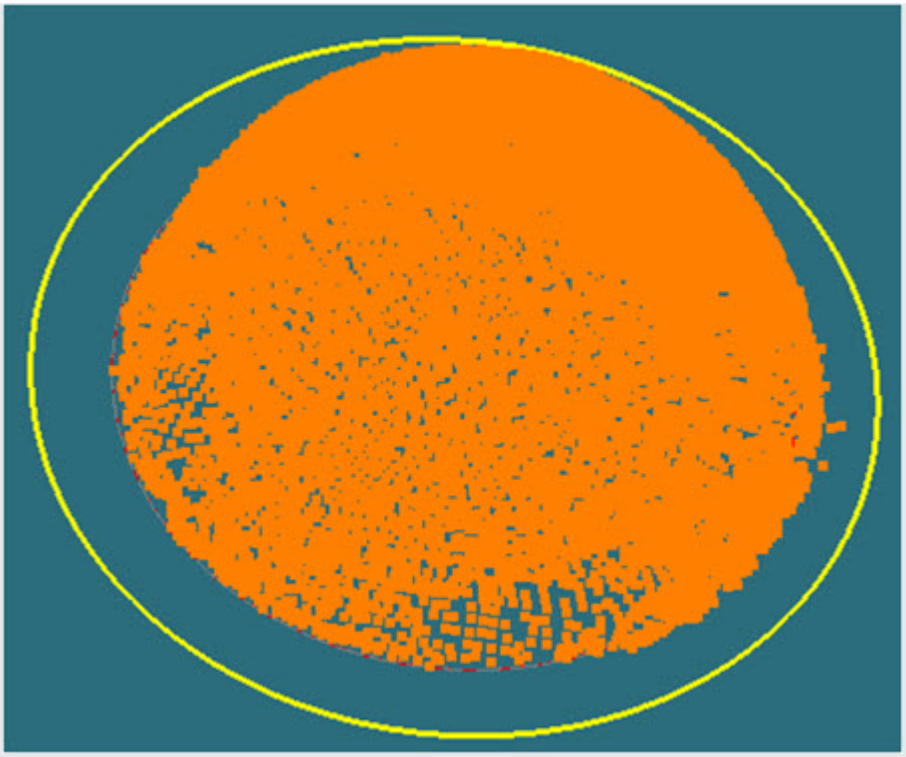
Laser extraction of a sphere previously required additional steps and manual selection to exclude adjacent surfaces.

In this image, no **Max incidence angle** is used:



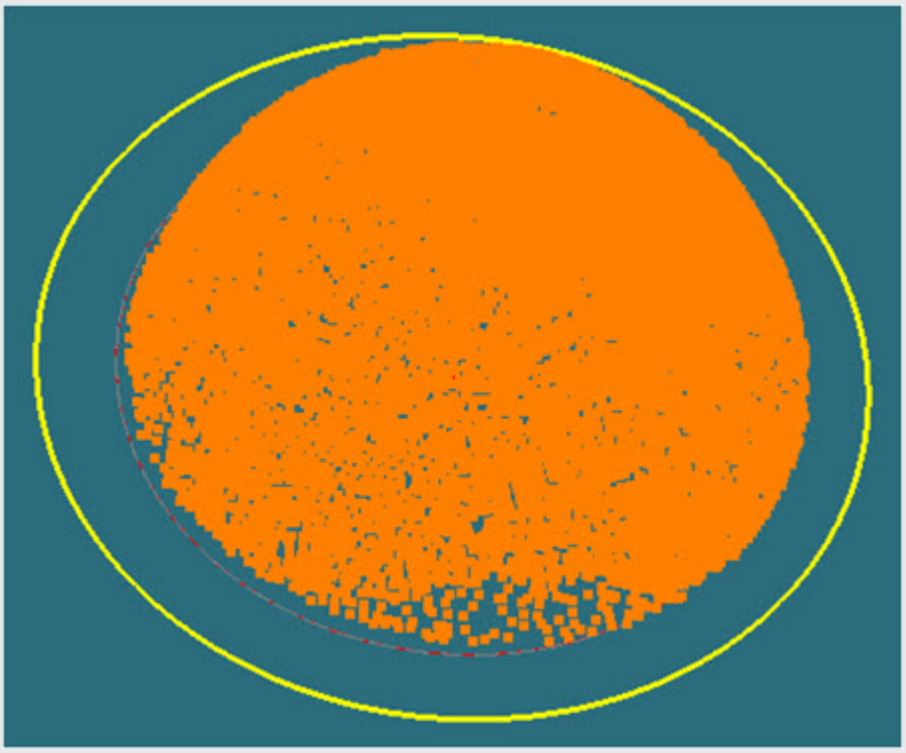
PC-DMIS uses data from the adjacent plane for sphere calculations.

In this image, a **Max incidence angle** of 60 is used:



PC-DMIS includes a few outlying points.

In this image, PC-DMIS uses a **Max incidence angle** of 45 degrees:

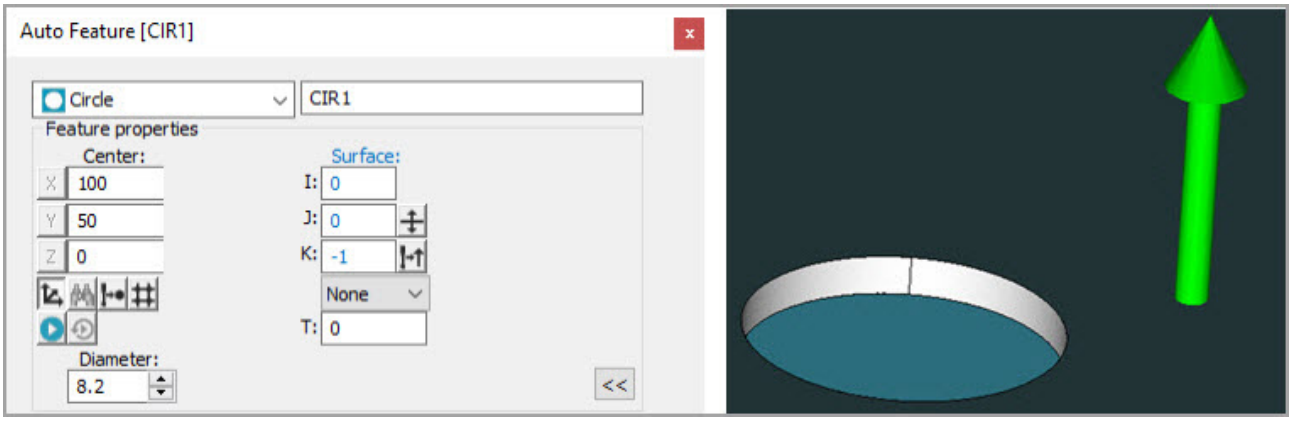


In this final example, the actual sphere data is best represented.

2D Features Using the Feature Extraction Angle Filter

Laser Auto features have a Horizontal and Vertical clipping zone. PC-DMIS initially evaluates all scanned points within the clipping zone.

For 2D features (Circle and Slots), this setting compares the estimated normal of each scanned point to the feature theoretical Surface Normal.



(A) - Surface vector

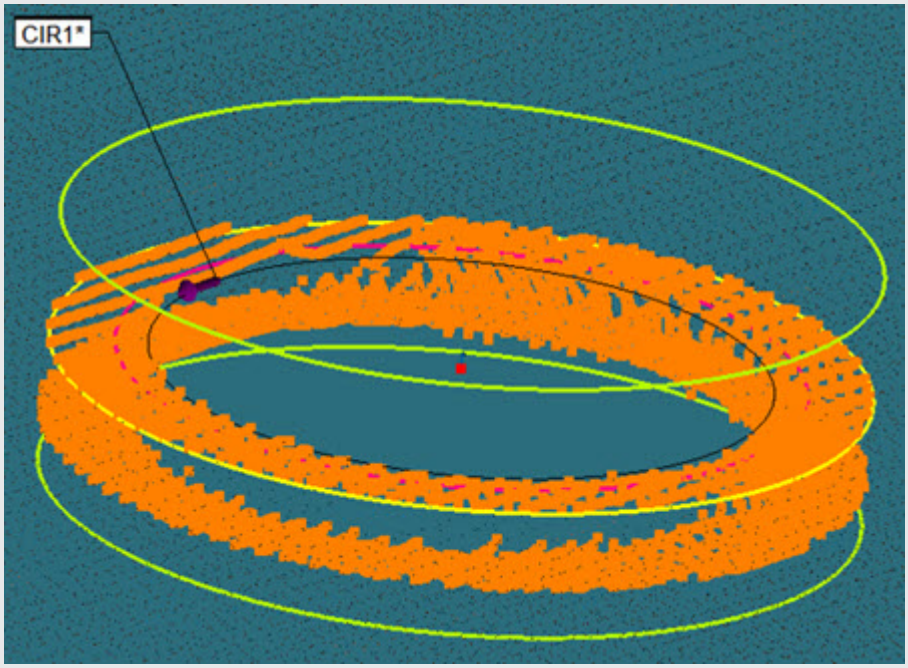
PC-DMIS excludes points with a vector that fall outside this angle when measuring the feature.

Example of the Feature Extraction **Angle** applied to a 2D laser Auto Circle feature

Suppose you created a laser Auto Circle from a sheetmetal part that was scanned on both sides.

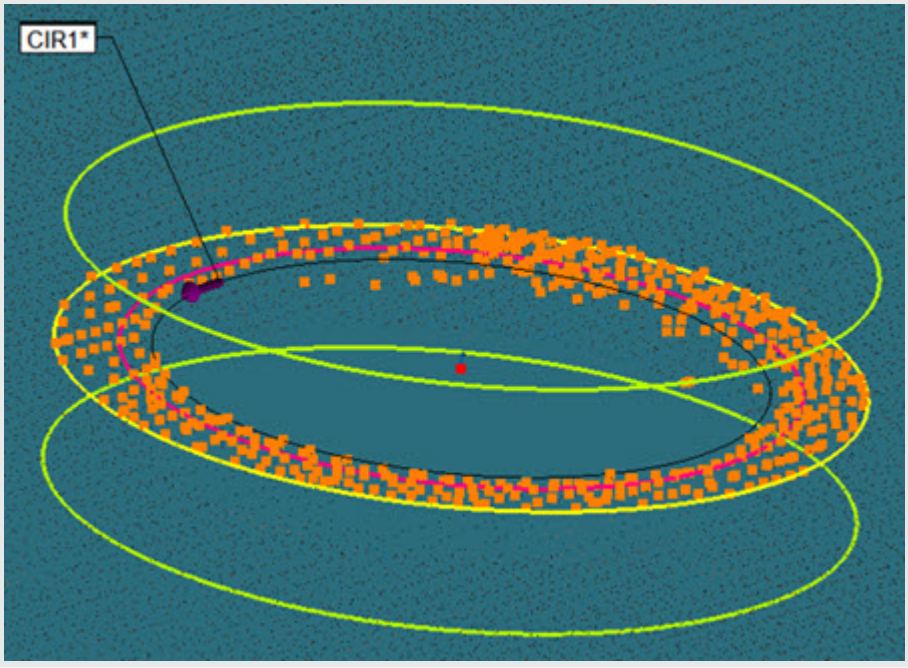
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case is larger than the sheetmetal thickness.

In this image, no **Angle** value is used:



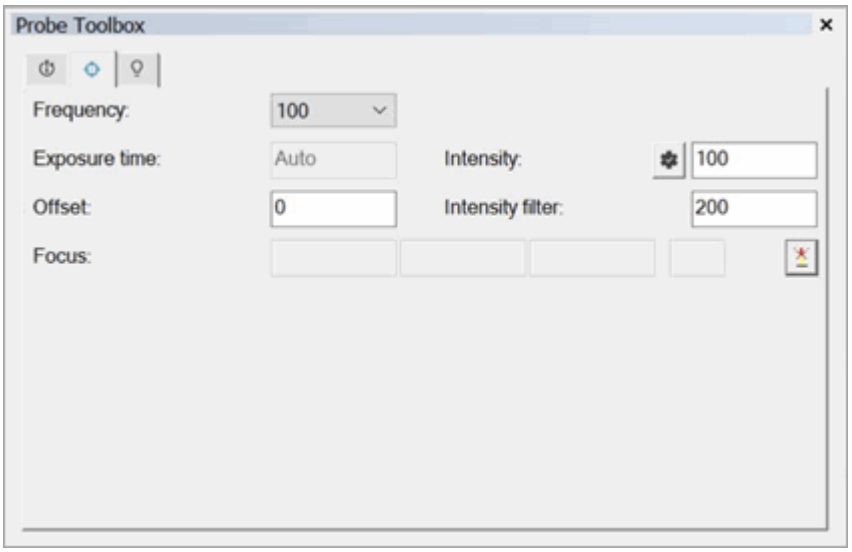
Since the normals of the scanned points are not taken into consideration, the extracted circle uses data from both sides of the part.

In this image, an **Angle** value of 75 degrees is used:



PC-DMIS compares the estimated normal of each point in the clipping zone to the laser Auto Circle, surface theoretical vector. PC-DMIS does not use the points with a vector which fall outside this angle for the feature calculation.

Laser Probe Toolbox: CWS Parameter tab



Probe Toolbox - CWS Parameter tab

The **CWS Parameter** tab in the Probe Toolbox (**View | Other Windows | Probe Toolbox**) is available once the system has been appropriately configured:

- You must configure the CWS as the active laser system. Usually, the factory does this locally during the startup procedure or by a service engineer.
- With the system configured, you must next define a probe with the correct properties. You can construct the probe in the **Probe Utilities** dialog box. You should use the OPTIV_FIXED selection and a lens that includes CWS. You should define this in the USRPROBE.DAT file if not provided by the factory.

+ TOLERANCE

Defines the upper tolerance value for the measurement.

- TOLERANCE

Defines the lower tolerance value for the measurement.

The **CWS Parameter** tab will only have the **+Tolerance** and **-Tolerance** options if your CWS is on an embedded controller.

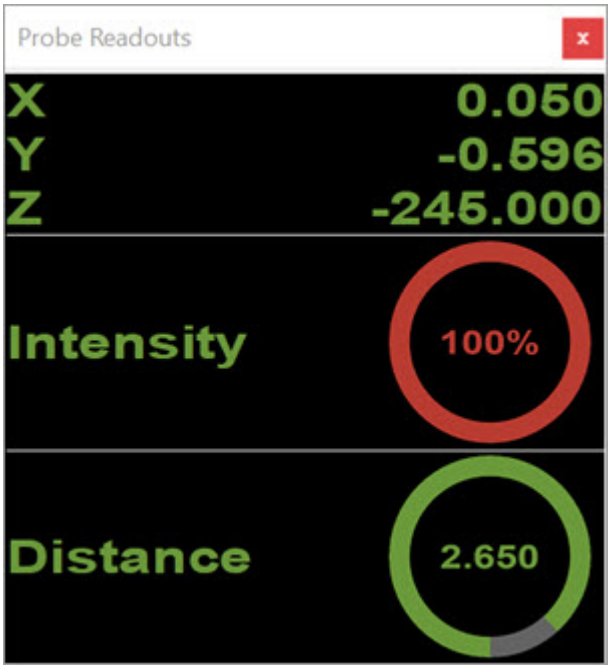
Frequency (Measurement Rate)

The measurement rate sets the number of measured values that the optical sensor records per unit of time. For example, when the measurement rate is set at 2000 Hz, 2000 measurement values are taken per second. The intensity indicator on the display can help you select the correct setting.

Setting Range

As a rule, you should strive to measure at the highest possible measurement rate in order to acquire as many measurement values in as little time as possible. In the case of surfaces with very low reflectivity, it may be necessary to reduce the measurement rate. This has the effect of illuminating the optical sensor's CCD-line longer and thus makes it possible to perform measurements, even if the reflected intensity is very low.

Overmodulation of the CCD-line on highly reflective surfaces and at small measurement rates can lead to measurement errors. If the intensity indicator on the CWS controller box displays a blinking **„Int: 999"**, or the Probe Readouts window displays an intensity value in red at or near 100%, overmodulation is occurring.



Probe Readouts window showing overmodulation

When overmodulation occurs, select the next-highest measurement rate. If the maximum measurement rate (2000 Hz on CHRocodileS, 1000 Hz on CHR150E) is already set, you can reduce the reflected intensity in one of two ways:

- Position the sensing head in the upper or lower threshold of the measurement range.
- Engage the **autoadaptfunction** (where the **Auto Intensity** parameter is set to **ON**). This adapts the lamp intensity continuously, depending on the part reflection. Here, a dark reference is not used. This is the method supported in PC-DMIS.

Exposure time (Brightness Value)

You can select the exposure time (brightness value) here if the **AUTO Intensity** parameter is set to **ON**.

The brightness of the lamp is modulated such that you achieve a defined percentage of the modulation amplitude. The value can lie in the range of 0% to 75%. For most applications, a recommended brightness value is between 20% and 40%.

Auto Intensity

This value defines the relative pulse duration of the LED, and with it the effective brightness of the light source.

If you are measuring a highly-reflective surface on which the highest measurement rate still results in overmodulation, then it makes sense to reduce the exposure time.

The best way to measure a poorly-reflective surface with a high measurement rate is to use a longer pulse duration.

Auto Intensity: OFF

Turn the **Auto Intensity** button () off to use the current light intensity of the LED.

Auto Intensity: ON

With the **Auto Intensity** button set to ON, the independent adjustment of flash time for the LED during an exposure time allows for you to automatically receive the best intensity settings when measuring on variable surfaces. This also allows for an optimal signal-to-noise ratio.

The system modulates the brightness of the lamp to achieve a defined percentage of the modulation amplitude. The value can lie in the range of 0% to 75%. For most applications, the recommended brightness value is between 20% and 40%.

Offset

This is the offset distance that the machine moves to in the measurement direction in addition to the measurement position.

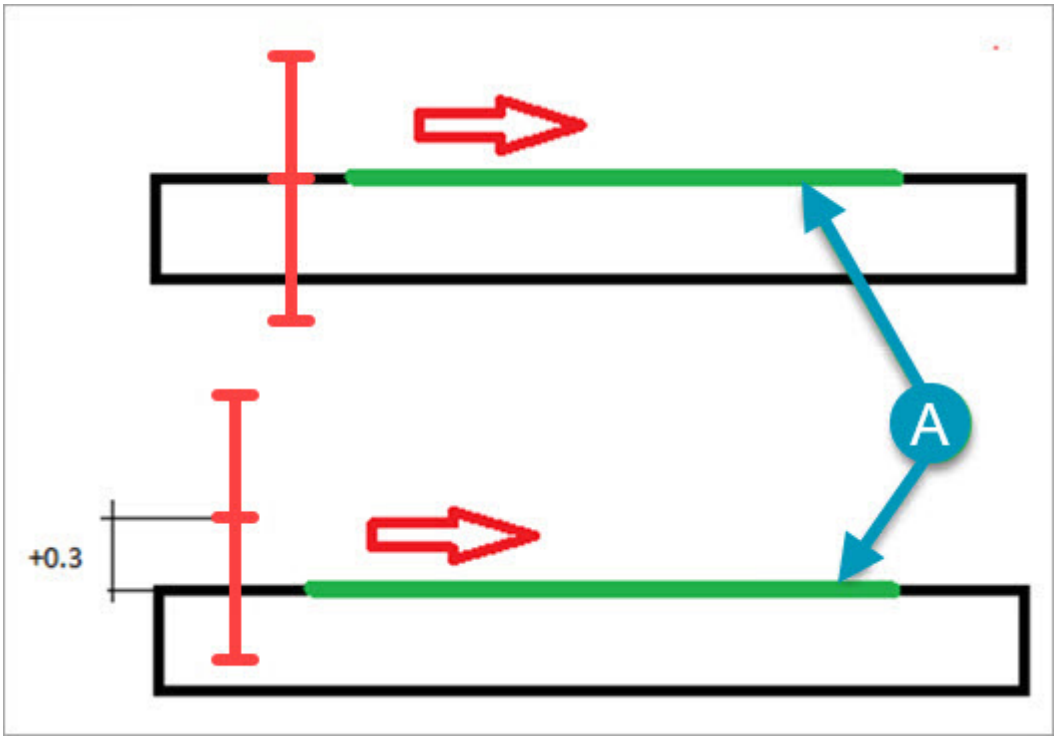
Offset support for the HP-OW (CWS) sensor requires that you have version 51.03.000 or later of the DC firmware installed.

For example, suppose you want to run a measurement routine with a feature to be measured using an HP-OW or CWS sensor. If the offset parameter does not equal 0 and the current DC firmware version does not support HP-OW offset parameter, PC-DMIS displays this error message:

PC-DMIS MESSAGE:

Laser parameter "Offset" not supported by machine controller.

Linear scan example before and after offset applied

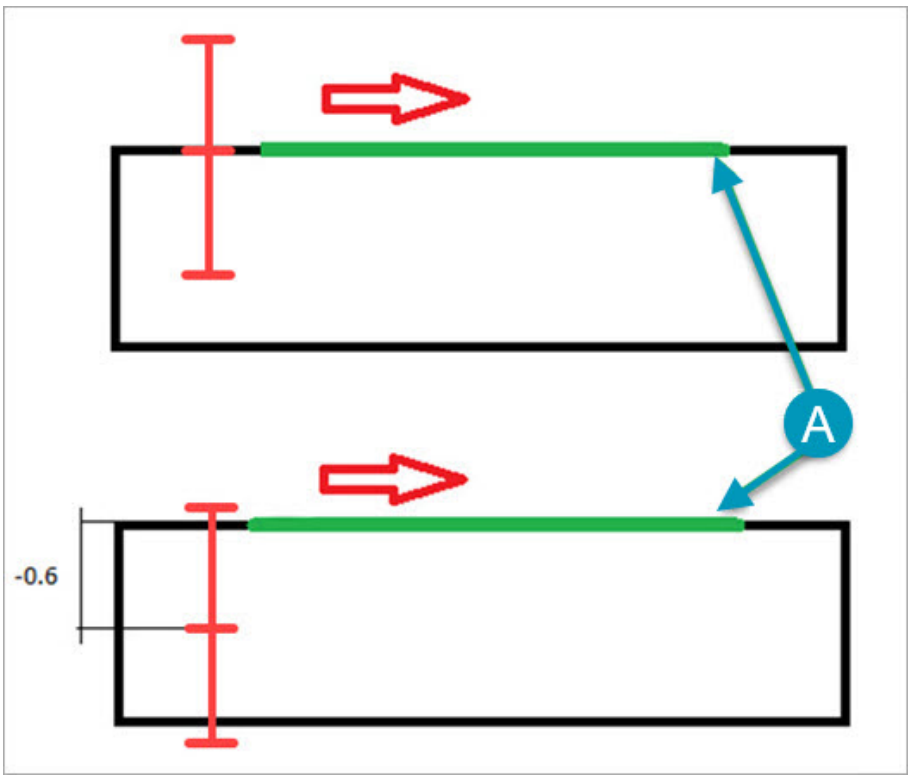


A *= Scan path*

Top *- Without an appropriate offset, the lower surface falls into the measurement range which affects the linear scan of the top targeted surface.*

Bottom *- With the appropriate offset (+0.3) applied, only the targeted top face of the feature is in the measurement range and the linear scan measurement is successful.*

Thickness scan example before and after offset applied



***A** = Scan path*

***Top** - Without an appropriate offset, the lower surface does not fall into the measurement range and the thickness scan fails.*

***Bottom** - With the appropriate offset (-0.6) applied, both faces of the feature are within the measurement range to successfully measure the thickness.*

Intensity filter

This value defines the threshold between noise and the measurement signal. The software recognizes that the peaks that fall beneath this threshold are invalid and show on the display as the measurement value "0".

For a valid measurement, the intensity should fall between 0 and 999 on CHRocodileS or 99 on CHR150E; otherwise, you must change the measurement rate.

If you measure the distance to a surface with low reflectivity, the intensity of the reflected light can be too low, and you must reduce the measurement rate. For a measurement rate under 1 kHz, the threshold value should be 40 on CHRocodileS or 25 on CHR150E. This prevents measurement values of too low an intensity, which rise only slightly above the noise, and would falsify the measurement.

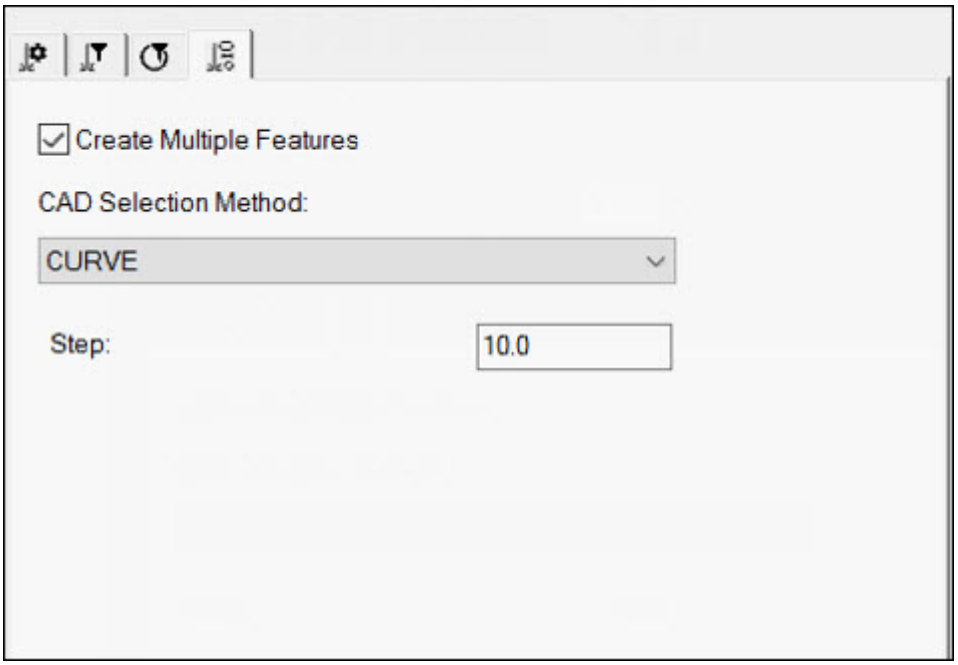
With a measurement rate of 1 kHz and higher (only for CHRocodileS), a threshold of 15 is best to fully exploit the device's dynamics.

Focus

This section has four boxes for the X, Y, Z, and Signal Quality. Click the **Auto Focus** button  on the right to execute a focus or surface measurement to display the X, Y, Z, and Signal Quality values.

For more details, see "CWS Parameters" in the PC-DMIS Vision documentation.

Laser Probe Toolbox: Laser AF Multiple Creation tab



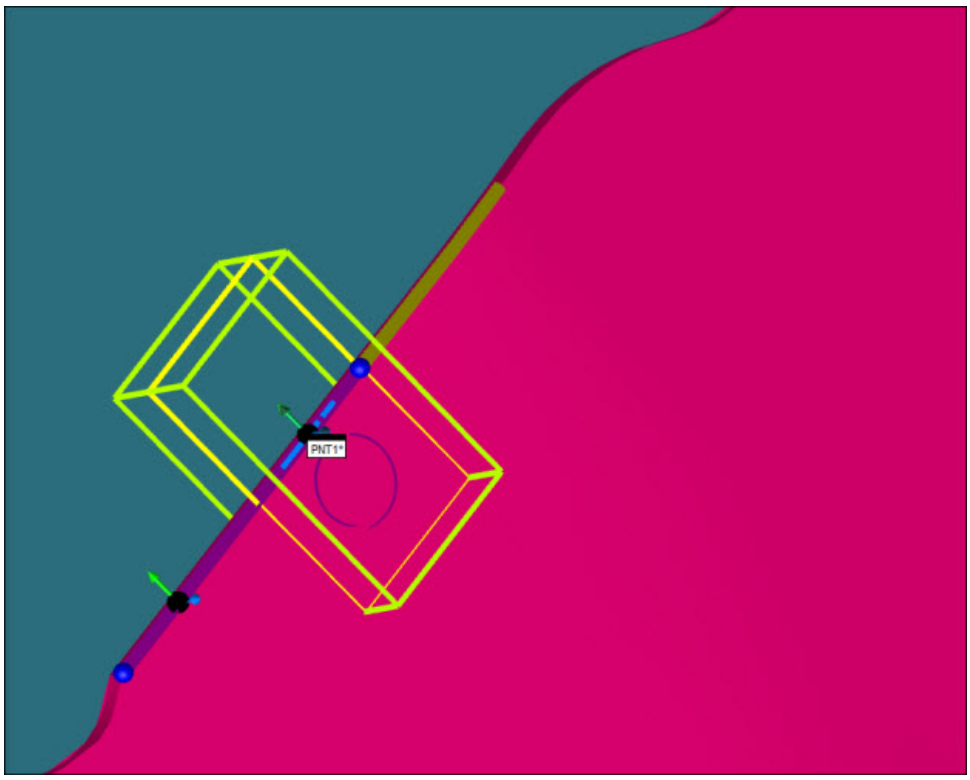
Probe Toolbox - Laser AF Multiple Creation tab

The **Laser AF Multiple Creation** tab is only available for the laser Edge Point Auto feature. This tab appears when the **Pointcloud** option on the **Laser Scan Properties** tab for the laser Edge Point Auto feature is set to a valid COP ID (the option is not set to **Disabled**).

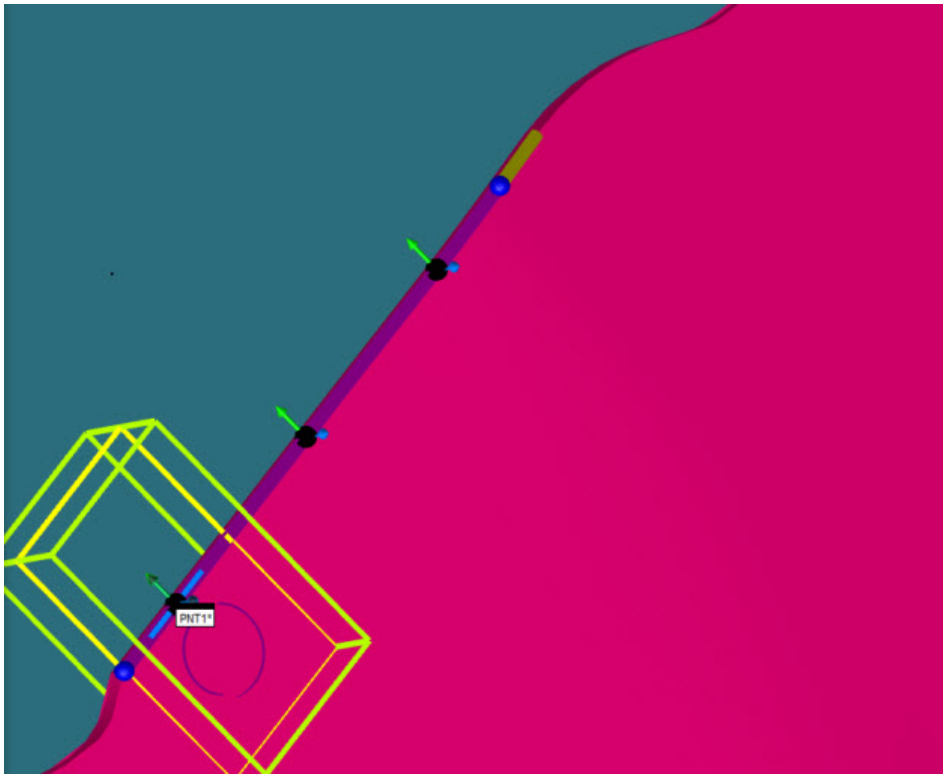
You can use this tab for extracted Auto features where the element is extracted from an existing COP object. You cannot use it for features that you directly measure (that is, features where the **Pointcloud** option is set to **Disabled**).

Create Multiple Features - To select curves on the model to create multiple features, select this check box. For Surface Point features, surfaces are selected instead. Note the following:

- The curves must be contiguous. To select or deselect them, press Ctrl. Consider these examples:

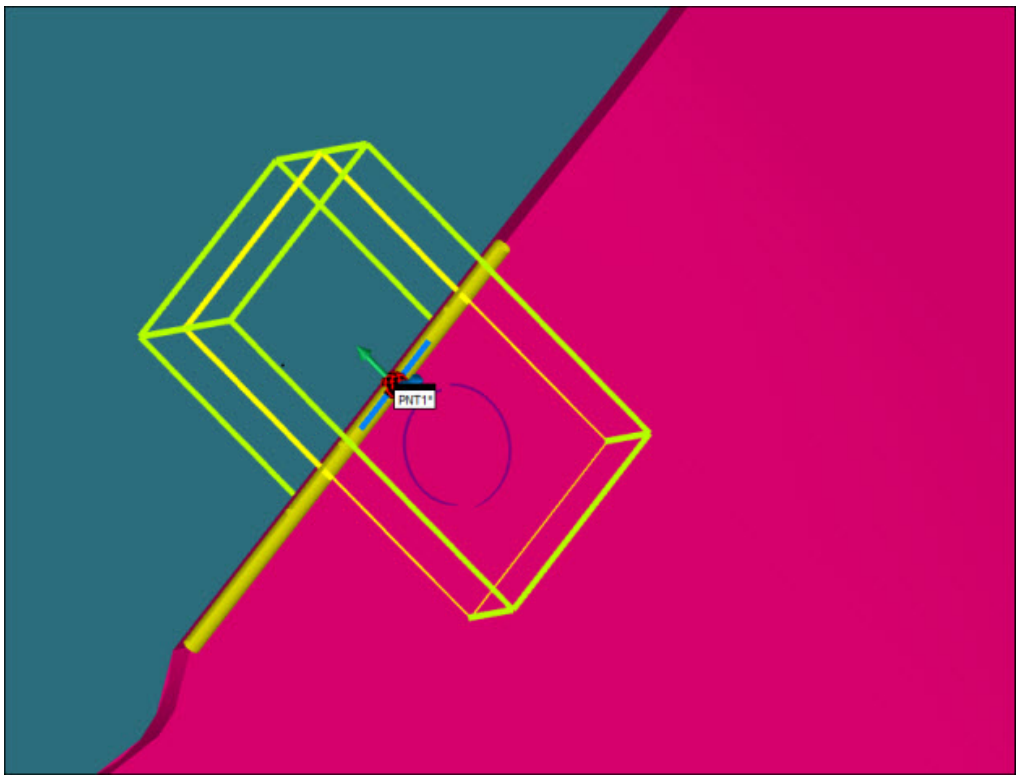


Using Ctrl to select additional contiguous curves



Using Ctrl to select additional contiguous curves

- The first point PC-DMIS creates on the curve is at a distance equal to the Horizontal Clipping value plus the Spacer value with respect to the start point of the curve itself. This is done on purpose so that PC-DMIS does not extract the first point off of the desired curve. For example:



First curve selection

- You can use the dragging functionality to select portions of the CAD curves. The features update accordingly.

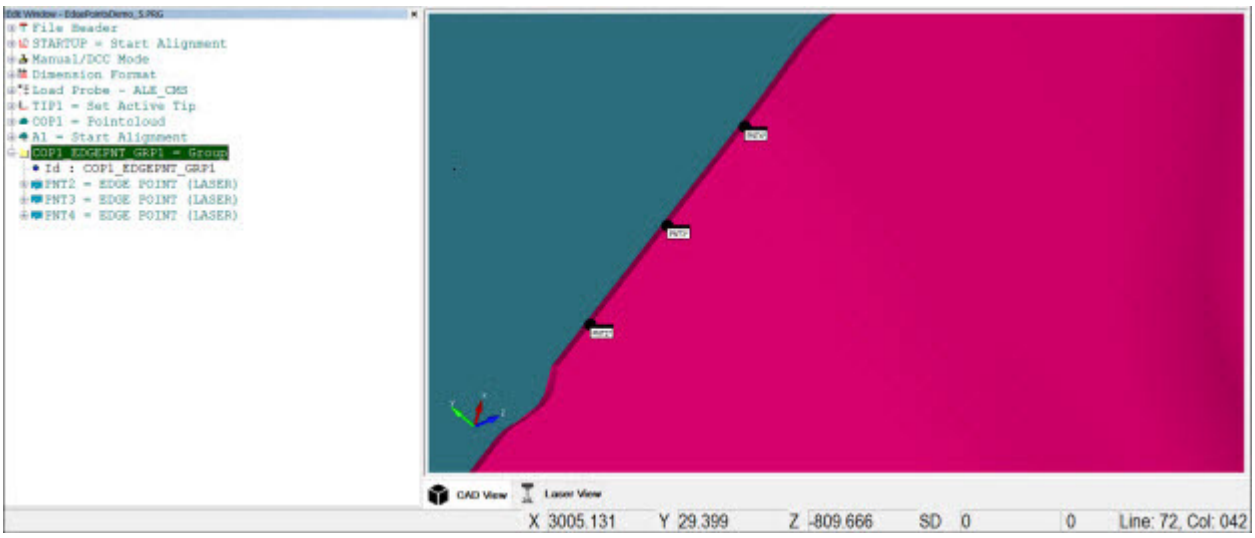
If you clear the **Create Multiple Features** check box, the edge point has surface and edge vectors set as a starting point to allow you to tune the extraction parameters. This has no effect on the vectors of the features that you create if you mark **Create Multiple Features**.

PC-DMIS creates the vectors for those features based on the selection of the surface close to the curve. In other words, the surface vector of the resulting features is the one on the surface (close to the curve) that you need to click to select the curve itself. Therefore, to avoid unpredictable or flipped vectors, we recommend that you do not click exactly on the curve.

CAD Selection Method - Select the desired CAD element.

Step - This option enables you to select the spacing along the selected curve or curves between the features you are creating.

A result of a multiple creation appears below:



Execution Modes

With PC-DMIS laser, you can use one of the following execution modes:

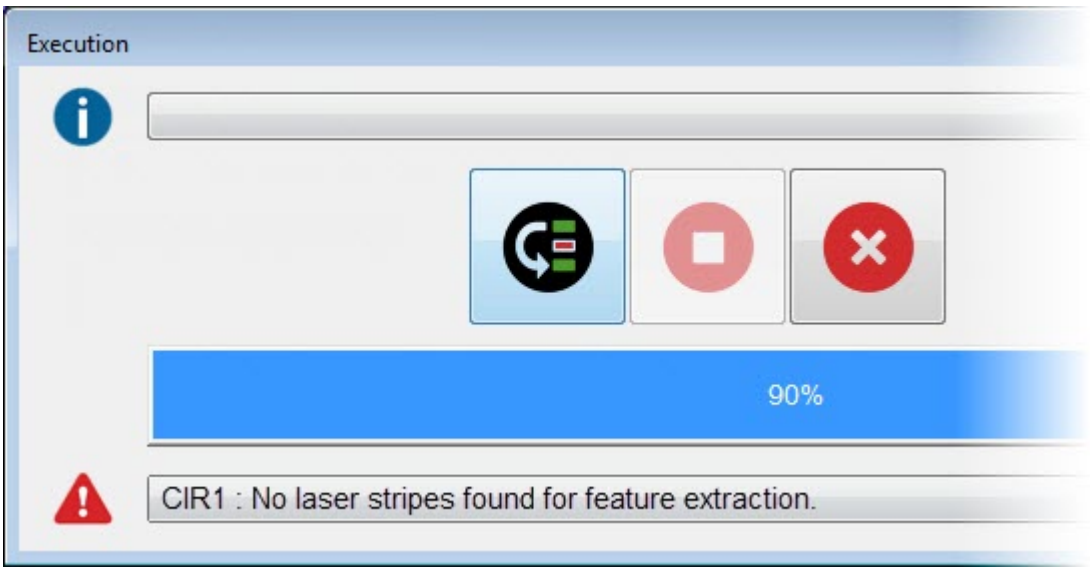
- Asynchronous Execution Mode (default mode)
- Sequential Execution Mode

Using Asynchronous Execution Mode

This is the default execution mode. In this mode, in order to speed up execution, the software ignores any feature calculation errors and proceeds to the next feature. If an error occurs during the execution of the measurement routine, the **Execution** dialog box displays these two options:

Cancel - This cancels the execution of the measurement routine.

Skip - This resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.



Execution dialog box

Asynchronous Execution Mode Example

Suppose you have three circles in sequence in your measurement routine. This execution mode behaves as follows:

Scan CIR1.

Begin extraction of CIR1 from its pointcloud.

Scan CIR2.

Begin extraction of CIR2 from its pointcloud.

Scan CIR3.

Begin extraction of CIR3 from its pointcloud.

If CIR2 fails to extract, it generates its error, but because the default execution mode continues execution, the calculation error may appear in the **Execution** dialog box while the machine is already scanning CIR3 or maybe even a later feature. Use Sequential Execution Mode if you want to pause the execution when measurement errors occur.

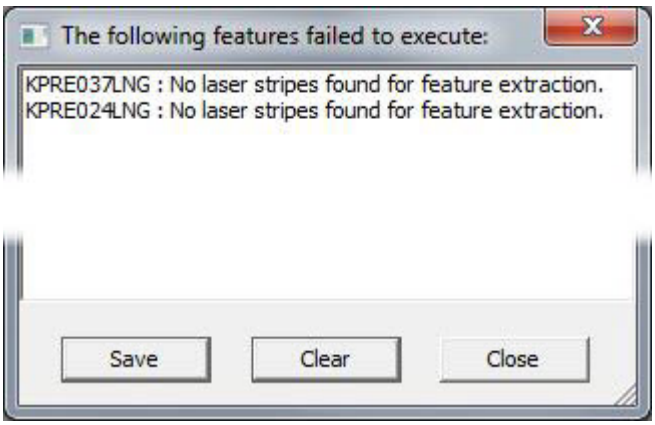
Using the On Error Command with this Mode

In Asynchronous Execution mode, if PC-DMIS encounters an error, and an On Error command has the Skip parameter defined as shown below, it hides the **Execution** dialog box and skips the feature that had the error:

```
ONERROR/LASER_ERROR, SKIP
```

Unless there are critical errors, the Skip parameter lets the measurement routine execute all the way through without anyone tending it.

After the entire measurement routine finishes execution, PC-DMIS displays the features that failed to execute in a dialog box. From that dialog box, you can click on any listed feature to locate the feature command in the Edit window and edit it as needed.



List of failed executed features dialog box

For detailed information on the On Error command, see the "Handling Laser Sensor Errors with On Error Command" topic.

Using Sequential Execution Mode

In Sequential Execution mode, when the measurement routine measures and calculates a feature, it does not proceed with the execution until it finishes calculating the current feature. This execution mode allows you to have concrete information about the problem feature when an error message does appear. In addition, execution stops when a message appears. This may help avoid collisions with the part. Sequential Execution mode is slower than the default (asynchronous execution) mode, but it allows you to monitor errors as they occur.

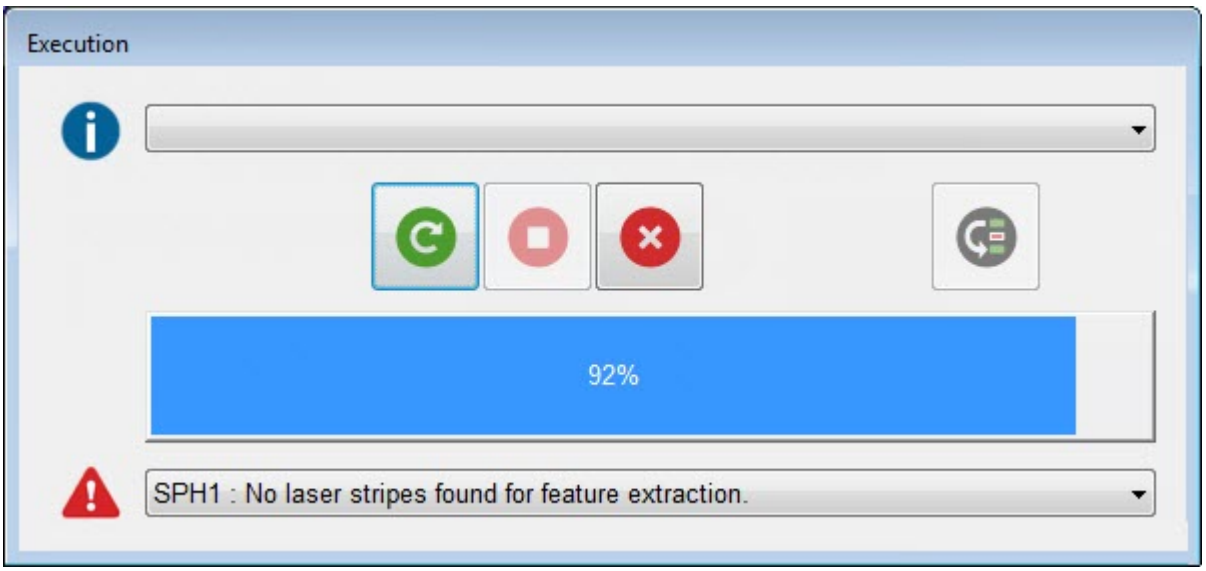
Generally, you should use this mode when executing a measurement routine for the first time, or when you want to test the machine's movements, laser parameters, or feature calculations.

If an error occurs during Sequential Execution mode, you have the following options in the **Execution** dialog box:

Cancel - This option cancels the execution of the measurement routine.

Skip - This option resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.

Try Again - This option retries the execution. It begins at the failed feature.



Execution dialog box

Enabling Sequential Execution Mode

To enable Sequential Execution mode, select **File | Execute | Sequential Execution** or, from the **Edit Window** toolbar, click the **Sequential Execution** icon.



Sequential Execution icon on the Edit window toolbar

The software shows this icon in a pressed state when in Sequential Execution mode. PC-DMIS only stays in Sequential Execution mode for the current execution. Afterward, it reverts to the default execution mode.

About the On Error Command

The On Error command does not work with Sequential Execution mode. PC-DMIS ignores any On Error command that it encounters. For detailed information on the On Error command, see the "Handling Laser Sensor Errors with On Error Command" topic.

Using Sound Events

Sound Events provide audible feedback in addition to the visual user interface. This allows you to perform measurement actions if away from the screen. To access the **Sound Events** tab of the **Setup Options** dialog box, select the **Edit | Preferences | Setup** menu item.

When you work with a laser device, these Sound Event options are particularly useful:

Laser Manual Calibration Bottom - This sound plays when calibration measurements for a given field should be taken in the upper region of the sphere.

Laser Manual Calibration Field Counter - This sound plays to indicate in which field measurements should be taken during calibration.

- 1 Beep - Far
- 2 Beeps - Left
- 3 Beeps - Right

Laser Manual Calibration Top - This sound plays when you need to take calibration measurements for a given field in the lower region of the sphere.

Laser Sensor Initialization End - This sound plays at the end of the laser sensor initialization.

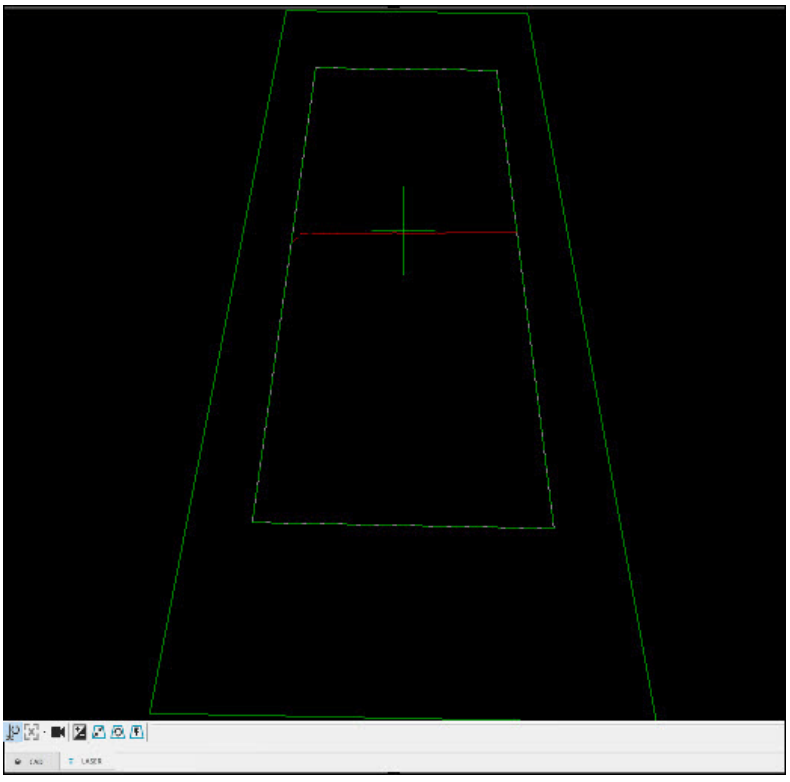
Laser Sensor Initialization Start - This sound plays at the beginning of the laser sensor initialization.

Laser Scan - This sound plays for each new step of the sensor calibration.

Using the Laser View

The Laser View is a view in the Graphic Display window that helps you visualize what the sensor "sees". You can access the Laser View whenever you click on the **Laser** tab.

You can use the Laser View during laser probe calibration, scanning, and Auto feature measurement. This tab shows what information PC-DMIS uses. During the scanning process, PC-DMIS disregards any data outside the clipping region rectangle. For more information, see the screen capture in "Laser Probe Toolbox: Laser Clipping Region Properties tab".



Graphic Display window - Laser tab

To turn the laser state on or off in the **Laser** tab, click the **Start/Stop** button (). When you make changes in the **Probe Toolbox**, you need to turn the laser state off and then back on to apply the changes in the **Laser** tab.

HP-L Sensor Addition

If you use an HP-L sensor, PC-DMIS shows these buttons:

 **AutoGain** - When the HP-L-5.8 sensor is in range of the part, click this button to learn the best gain setting and update the Probe Toolbox accordingly.

 **AutoClip** - Click this button to automatically set the clipping according to the data present in the **Laser** tab.

 **Reset Clipping** - Click this button to erase the existing clipping. This resets the entire sensor view for the selected scan zoom mode. For more information, see "Scan Zoom States (for HP-L Sensors)".

 **Center Part** - Click this button to center the part in the sensor's field of view.

In addition, for HP-L sensors, you can drag the clipping region with the mouse. This provides an alternative to adjusting the clipping region by typing values in the **Probe Toolbox**.

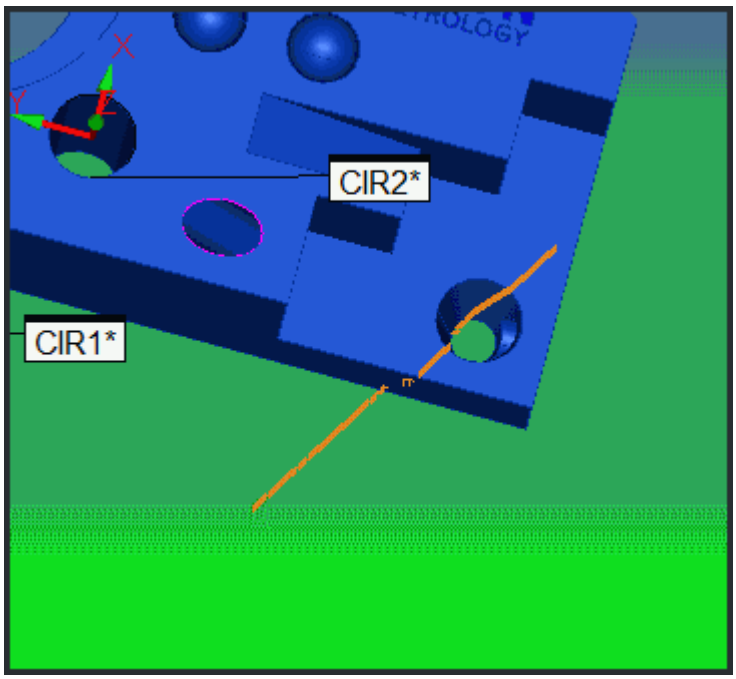
For details on the HP-L-10.10 sensor options, see the "Using Live View with the HP-L-10.10 Sensor" topic in the PC-DMIS Laser documentation.

Using the Scan Line Indicator

PC-DMIS Laser displays a colored scan-line indicator in the Graphic Display window to represent the location of the actual beam's scan line in 3D space. The indicator only functions when you run PC-DMIS in Online mode with an actual laser sensor that points at the part in real time.

To turn the scan-line indicator (and the Laser View) on or off, in the **Laser** tab, click the **Start / Stop** icon ().

If the beam is within range, it appears in the Graphic Display window and blinks whenever the laser beam pulses. As the beam moves toward the part, the indicator begins to change colors. As it nears the desired focal range, it changes colors from red, to orange, then yellow, then yellow-green, and finally to green.



A sample scan-line indicator (in orange) shows that the beam's scan-line position is too far above the part

This green color signifies that the beam is at the optimal distance away from the part for scanning.



A sample scan-line indicator (in green) shows that the beam's scan-line position is at the optimal focal distance

If you move the beam too close to the part, it again moves away from the desired green color and toward a red color.

Understanding the Visualisation Tools

PC-DMIS provides you with graphical overlays that it draws on top or around features that you create or edit in the Graphic Display window. These colored overlays give a visual perspective for matching colored parameters or settings in the **Probe Toolbox** and in the **Auto Feature** dialog box.

You can turn these visualisation overlays on or off with the **Visualisation Tools ON/OFF** icon from the **Laser Scan Properties** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**).



Visualisation Tools ON/OFF icon

Below are some examples. These examples cover all of the possible graphical overlays.

Explanation of Colored Overlays

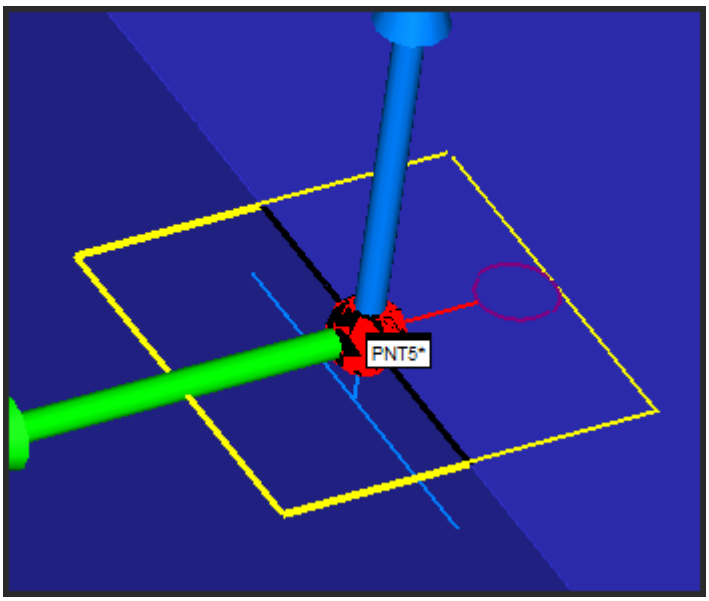
- **Yellow Line or Circle** - The **Overscan** region.
- **Blue Line or Circle** - The feature's **Depth** value.
- **Red Line** - The feature's **Indent** value.
- **Purple Circle** - The feature's **Spacer** value.
- **Pink Circles or Pink Rectangles** - The feature's **Ring Band** value.

Cones and Cylinders Overlays

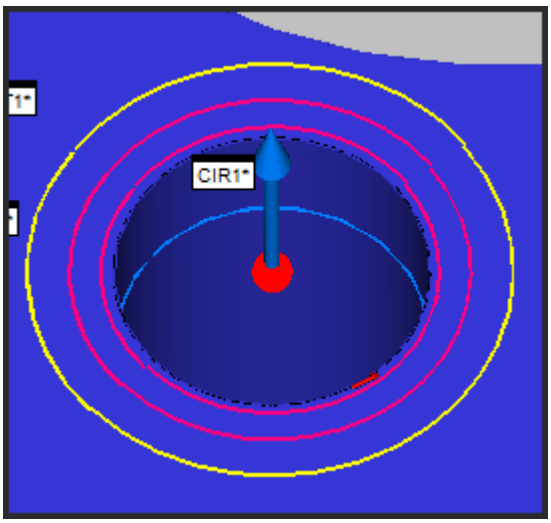
- *DCC Cylinders and Cones* show their bounds (the start and end points plus the **Overscan** value) in a light sea-green color. See the image of the DCC cone example below.
- *Portable Cylinders and Cones (or Feature Extraction only features)* show their bounds (the start and end points minus the **Vertical Clipping** value) in a lime-green color. See the image of the Portable cylinder example below.

For information on specific parameters or features, see the appropriate topics in the "Creating Auto Features with a Laser Sensor" section of the PC-DMIS Laser documentation.

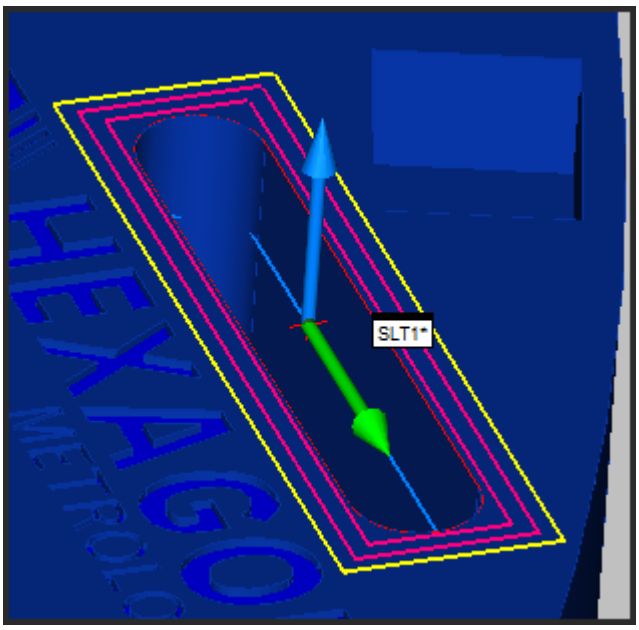
Some Sample Features with Overlays



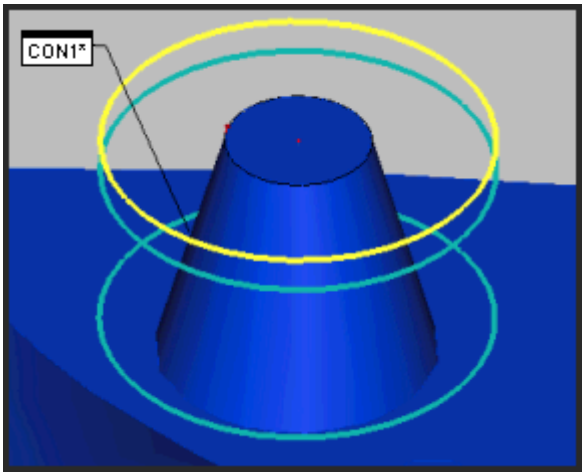
Sample Edge Point Auto feature



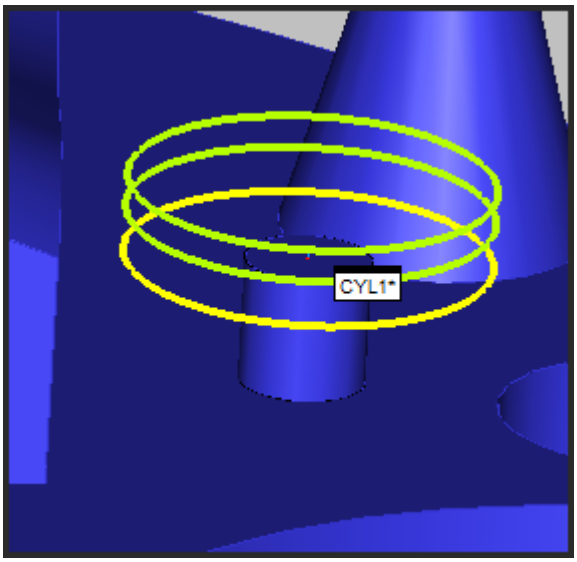
Sample Circle Auto feature



Sample Slot Auto feature



Sample DCC Cone Auto feature



Sample Portable Cylinder Auto feature

Pointcloud Scanning Colors

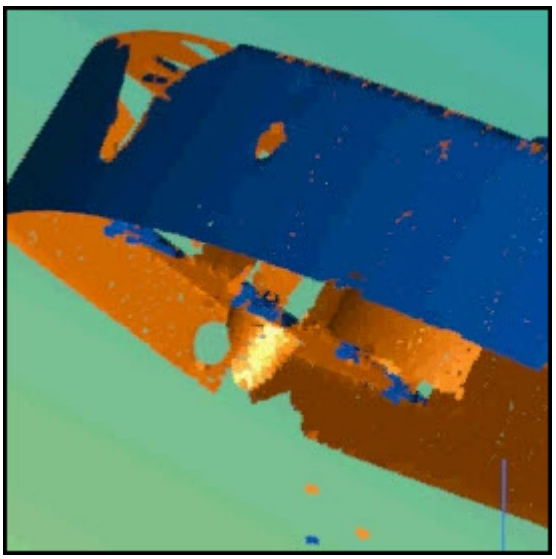
The following colors can help you interpret scanned pointclouds:

Blue - Existing scanned points of the outside of a part. Blue is the default outside color for a pointcloud. For information on how to change this color, see "Manipulating Pointclouds".

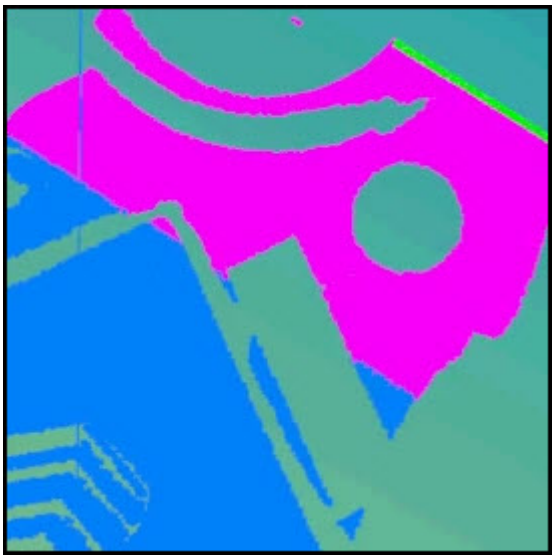
Orange - Existing scanned points of the inside of a part.

Magenta - Points currently being scanned.

Examples



Blue shows existing scanned points from the outside of a part. Orange shows the existing scanned points from the inside of a part.



Magenta shows the points currently being scanned.

Using the Laser Toolbars

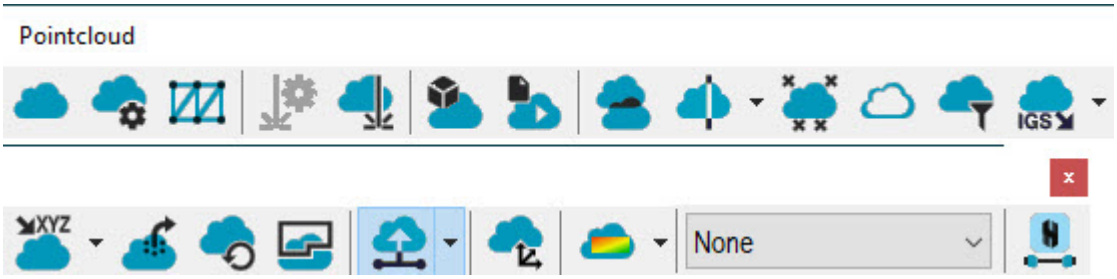
To decrease the time it takes to program your part, PC-DMIS Laser offers you a variety of toolbars composed of frequently-used commands. You can access these toolbars in two ways.

- Select the **View | Toolbars** submenu and select a toolbar from the menu.
- Right-click in the PC-DMIS **Toolbar** area and select a toolbar from the shortcut menu.

For a description of the standard PC-DMIS toolbars, see the "Using Toolbars" chapter in the PC-DMIS Core documentation.

The toolbars specific to laser functionality are below.

Pointcloud Toolbar



Pointcloud toolbar

The **Pointcloud** toolbar provides all pointcloud operations, features, and functions. You can access it from the **View | Toolbars | Pointcloud** menu, depending on your configuration.

> All options may not be available. Some options require specific licensing to enable them.

The following options are available from this toolbar:

Pointcloud - This button opens the **Pointcloud** dialog box that you can use to create pointcloud features. For details on this dialog box and how to create pointcloud features, see "Manipulating Pointclouds" in the "Using Pointclouds" chapter of the PC-DMIS Laser documentation.

 **Pointcloud Operator** - This button opens the **Pointcloud Operator** dialog box that you can use to perform different operations on pointcloud (COP) commands and other Pointcloud operator commands. For details on this dialog box and how to create pointcloud operators, see "Pointcloud Operators" in the PC-DMIS Laser documentation.

 **Pointcloud Mesh** - This button opens the **Mesh Command** dialog box that you can use to define a mesh command for pointclouds. For details, see "Creating a Mesh Feature" in the PC-DMIS Laser documentation. This option is only available if you have the Mesh and Big COP licenses.

 **Portable Scanning Widget** - This button displays the **Portable Scanning Widget** toolbar. When you connect to a portable device, and the active probe is a laser scanner, PC-DMIS automatically shows the **Portable Scanning Widget** toolbar. For details on the **Portable Scanning Widget** toolbar, see "Portable Scanning Widget Toolbar" in the PC-DMIS Portable documentation.

 **Pointcloud Data Collection Parameters** - This button opens the **Laser Data Collection Settings** dialog box that you can use to define data filtering and to define an exclusion plane for your pointcloud data. For details on this dialog box, see the "Laser Data Collection Settings" topic.

 **Simulate COP from CAD** - Click this button to open the **Simulate COP from CAD** dialog box. You can use the dialog box to create a theoretical pointcloud on the CAD surfaces that you select. For details, see "Simulate COP from CAD Dialog Box".

 **Simulate Pointcloud** - Click this button to open the **Simulate** dialog box. You can use the dialog box to select and import a pointcloud file. Once you import the pointcloud file, the button icon changes to the **Simulate Pointcloud - Stop** icon  . PC-DMIS then simulates the scanning of the imported pointcloud data. For details on simulating a scan of an imported pointcloud, see "Simulate Scanning by Importing a Pointcloud" in this documentation.

 **Pointcloud Boolean Operation** - This button opens the **Pointcloud Operator** dialog box with the Boolean operator selected. For details on the dialog box and on how to create a Boolean pointcloud operator, see "BOOLEAN" in the "Pointcloud Operators" chapter of the PC-DMIS Laser documentation.

 **Cross Section Pointcloud** - This button opens the **Pointcloud Operator** dialog box with the CROSS SECTION option selected.

To display the **Pointcloud Cross Section** toolbar, click the drop-down arrow:



For details on pointcloud cross sections and how to use the **Pointcloud Cross Section** toolbar, see "CROSS SECTION" in the "Pointcloud Operators" chapter of this documentation.

 **Clean Pointcloud** - When you click this button, the CLEAN operation immediately eliminates outlier COP points based on the default MAX DISTANCE of the points to the CAD. If the distance of a point is greater than the **Max distance** value, the software considers the point as an outlier or not belonging to the part. To use this operation, you must establish at least a rough alignment (see "Creating a Pointcloud/CAD Alignment") and a CAD model. For details on the CLEAN pointcloud operator, see "CLEAN" in the PC-DMIS Laser documentation.

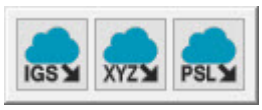
 **Empty Pointcloud** - When you click this button, PC-DMIS immediately removes all the data from the currently selected COP. Be aware that this change is permanent, so use with caution. For details on the EMPTY pointcloud operator, see "EMPTY" in the PC-DMIS Laser documentation.

 **Filter Pointcloud** - This button opens the **Pointcloud Operator** dialog box with the FILTER operation selected. The operation filters data to a smaller subset of points. For details on the FILTER pointcloud operator, see "FILTER" in the PC-DMIS Laser documentation.

 **Pointcloud Export** - This button opens the **Pointcloud Operator** dialog box for the currently selected export option.

For details on how to export supported file types, see "Pointcloud EXPORT" in the PC-DMIS Laser documentation.

To display the **Pointcloud Export** toolbar, click the drop-down arrow:

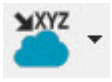


The options available are:

 **Export Pointcloud in IGES Format** - This button opens the **Pointcloud Operator** dialog box with the EXPORT IGES operation selected. The Export IGES operation exports the data in a COP or operator command in IGES format to an IGES file.

 **Export Pointcloud in XYZ Format** - This button opens the **Pointcloud Operator** dialog box with the EXPORT XYZ operation selected. The Export XYZ operation exports the data in a COP or operator command in XYZ format to an XYZ file.

 **Export Pointcloud in PSL Format** - This button opens the **Pointcloud Operator** dialog box with the EXPORT PSL operation selected. The Export PSL operation exports the data in a COP or operator command in PSL format to a PSL file.

 **Pointcloud Import** - This button opens the **Pointcloud Operator** dialog box for the currently selected import option.

For details on how to import supported file types, see "Pointcloud IMPORT" in the PC-DMIS Laser documentation.

To display the **Pointcloud Import** toolbar, click the drop-down arrow:



The options available are:

 **Import Pointcloud in XYZ Format** - This button opens the **Pointcloud Operator** dialog box with the **IMPORT XYZ** operation selected. The Import XYZ operation imports data from an external file into a COP command in the XYZ format.

 **Import Pointcloud in PSL Format** - This button opens the **Pointcloud Operator** dialog box with the **IMPORT PSL** operation selected. The Import PSL operation imports data from an external file into a COP command in the PSL format.

 **Import Pointcloud in STL Format** - This button opens the **Pointcloud Operator** dialog box with the **IMPORT STL** operation selected. The Import STL operation imports data from an external file into a COP command in the STL format.

 **Import Pointcloud in NSD Format** - This button opens the **Pointcloud Operator** dialog box with the **IMPORT NSD** operation selected. The .nsd file contains X, Y, Z points inside a binary file. These files are typically created by the 3DReshaper Meteor application.

 **Purge Pointcloud** - When you click this button, PC-DMIS immediately removes all data points that do not belong to this operator. It is irreversible and affects all other operator commands that refer to the same COP container, so use with caution. For details on the Purge pointcloud operator command, see "PURGE" in the PC-DMIS Laser documentation.

 **Reset Pointcloud** - When you click this button, PC-DMIS immediately reverses the most recent Surface Colormap, Point Colormap, Select or Clean (unless Purge has been done) operations. For details on the Reset pointcloud operator command, see "RESET" in the PC-DMIS Laser documentation.

 **Select Pointcloud** - Click this button to use the default Polygon selection method to select and remove a portion of the COP.

After you click this button:

- To define the polygon vertices, click in the Graphic Display window.
- To delete the last vertex, press the Delete key.
- To close the polygon selection, double-click or press the End key. PC-DMIS removes the portion of the pointcloud enclosed by the polygon.
- To abort, press the Esc key.

For details on the Select pointcloud operator command, see the "SELECT" topic in the PC-DMIS Laser documentation.



The **Select Pointcloud** option differs from the use of the pointcloud operator. It immediately applies the function and is not added as a command. To create the command, open the pointcloud operator and choose the **Select** method.

 **TCP/IP** - This button performs the currently selected operation described below.

To display the **TCP/IP** toolbar, click the drop-down arrow:



The options available are:

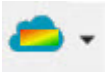
 **TCP/IP Pointcloud Server receive data** - This button places PC-DMIS in a "watch" state, where it is ready to receive a pointcloud file from a client

application. The client application must initiate sending the pointcloud data. This button only appears when you run PC-DMIS in Offline mode.

 **TCP/IP Pointcloud Server Connection with Local Copy** - This button establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data *remains inside* the measurement routine. For details on the TCP/IP Pointcloud server connection, see "TCP/IP Pointcloud Server".

 **TCP/IP Pointcloud Server Connection without Local Copy** - This button establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data *is deleted from* the measurement routine. For details on the TCP/IP Pointcloud server connection, see "TCP/IP Pointcloud Server".

 **Pointcloud Alignment** - This button opens the **Pointcloud/CAD Alignment** dialog box that you can use to create Pointcloud to CAD and COP to COP alignments. For details, see "Pointcloud/CAD Alignment Dialog Box Description" in the "Pointcloud Alignments" chapter of the PC-DMIS Laser documentation.

 **Pointcloud Colormap** - This button opens the dialog box for the operator shown on the button.

To display the **Pointcloud Colormap** toolbar, click the drop-down arrow:



The **Pointcloud Colormap** toolbar allows you to select between the **Surface Colormap**, **Point Colormap** and **Thickness Colormap** options.

From left to right, the buttons are:

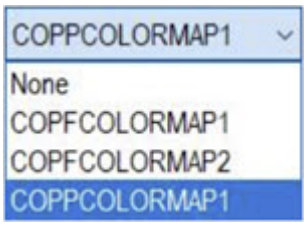
 **Surface Colormap** - This button opens the **Pointcloud Operator** dialog box with the Surface Colormap operator selected. The SURFACE COLORMAP operation applies a colored shading to the CAD model. The software shades the model according to the deviations of the pointcloud compared to the CAD. The Pointcloud Surface Colormap operator uses the colors defined in the **Edit Dimension Colors** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes.

You can create multiple surface colormaps in a PC-DMIS measurement routine. However, only one is active. The last surface colormap that was applied and created, or the last one executed, is always the currently active colormap. You can also select

which colormap is the active one from the **Colormaps** list box. You can also show or hide the active colormap from the **Activate Colormap** button ( ) on the **Graphic Items** toolbar, or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**). For details on how to show and hide colormaps with the **Activate Colormap** option, see the "Show / Hide Colormaps" section in the "Surface Colormap" topic.

When you activate a new colormap, PC-DMIS displays its associated scale with tolerance values and any annotations in the Graphic Display window.

To activate a colormap from the **Colormaps** list, click the **Colormaps** list box and from the list of defined Colormap operators, select the colormap:



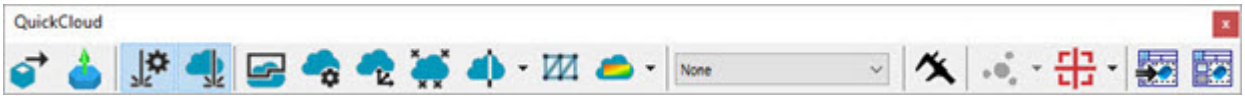
For details on the Pointcloud Surface Colormap operator, see "SURFACE COLORMAP" in the PC-DMIS Laser documentation.

Point Colormap - This button opens the **Pointcloud Operator** dialog box with the Point Colormap operator selected. The Point Colormap operation evaluates the deviations of the data points contained in a COP command compared to a CAD object. You can use this command to color the entire Pointcloud, or to display the points as dots, needles and/or text. For details on the Pointcloud Point Colormap operator, see "POINT COLORMAP" in the PC-DMIS Laser documentation.

Thickness Colormap - This button opens the **Pointcloud Operator** dialog box with the Thickness Colormap operator selected. The Thickness Colormap allows you to show and measure the part thickness as a colormap using only the Mesh or Pointcloud (COP) data object. You can also compare the measured thickness to the nominal CAD model thickness. For details on the **Thickness Colormap** option, see "Pointcloud Thickness Colormap" in this documentation.

Receive data from HHScan - When you click this button to On, PC-DMIS waits to receive a pointcloud or mesh from the Hexagon HHScan application. When this button is in the On state, it has a darker background color . For details, see the "Receive a Pointcloud or Mesh from the HHScan Application" topic in the PC-DMIS Laser documentation.

QuickCloud Toolbar



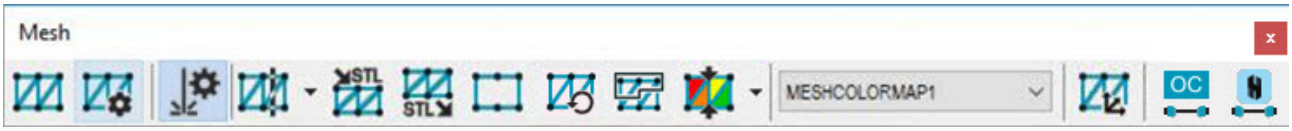
QuickCloud toolbar

The **QuickCloud** toolbar is only available when PC-DMIS is licensed and is configured as a Portable device. It provides the buttons to complete all the steps from beginning to end for working with COP.

For detailed information on this toolbar, see "QuickCloud Toolbar" in the "PC-DMIS Portable" documentation.

For details on all **Pointcloud** toolbar functions, see "Pointcloud Toolbar" in this documentation.

Mesh Toolbar



Mesh toolbar

The **Mesh** toolbar provides all mesh operations, features, and functions. It is accessible from the **View | Toolbars | Mesh** menu.

You must have the Mesh license enabled to use or view this option.

The following options are available from this toolbar:

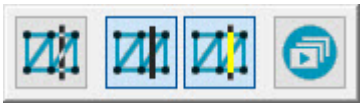
Mesh - This button opens the **Mesh Command** dialog box that you can use to create mesh features from any number of pointclouds. For details on this dialog box and how to create mesh features, see the "Creating a Mesh Feature" topic.

Mesh Operator - This button opens the **Mesh Operator** dialog box that you can use to perform different operations on a Mesh and other Mesh operator commands. For details on the dialog box and how to create mesh operators, see the "Creating a Mesh Operator" topic.

Portable Scanning Widget - This button displays the **Portable Scanning Widget** toolbar. When you connect to a portable device, and the active probe is a laser scanner,

PC-DMIS automatically shows the **Portable Scanning Widget** toolbar. For details on the **Portable Scanning Widget** toolbar, see "Portable Scanning Widget Toolbar" in the PC-DMIS Portable documentation.

Mesh Cross Section - This button opens the **Mesh Operator** dialog box that you can use to create a cross section from an existing mesh. Click the drop-down arrow to display the **Mesh Cross Section** toolbar:



For details on Mesh cross sections and using the **Mesh Cross Section** toolbar, see "Mesh CROSS SECTION Operator" in this documentation.

Import Mesh in STL Format - This button opens the **Mesh Import Data** dialog box that you can use to import an STL mesh data file. If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created, and the software imports the STL data. If a Mesh object already exists in the PC-DMIS Edit window, then the software adds the STL data to the Mesh object.

For details, see the "Mesh IMPORT Operator" topic.

Export Mesh in STL Format - This button opens the **Export Mesh Data** dialog box that you can use to export a Mesh in an STL, ASCII, or STL Bin file format.

For details, see the "Mesh EXPORT Operator" topic.

Empty a Mesh - This button empties the first mesh relative to the cursor position in the Edit window.

> Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.

For details, see the "Mesh EMPTY Operator" topic.

Reset Mesh - Use this button to revert all Mesh SELECT operations and return the original Mesh object. For details on the Reset Mesh operator command, see "Mesh RESET Operator" in the PC-DMIS Laser documentation.

Select Mesh - Use this button to select and delete a subset of triangles contained in the mesh data object. When you use this button, the selection method uses a polygon to remove triangles in the 3D view.

The **Select Mesh** button option differs from the use of the Mesh Select operator from the dialog. When you click this button, PC-DMIS immediately applies the function but does not add a Select command. To create the command, open the **Mesh Operator** dialog and choose the **Select** function.

After you click this button:

- Click in the Graphic Display window to define the polygon vertices.
- Press the Delete key to delete the last vertex.
- Double-click your left mouse button or press the End key to close the polygon selection. PC-DMIS removes the portion of the mesh enclosed by the polygon.
- Press the Esc key to abort.

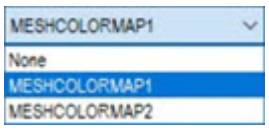
For details, see the "Mesh SELECT Operator" topic.

 **Colormap a Mesh:** This button opens the **Mesh Operator** dialog box that you can use to create a Mesh COLORMAP operator. For details, see the "Mesh COLORMAP Operator" topic.

The **Colormap a Mesh** operation applies a colored shading to the selected mesh. PC-DMIS shades the model according to the deviations of the mesh compared to the CAD. The **Colormap a Mesh** operation uses the colors defined in the **Edit Dimension Colors** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes. For details on the **Colormap a Mesh** operator, see the "Mesh COLORMAP Operator" topic in the PC-DMIS Laser documentation.

You can create multiple colormaps in a PC-DMIS measurement routine. However, only one is active. The last colormap that was applied and created (pointcloud surface colormap or mesh colormap), or the last one executed, is always the currently active colormap. You can also select which colormap is the active one from the **Colormaps** list box. When you activate a new colormap, PC-DMIS displays its associated scale with tolerance values and any annotations in the Graphic Display window.

To do this, click the **Colormaps** list box and select the colormap from the list of defined colormap operators:



Click the drop-down arrow to display the **Mesh Colormap** toolbar:



The **Mesh Colormap** toolbar allows you to select between the **Colormap a Mesh** and **Thickness Colormap** options. For details on the **Thickness Colormap** option, see "Thickness Colormap" in this documentation.

Mesh Alignment: This button opens the **Mesh/CAD Alignment** dialog box that you can use to create Mesh-to-CAD alignments.

For details, see the "Mesh ALIGNMENT" topic.

Receive a mesh from OptoCat: When you click this button ON, PC-DMIS waits to receive a mesh from the OptoCat application. When the **Receive a mesh from OptoCat** button is in the On state, it has a darker background color . For details on how this works, see the "Receive a Mesh from OptoCat" topic.

Receive data from HHScan - When you click this button to On, PC-DMIS waits to receive a pointcloud or mesh from the Hexagon HHScan application. When this button is in the On state, it has a darker background color . For details, see the "Receive a Pointcloud or Mesh from the HHScan Application" topic in the PC-DMIS Laser documentation.

HxGN Robotic Automation Toolbar



HxGN Robotic Automation toolbar

The **HxGN Robotic Automation** toolbar provides all the options necessary to connect to and communicate with a system running the HxGN Robotic Automation application. You can access the toolbar from the **View | Toolbars | HxGN Robotic Automation** menu.

> The **HxGN Robotic Automation** toolbar is only available if you have the ROY-AUTOMATE license enabled.

The following options are available from this toolbar:

Export to HxGN Robotic Automation - Click this button to begin the export of PC-DMIS measurement routines and inspection plans to HxGN Robotic Automation. For details on the PC-DMIS to HxGN Robotic Automation workflow, see the "PC-DMIS Workflow Using HxGN Robotic Automation" topic in this documentation.

 **TCP/IP Pointcloud Server receive data** - This button places PC-DMIS in a "watch" state, where it is ready to receive a pointcloud file from a client application. The client application must initiate sending the pointcloud data. This button only appears when you run PC-DMIS in Offline mode.

Using Pointclouds

The Pointcloud command (COP) allows you to store XYZ coordinate data that can come directly from a laser sensor through one or more referring scan commands. You can also enter data directly into a COP from other PC-DMIS features or external data files.

You can add pointclouds to your measurement routine in these ways:

- Select the **File | Import | Pointcloud** submenu and select a data file to import (XYZ, PSL, STL, or NSD).

  STL: The STL file type is the same file type that is covered in the "Importing an STL File" topic of the PC-DMIS Core documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

  XYZ: The XYZ file type is the same file type that is covered in the "Importing an XYZ File as CAD Data" topic of the PC-DMIS Core documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

- Select the **Insert | Pointcloud | Feature** menu item to open the **Pointcloud** dialog box.

- Manually type the COP command into the Edit window. Press F9 on the COP command in the Edit window to open the **Pointcloud** dialog box. For information on the COP command mode text, see "COP Command Mode Text".

- From the **Pointcloud** toolbar, click the **Pointcloud** button () to open the **Pointcloud** dialog box.

For information on how to manipulate pointclouds from the **Pointcloud** dialog box, see the "Manipulating Pointclouds" topic.

PC-DMIS uses additional, laser sensor-related commands and tools that support pointcloud functionality. They are:

- Pointcloud Operators
- Pointcloud Alignments
- Pointcloud Point Information
- Laser Data Collection Settings

Your PC-DMIS license must contain a license with either the **Small COP (COP)** or **Big COP** option to use COP capability.

About the Small COP (COP) and Big COP Laser Options

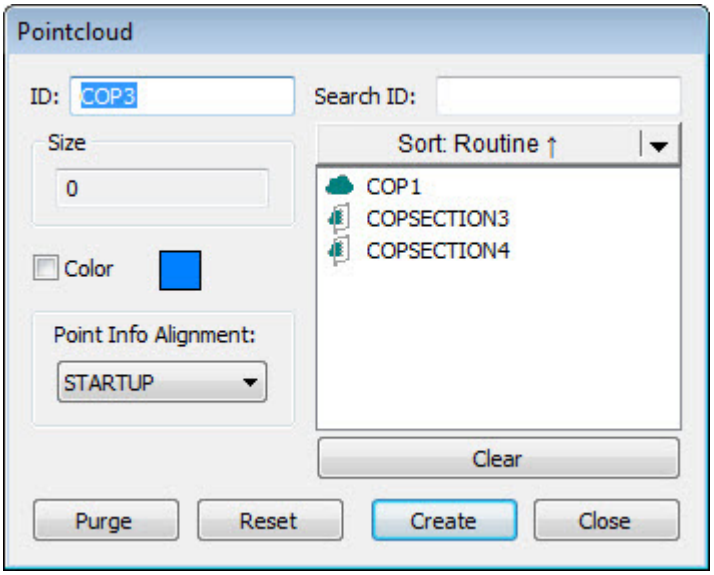
The PC-DMIS CAD++ license includes the **Small COP (COP)** option. It provides limited pointcloud functionality.

The PC-DMIS Laser option (does not include vision probes) includes the **Big COP** option. This option provides full pointcloud functionality. You can purchase it separately for other configurations.

The following list describes the differences in functionality between the **Small COP (COP)** and **Big COP** licensing options:

- If **Small COP (COP)** is enabled, and **Big COP** is disabled, PC-DMIS limits the pointcloud size to 500,000 points. The pointcloud automatically re-sizes to stay within the limit.
- Pointcloud alignment is enabled only when **Big COP** is enabled.
- Meshing is enabled only if both **Big COP** and **Mesh** are enabled.
- If the **Small COP (COP)** and **Big COP** options are disabled, pointcloud functionality is disabled.

Manipulating Pointclouds



Pointcloud dialog box

The **Pointcloud** dialog box only has an effect if the COP command contains data.

To open the **Pointcloud** dialog box, click the **Pointcloud** button () on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Feature**.

The dialog box contains the following elements:

ID - This box displays the unique identity of the pointcloud command.

Search ID - If there's a long list of operators defined, you can search using the **Search ID** box to locate specific operators in the list. When you start to enter the operator's ID into the box, the list automatically filters based on your entry.

Size - This box displays the total number of points in the pointcloud.

Color - This check box allows you to set the color for the scanned points in the pointcloud on the outside of a part. To change the pointcloud color, select the **Color** check box and then click on the **Color** box to select a color from the **Color** dialog box. For additional information on pointcloud colors, see "Pointcloud Scanning Colors".

Commands List - This area contains the list of features or scans that send data to the COP command in the dialog box. A **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine**, or **Time**. Select the option from the list and then click the **Sort** button.

Point Info - With the **Pointcloud** dialog box open, you can click on a pointcloud point in the Graphic Display window to open the **Pointcloud Point Information** dialog box. The **Pointcloud Point Information** dialog box contains information about the point with respect to the alignment. This box contains the point's numerical ID, its coordinates, and the estimated normal of the point. Corresponding CAD points also appear with CAD coordinates and CAD normal. Finally, the deviation between the point and CAD is shown with the scale for the deviation arrow specified in the dialog box. Point selection has no associated operator command. With the **Pointcloud Point Information** dialog box open, two scenarios are possible when you click the **Create Point** button:

- If there is a CAD model in the measurement routine, and the pointcloud is aligned, a **Laser Surface Point** is created, inserted, and resolved at the selected position.
- Otherwise, a **Constructed Offset** point is created and inserted in the measurement routine.

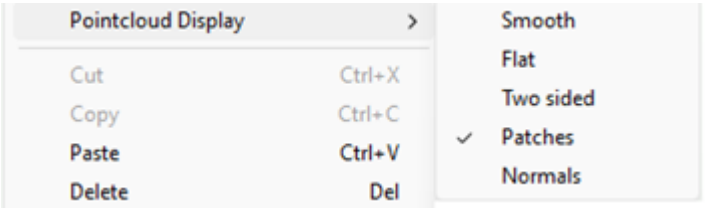
Purge / Reset - The **Reset** button restores all the data stored in a COP command. The **Purge** button permanently deletes all the data in a pointcloud that is not currently shown, selected, or filtered. This causes the pointcloud to only keep the visible data.

For information on viewing pointcloud point deviation information, see "Pointcloud Point Information".

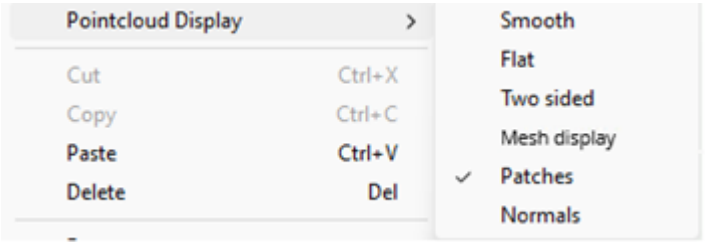
Pointcloud Graphical Representation

You can set the graphical representation of a selected pointcloud (COP). PC-DMIS stores the setting when you save the measurement routine. To do this, right-click a COP in the Edit window, or right-click the COP label in the Graphic Display window to view the **Pointcloud Display** menu.

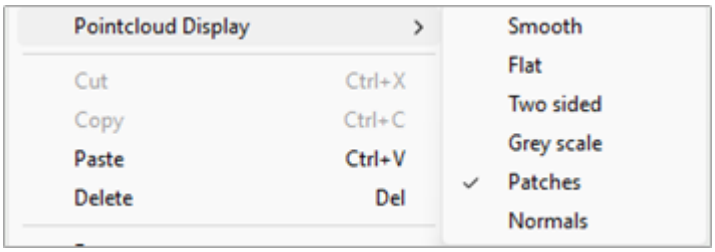
Below are examples of possible display options:



The Pointcloud Display menu for pointcloud data without intensity (grey scale) values



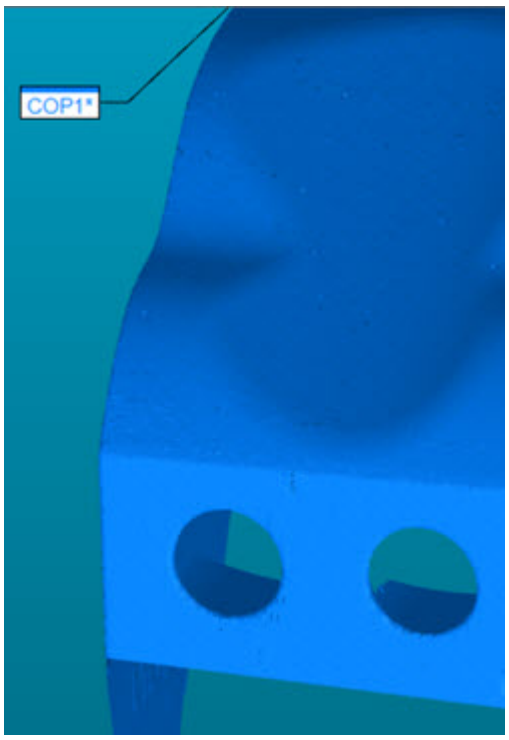
The Pointcloud Display menu for pointcloud data without intensity (grey scale) values and the COP scanned with Mesh display on



The Pointcloud Display menu for pointcloud data with intensity (grey scale) values

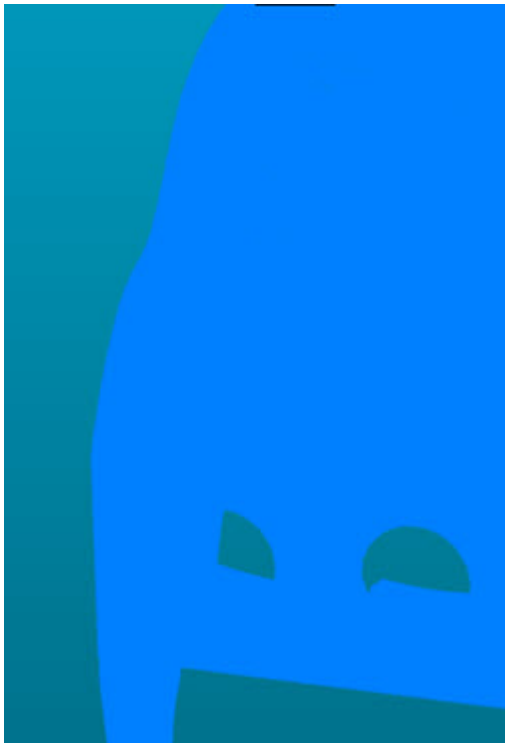
The **Pointcloud Display** options are:

Smooth - This option provides a shaded appearance using the defined COP color.



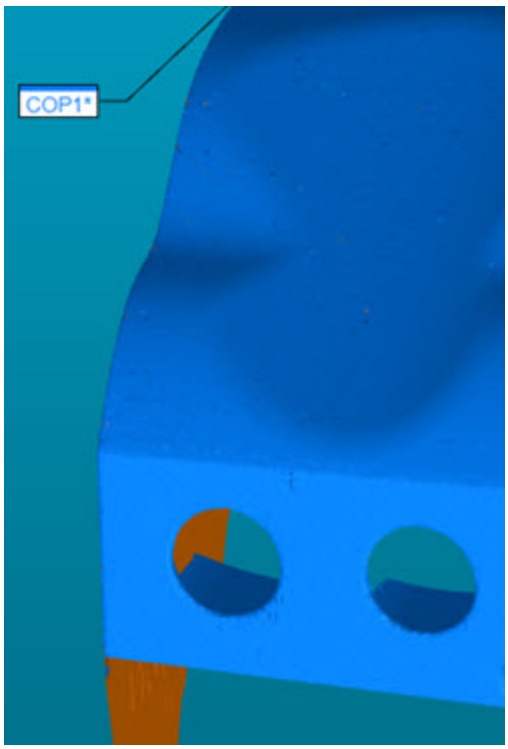
Example of the Pointcloud Display set to Smooth

Flat - This option shows the COP in a non-shaded graphical view. This selection requires the least amount of graphical memory.



Example of the Pointcloud Display set to Flat

TwoSided - This option shows a shaded appearance where the scanned side of the part is the defined COP color and the non-scanned side is the contrasting color.



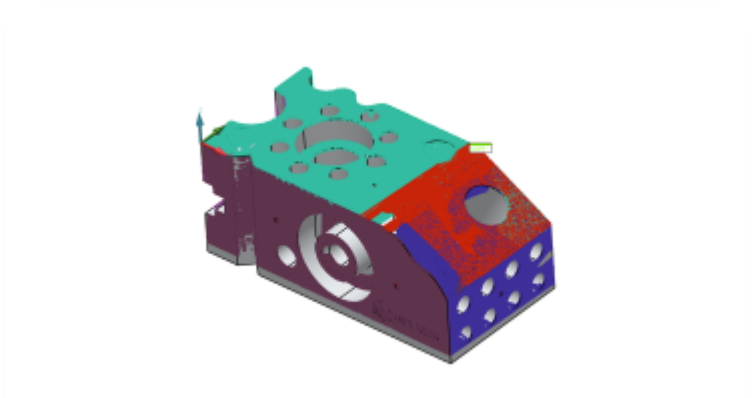
Example of the Pointcloud Display set to Two sided

Gray scale - This option is available if the data contains intensity values (for example, scanned pointcloud data using the ATS600, ATS800, AS1, or HPL10.10 scanner). This option is also available if you import a pointcloud that contains intensity values. When you select this option, PC-DMIS colors the pointcloud display in the Graphic Display window in grey scale.



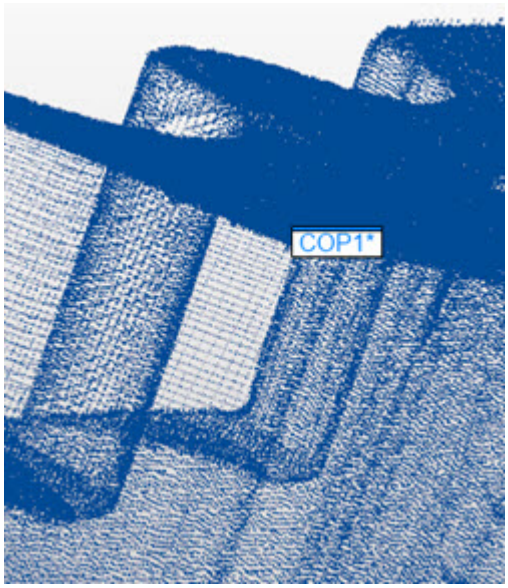
Example of the Pointcloud Display in gray scale

Patches - This option applies a different, randomly assigned color to each separate scan patch. This can be helpful for identifying areas where scan patches overlap. You can see in the image below that the red and purple patches overlap. Always try to avoid overlapping patches when gathering pointcloud data as they often lead to more noise being present. In turn, this can affect the accuracy of colormaps or extracted features in the overlapping areas.



Example of Patches option applied to the pointcloud

Normals - This option shows the normals of each point in a pointcloud.

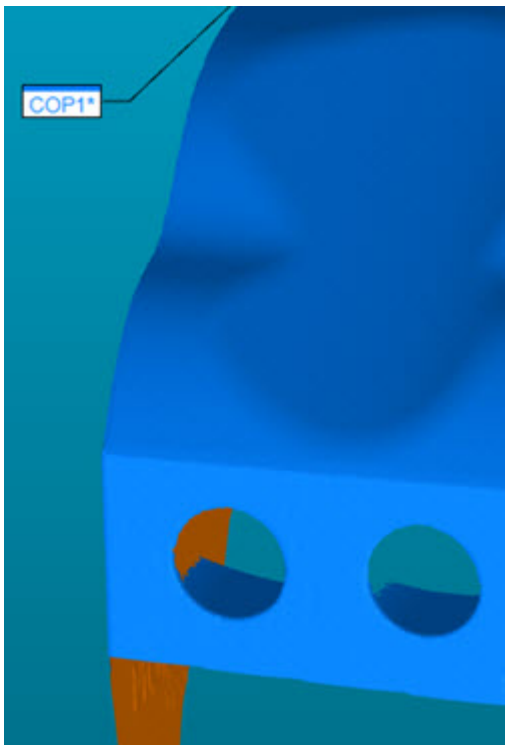


Example of the Pointcloud Display set to Normals display

You can define the length of the line segment that PC-DMIS uses to represent the normal at each point of the pointcloud in one of these ways:

- Update the value in the **Vector length** box found in the Pointcloud section in the **OpenGL** tab of the **CAD and Graphics Setup** dialog box. The value you type updates the `PointcloudNormalsLengthMM` entry in the Settings Editor application with the same value.
- From the OpenGLSettings section of the Settings Editor, update the value for the `PointcloudNormalsLengthMM` setting. The value you type updates the **Vector length** value found in the **Pointcloud** section on the **OpenGL** tab of the **CAD and Graphics Setup** dialog box. For details, see the "PointcloudNormalsLengthMM" topic in the PC-DMIS Settings Editor documentation.

Mesh display - This option allows the software to show the pointcloud as a mesh display.



Example of the Pointcloud Display set to 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 Area".

Mesh display is a *display setting only*. The underlying data is a pointcloud. This means that if you edit the COP or perform any COP operation on the pointcloud, the mesh display is lost, and the display reverts to points.

COP Command Mode Text

The COP command inside the Edit window's Command Mode looks like this:

```
COP1 =COP/DATA,SIZE=0
   REF,,
```

The COP command must precede any scan referring to it in the measurement routine. You can see in the code below that `REF,SCN2` points to the `SCN2` scan and uses its data:

```
COP2 =COP/DATA,SIZE=0
   REF,SCN2,,
```

You can have more than one scan refer to a COP command.

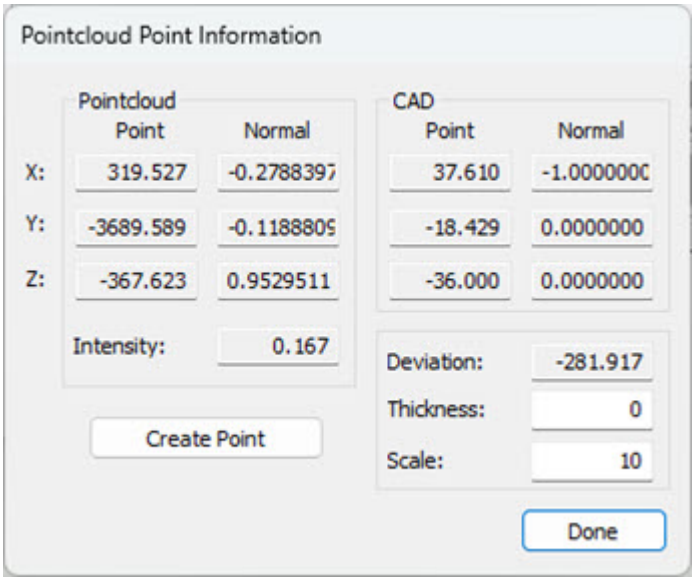
Be aware that if you cut a COP command and paste it again, the resultant command gets pasted without the data points. If you need to move your COP command to a different location in the Edit window, you need to re-create the COP command at the desired location and delete the earlier one.

Pointcloud Point Information

With the **Pointcloud Point Information** dialog box, you can view point-specific information.

To access this dialog box:

1. Click the COP command in the Edit window to select it and then press the F9 key. The **Pointcloud** dialog box for the COP command opens.
2. Click a point on the pointcloud (COP) in the Graphic Display window. This opens the **Pointcloud Point Information** dialog box.



Pointcloud Point Information Dialog Box

From this dialog box, you can view the **XYZ** and the **Normal** point vector values for the Pointcloud point, as well as the **ID** for the selected point. It also shows the corresponding CAD's **XYZ** and **Normal** vector values.

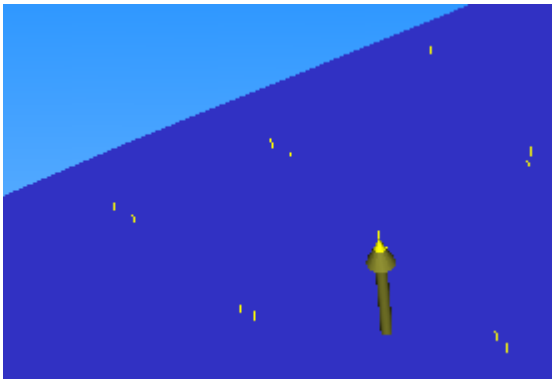
Intensity - This box shows the pointcloud intensity values if available. If there are no pointcloud intensity values, PC-DMIS hides this field. PC-DMIS currently stores intensity values when you use the ATS600 Laser Tracker (SURFACE_LEICA probe), the HPL10.10 laser sensor, or the AS1 laser sensor with a portable device running RDS version 6.6 or newer.

Deviation - This box displays the distance from the Pointcloud point to the corresponding CAD point.

Thickness - The software adds this value to the deviation from the CAD value that it computes when you click on a Pointcloud point. This value is useful, for example, if you have a CAD surface model, and you want to add a material thickness.

Scale - This value determines the scale that the deviation arrow uses in the Graphic Display window. For example, a scale of 10 displays an arrow with a length that is ten times the length of the deviation.

The deviation arrow appears when you select a point from the Graphic Display window. The arrow indicates the direction of the point deviation from the CAD.



Point Deviation Arrow

Create Point button - This button creates a Constructed Extracted Surface Point for the selected point. The software names the Constructed Extracted Surface Point with the following convention:

```
CONSTR/POINT,EXTRACTED_SURFACE_POINT
```

PC-DMIS then adds the point to the measurement routine.



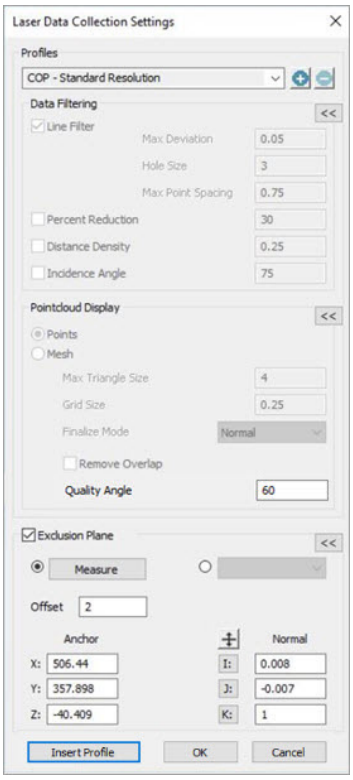
Constructed Point from Pointcloud

Using Point Data for Auto Features

With the **Auto Feature** dialog box open, you can click on desired points from the pointcloud to provide input data for a given Auto feature. For more information, see "Auto Feature Extraction".

Laser Data Collection Settings

Open the **Laser Data Collection Settings** dialog box from the **Operation | Pointcloud | Data Collection** menu option, or click the **Pointcloud Data Collection Parameters** button ( ) on the **Pointcloud** toolbar or **QuickCloud** toolbar.



Laser Data Collection Settings dialog box

The **Laser Data Collection Settings** dialog box allows you to select, define, and save scan profiles. You can also define the exclusion plane and the pointcloud display for laser scanned data.

You can click the **Collapse** button << to hide sections of the **Laser Data Collection Settings** dialog box or click the **Expand** button >> to display hidden sections of the dialog box.

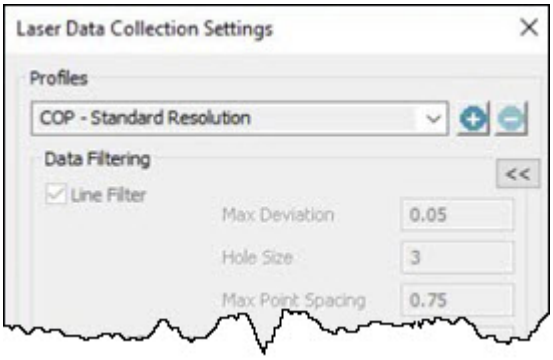
PC-DMIS supports many sensors including the AS1 and the AS1-XL sensors.

The AS1-XL sensor is capable of measuring with a much larger stand-off than other laser sensors, so it requires a different set of scanning parameters.

When PC-DMIS auto-detects that the AS1-XL sensor is the active sensor, it automatically switches the scan profile with the appropriate parameter settings required by the sensor. The profile name remains the same, but you can see that some of the parameter changes of the profile setting are visible in the **Laser Data Collection Settings** dialog box. Other changes are made in the background which are not exposed.

If you switch from an AS1-XL sensor to a different laser sensor, PC-DMIS auto-detects the change and updates the scan profile again with the applicable parameters.

Profiles Section



The **Profiles** section of the **Laser Data Collection Settings** dialog box (**Operation | Pointcloud | Data Collection**) allows you to select from a list of pre-configured scan profiles. You can also create your own scan profiles. You must use the Line Filter when scanning with the **Mesh display** option.

You can click the **Collapse** button << to hide sections of the **Laser Data Collection Settings** dialog box or click the **Expand** button >> to display hidden sections of the dialog box.

The pre-configured scan profiles that come with PC-DMIS are:

Profile Name	Description
Pointcloud (COP) Profiles	

COP - No Filters	With this profile selected, PC-DMIS clears all filters set on this dialog box and stores all points from the laser sensor in the COP.
COP - Standard Resolution	This is the default laser scan profile for new installations. As such, you cannot edit it. You can use this profile to scan parts with details of 1 mm or larger.
COP - Fine Resolution	You can use this profile to scan parts with details of 1 mm or smaller.
COP - Very Fine Resolution	You can use this profile to scan parts with details of 0.5 mm or smaller.
Mesh Display Profiles	
Mesh - Normal	The scan speed and the drawing of scanned points is good, and the mesh display has medium resolution.
Mesh - Detailed	The scan speed and the drawing of scanned points is slow, but the mesh display shows more details.
Mesh - Smooth	The scan speed and the drawing of scanned points is faster, but the mesh display resolution is low.

PC-DMIS stores the user-defined scan profiles in the "*C:\Users\<user_name>\AppData\Local/Hexagon\PC-DMIS\<version>\ScanningProfiles*" folder, where:

- <*user_name*> is the name of the user logged onto the computer who is running the PC-DMIS application.
- <*version*> is the version of the installed PC-DMIS application.

You cannot change the configurations of the pre-configured profiles. If you do make a change, PC-DMIS renames the profile to "Custom(n)", where "(n)" represents a numerical index value that updates for each new custom profile PC-DMIS creates. For example, the software names the first custom profile that you create as "Custom1", the second one that

you create as "Custom2", and so forth. You can click inside the profile name box and edit the name of any custom profile.

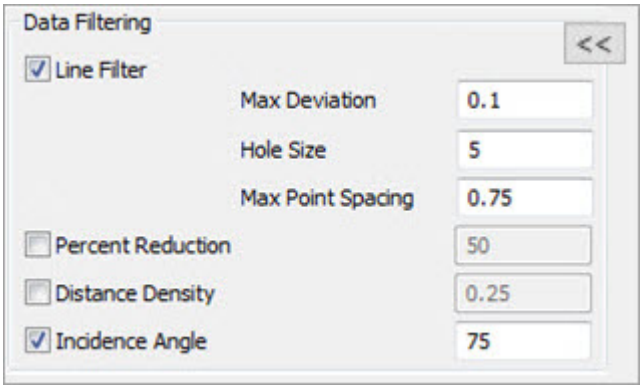
If you open a measurement routine from an earlier version of PC-DMIS, but the Data Collection settings do not match any of the existing profiles, the software automatically creates a new custom profile with those settings.

PC-DMIS uses the last-used profile for any new measurement routine.

The first time that you start PC-DMIS after you upgrade from a version earlier than 2026.2, PC-DMIS keeps the profile that you last used in the earlier version. The one exception is COP - No Filters. If that was your last-used profile, PC-DMIS selects COP - Standard Resolution instead, so that you get the same default as a new installation.

You can click the **Add** button to make a copy of the current profile. You can then rename it and make any changes to the profile's configuration. Click the **Delete** button to delete the current profile.

Data Filtering Area



Data filtering allows real-time filtering of the data. It removes the data as you scan.

You can click the **Collapse** button << to hide sections of the **Laser Data Collection Settings** dialog box or click the **Expand** button >> to display hidden sections of the dialog box.

The **Data Filtering** section provides these options:

Line Filter - Select this check box to enable real-time filtering for individual lines. It provides smoothing and point reduction of incoming data from the laser sensor.

Mark the **Line Filter** check box to enable these options:

> **Max Deviation** - As the software evaluates each incoming scan line, the software can move or smooth points in relation to their neighboring points. This setting defines the maximum allowed value that the software can move or smooth a point.

Hole Size - This setting defines the minimum hole or gap size during a scan. When PC-DMIS evaluates a scan line and detects a hole or gap of this size (or larger), the filter treats the scan segments as separate lines. In most cases, you can set the **Hole Size** value to the size of the smallest hole on the physical part.

Max Point Spacing - This setting defines the maximum distance between two consecutive points that the software uses when it analyzes the incoming scan data to reduce the number of points. If the scan surface is curved, the resulting point spacing is typically smaller than the **Max Point Spacing** value.

When this parameter is set to zero, no point reduction takes place. Typically, you should set this value to less than 1/3 of the hole size.

The **Max Point Spacing** setting determines the resolution of the scanned points. For most parts, you can use the default values from the table below. To obtain higher resolution when scanning parts with small details, you can use a smaller **Max Point Spacing**. A smaller **Max Point Spacing** results in fewer scanned points being filtered and increases the total COP size.

	Max Point Spacing
Large Details	1 mm / 0.03937 inch
Default	**0.75 mm / 0.02953 inch**
Small Details	0.5 mm / 0.01968 inch
Fine Details	0.25 mm / 0.00984 inch

Percent Reduction - This option removes a percentage of the pointcloud data collected.

1. Select the **Percent Reduction** option, and in the box to its right, type a percentage value between 0-100 inclusive. The value is the percent of the collected pointcloud data that you want the software to filter out. If you enter zero, no filtering takes place.

2. Click **OK** to apply this to your measurement routine.

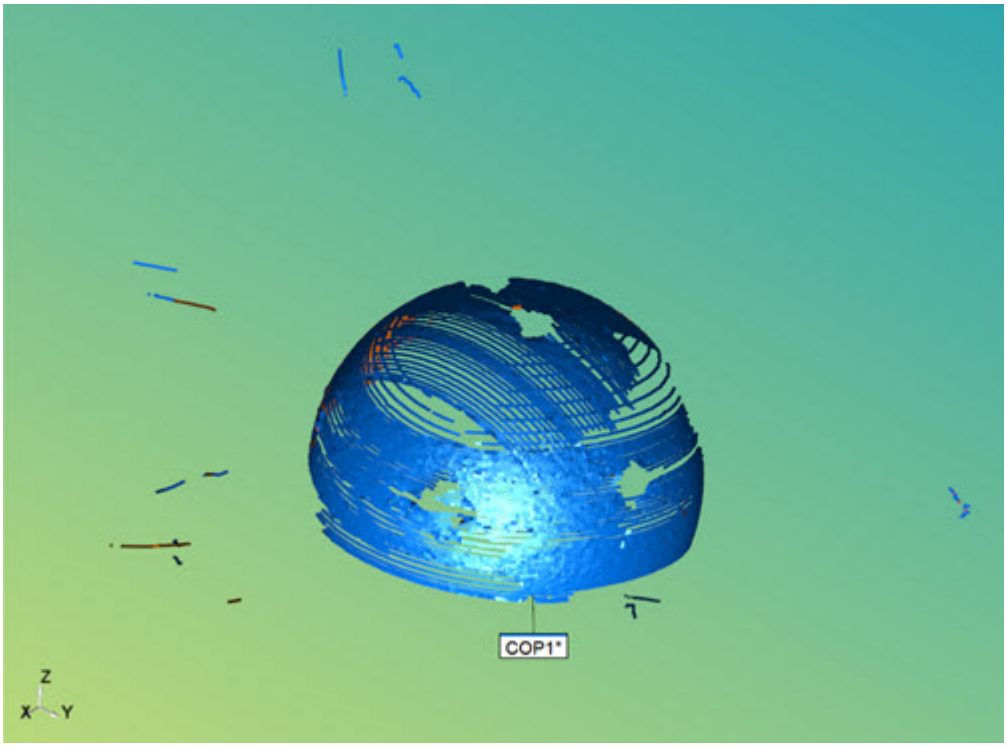
Distance Density - This option filters the data based on the point distance value. If the distance between a point and its neighboring points is less than this value, the software discards the point. This option becomes available if you select the **Points** option in the **Pointcloud Display** section of the dialog box.

1. Select the **Distance Density** option and in the box to its right, type a distance value in the measurement routine's units. Values that are greater than or equal to zero are

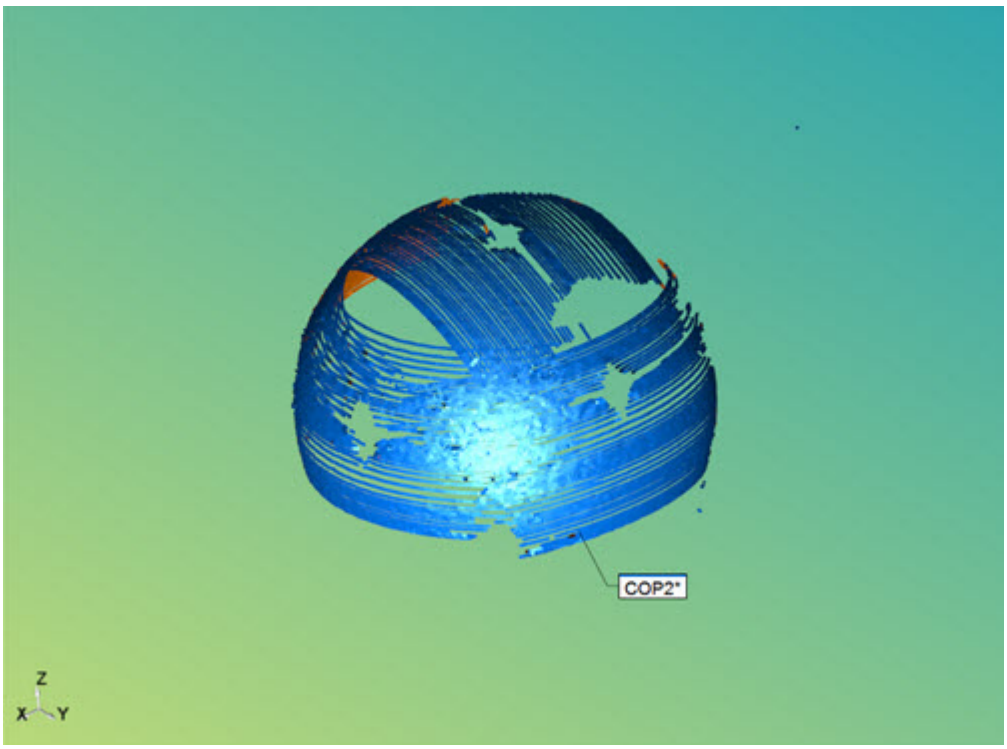
valid. 1 mm is the default value. If your measurement routine uses inches, the software converts 1 mm to inches.

2. Click **OK** to apply the filtering.

Incidence Angle - This option filters out all scanned points that have an incident angle greater than the entered value. The **Incidence Angle** check box is marked by default with a default value of 75. The angle is calculated between the estimated surface normal and the scan direction of the laser sensor. The smaller the value, the more points are filtered out.



Shiny sphere with no incidence angle applied

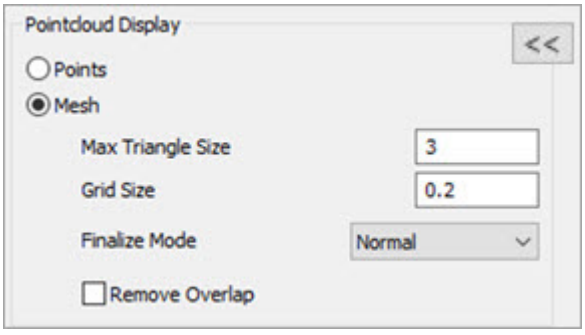


Shiny sphere with incidence angle on at default value 75

You can apply the **Incidence Angle** filter in real-time while scanning. During the scan process, the software determines the angle of the scan line that is relative to the measured surface. The software then automatically removes and discards any points that are greater than the specified angle.

The **Incidence Angle** option is not available if you are using the HP-L-10.10 laser sensor.

Pointcloud Display Area



The **Pointcloud Display** section allows you to select the display setting during a scan. You can choose to show the pointcloud as points or as a mesh display. If you select the **Mesh** option for scanning, it may allow you to easily see areas which need more data coverage.

You can click the **Collapse** button << to hide sections of the **Laser Data Collection Settings** dialog box or click the **Expand** button >> to display hidden sections of the dialog box.

> The **Mesh** option is only available for Portable systems which have the Mesh license.

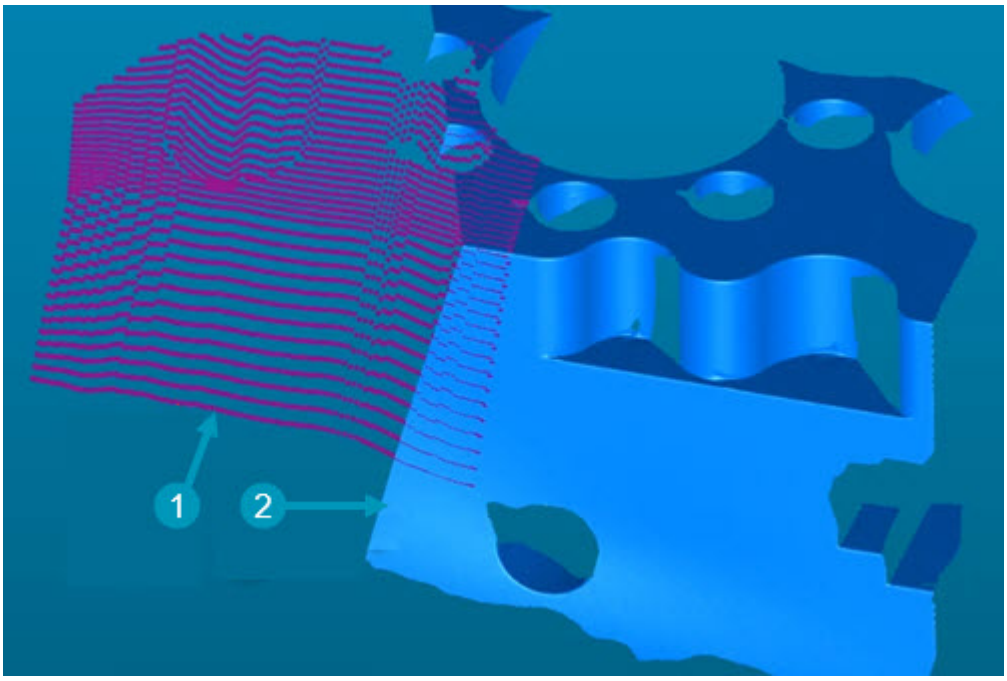
Points - This option displays the pointcloud as a set of points.

Mesh - This option shows the laser pointcloud data as a mesh and is only available for Portable systems. You must use the Line Filter when scanning with the **Mesh** display.

During the scan, PC-DMIS shows the active scan pass as a pointcloud. When the software completes the scan pass, it displays the scan as a mesh. The mesh display is a temporary graphical rendering only.

If you modify the pointcloud (for example, if you perform a Select, Clean or Filter operation), or if you close and then reopen the measurement routine, the mesh display is lost and PC-DMIS displays the data as a pointcloud.

After a scan with the **Mesh** display option, you can choose to keep only the pointcloud, or you can create a mesh data object. If you choose to create a mesh data object, the software also keeps the original pointcloud.



Example showing the active (1) and prior (2) scan passes

The mesh display is relative to the orientation of the laser sensor. While scanning, if the laser sensor orientation changes more than 25 degrees in a single scan pass, the software meshes the collected data and automatically creates a new scan.

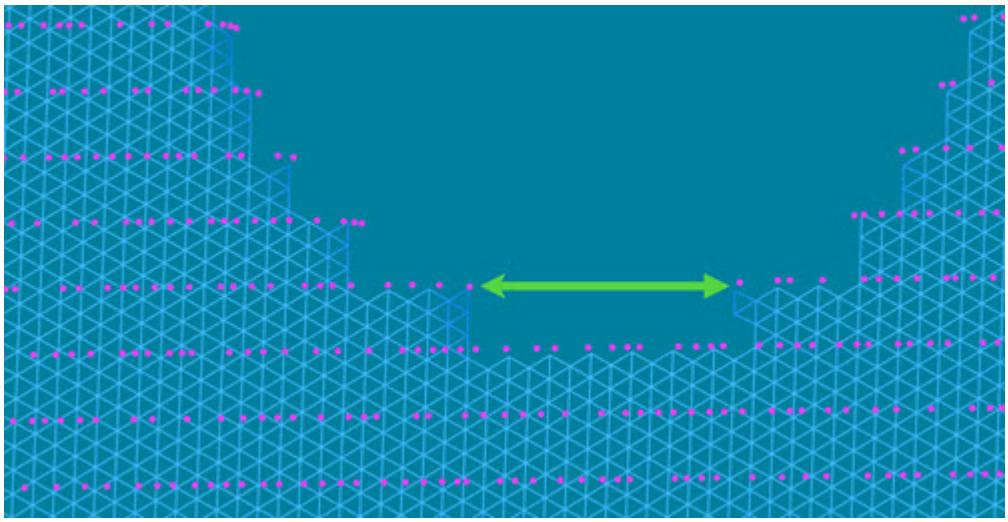
The **Max Triangle Size** and **Grid Size** values define the settings for the displayed mesh while scanning.

If you modify the pointcloud (for example, if you perform a Select, Clean or Filter operation), or if you close and then reopen the measurement routine, the mesh display is lost and PC-DMIS displays the data as a pointcloud.

- If the scan speed is slow, and more than one point is in a grid square, PC-DMIS keeps the best point.
- If the scan speed is fast, it is possible to have a grid square without any data. This may cause gaps in the displayed mesh.

  Max Triangle Size - The software uses this value to recognize holes or gaps in the pointcloud data. If the distance between any two points is greater than this value, the software does not create any triangles in that area. If there are hole features on your part, you typically need to set this value to be slightly smaller than the smallest hole. This prevents the mesh display from filling the hole (see the image below).

  The default value for the **Max Triangle Size** is 5 mm. The software converts this to inches if your measurement routine is using that unit. Valid range values depend on the size of the part.



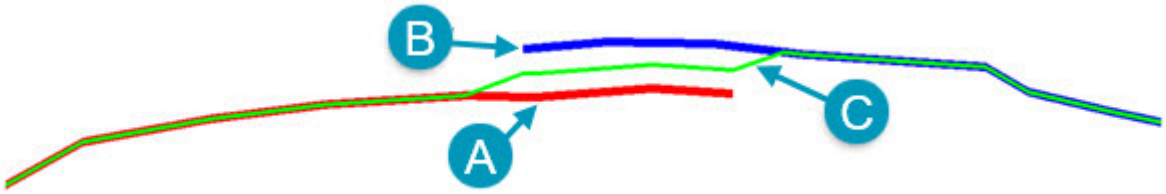
This example shows the distance between the two points is greater than the Max Triangle Size value. PC-DMIS does not create any triangles in this area.
Blue triangles = Mesh Display. The size of the blue triangles is determined by the Grid Size value.
Purple points = Scanned points.

Grid Size - This value defines the size of each triangle in the mesh display grid. This value also affects the display resolution and how refined the mesh appears. When you use a small value, it takes more time to generate the mesh display while scanning, but results in a higher resolution. Be aware that this value is critical; a small value can negatively affect the data collection speed.

Finalize Mode - When you create the mesh with the **Create Grid Mesh** button on the **Portable Scanning Widget** toolbar (or the **Grid Mesh** option on the **Mesh** dialog box), the software reduces and smooths the mesh display, and removes the overlap. The **Finalize Mode** option defines the amount of smoothing to apply. The options are:

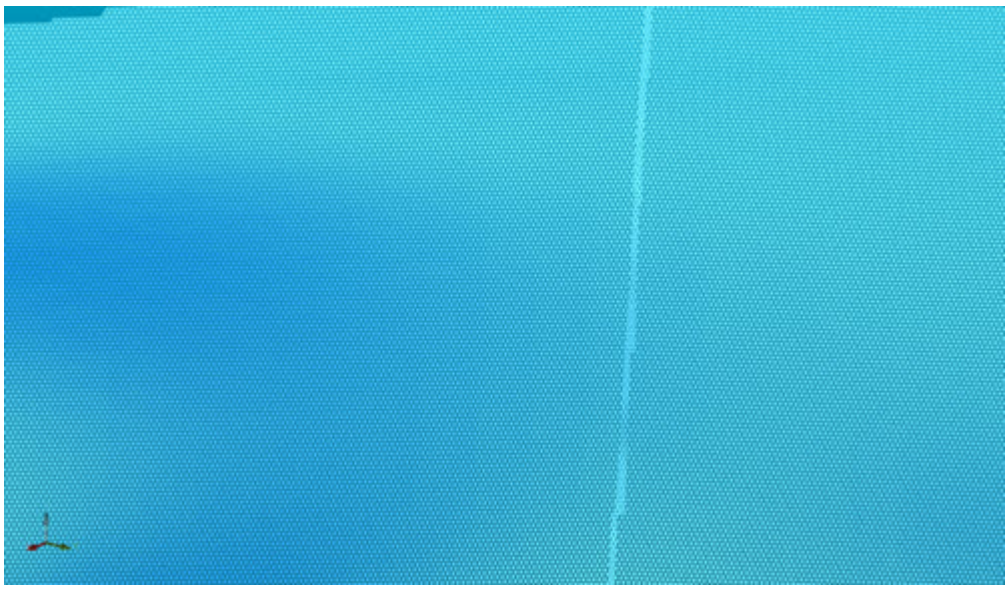
- **Precise** (least amount of smoothing)
- **Normal**
- **Smooth** (most amount of smoothing)

Remove Overlap check box - When you select this check box, PC-DMIS averages the overlapping areas of multiple scan passes and then blends them together in real-time while the scan takes place. This results in the software removing overlapping data from the mesh display. Note that the pointcloud (COP) object contains all the original scanned points. If the graphical mesh rendering is too slow, you may want to disable this function.



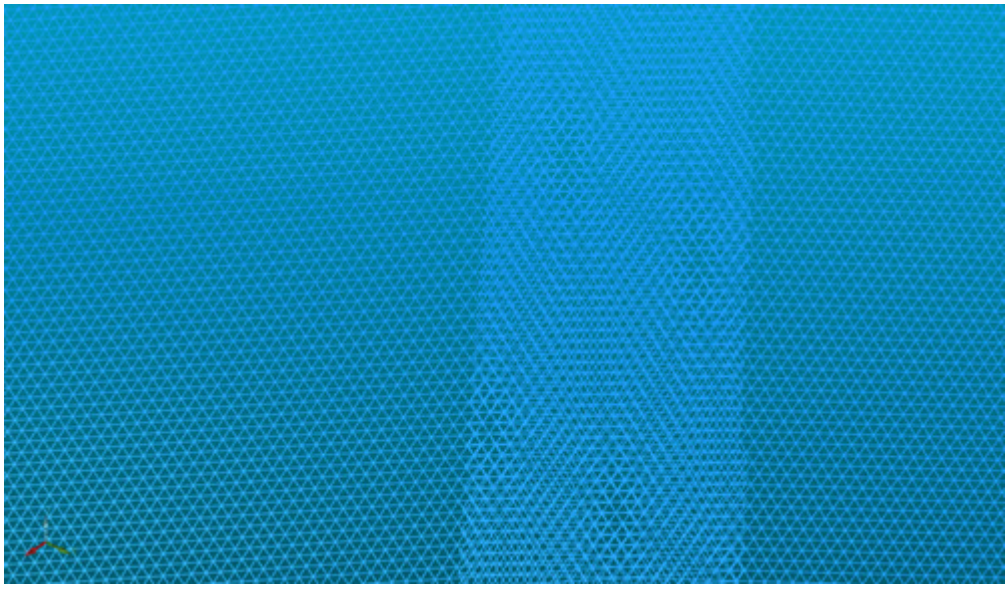
(A) - Scan Pass 1
(B) - Scan Pass 2
(C) - Stitched area

The overlapping scan passes must be within a distance lower than the point density in order to be stitched.



Example of a scan as a Mesh display with the Remove Overlap option selected

During a scan with the **Mesh** option, if you clear the **Stitch Patches** check box, the software overlays multiple scan passes on top of each other.



Example of a scan as a Mesh display with the Remove Overlap option NOT selected

Quality Angle - When you select the **Mesh** option from the **Pointcloud Display** area and you perform a laser scan, PC-DMIS displays scanned triangles with an angle greater than the **Quality Angle** setting in the Graphic Display window. The software shows triangles that were scanned with a good sensor-to-surface orientation in green. Triangles outside the **Quality Angle** value are in red.

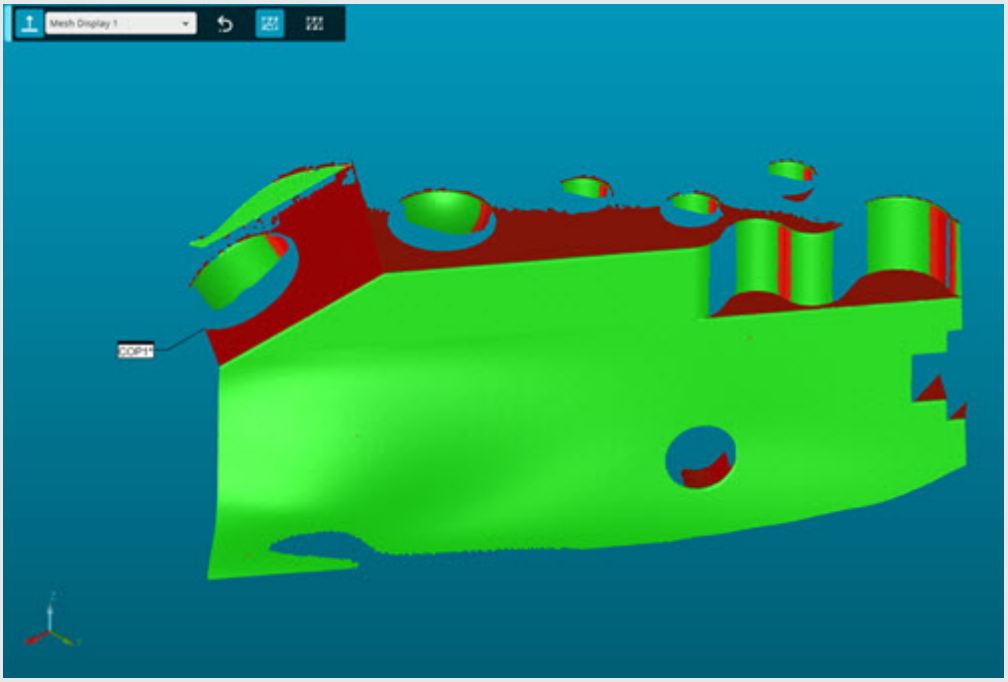
You can re-scan the areas with the scan line more perpendicular to the part surface to obtain better quality triangles.

If you select the **Remove Overlap** option and you re-scan the low-quality areas with a better scanner-to-surface orientation, PC-DMIS may replace the red triangles with the new scanned data.

Suppose you want to enable the display of the low-quality triangles. To do this, from the **Portable Scanning Widget** toolbar, select the **Low Quality Triangles On/Off** button ( ).



Use this button to turn the display of the red and green triangles on and off.



Example showing the display of the red and green triangles when you select the Low Quality Triangles button

For details on the **Portable Scanning Widget** toolbar, see the "Portable Scanning Widget Toolbar" topic in the PC-DMIS Portable documentation.

If you perform any Pointcloud operation, or if you close and reopen the measurement routine, you lose the Low Quality Triangle red and green display.

Example Workflow: Scan as Mesh Display

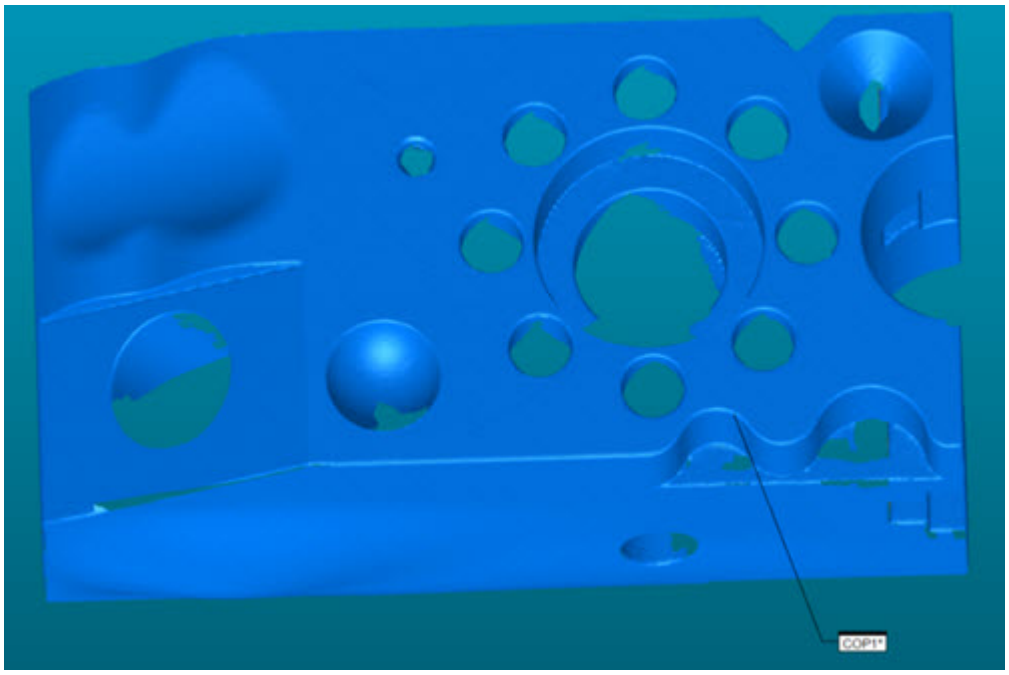
1. Select a mesh profile from the **Portable Scanning Widget**.

   You can also create your own custom mesh profile. For details, see the "Profiles Section" in this documentation.

2. Scan the part. PC-DMIS displays the COP as a Mesh, but the data is a Pointcloud.

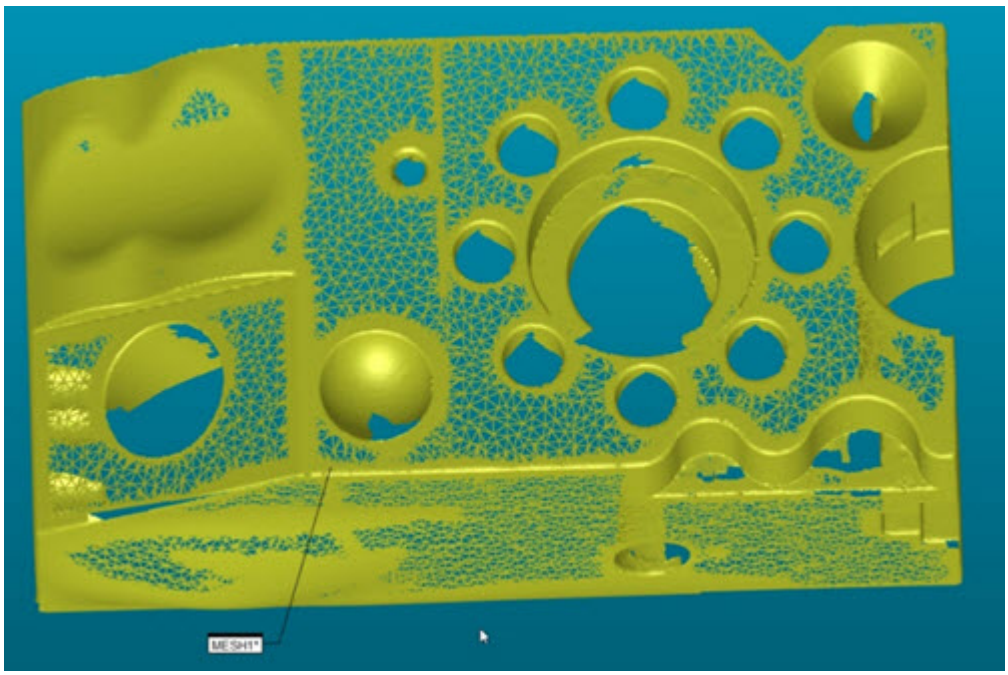
   > The Mesh display is a temporary graphical rendering. For details on graphical representation of the pointcloud, see "Pointcloud Graphical Representation" in this documentation.

   Example of the pointcloud displayed as a Mesh

   > The next step to create the Mesh object is optional. If you stop at step 2, all the scanned data is a pointcloud.

3. OPTIONAL: Create the Mesh. PC-DMIS uses the **Max Triangle Size**, **Grid Size**, and **Finalize Mode** options to reduce, smooth, and, if you select the **Remove Overlap** check box, remove the overlap. It then computes the final Mesh object.



Your measurement routine contains both the original scanned pointcloud (COP) and the Mesh data objects.

Exclusion Plane Section



You can use exclusion planes to remove all points within the defined area of the plane. To enable this feature, select the **Exclusion Plane** check box.

You can click the **Collapse** button << to hide sections of the **Laser Data Collection Settings** dialog box or click the **Expand** button >> to display hidden sections of the dialog box.

When you select the **Exclusion Plane** check box, the software activates the **Pointcloud Data Collection Parameters** button ( ) for the defined exclusion plane. If the button on the toolbar is in the active state, filtering is enabled. Once activated, the software uses the exclusion plane the next time you execute the measurement routine.

You can tell when the exclusion plane is active in your measurement routine by how the **Pointcloud Data Collection Parameters** button appears on the **QuickCloud** or **Pointcloud** toolbar. If the button appears to be pressed in, the exclusion plane is active; otherwise, it is not active.

There are three ways to define the exclusion plane:

1. Measure

Use a contact probe or laser sensor to measure the exclusion plane.

Click the **Measure** button and then take three hits with a contact probe to measure the exclusion plane. With a laser sensor, scan the area of the plane. If an alignment already exists, the plane is automatically defined in that alignment. If not, the plane is defined using the machine coordinates. If that changes, you need to redefine the plane.

2. Entering the XYZ and IJK values

You can also define the exclusion plane by its normal vector and an anchor point. The exclusion plane is independent of the data filtering.

To define an exclusion plane:

1. Edit the XYZ anchor positions if necessary.
2. Click the **I**, **J**, or **K** normal button that your plane is relative to and edit the value if necessary. To automatically change the direction of the normal value, you can click the **Reverse Direction** button  .
3. If PC-DMIS is in Online mode, you can click the **Measure** button to measure your defined exclusion plane.
4. Click **OK** to save your settings.

3. Select an existing plane

Select an existing plane (a plane that already exists in the measurement routine) from the **Exclusion Plane Feature** list. The Anchor and Normal (vector) fields update accordingly.

By selecting an existing plane, when the measurement routine is re-executed and the plane is re-measured, this becomes the new exclusion plane used for the COP. This is useful for portable devices if the device is moved or if the part is moved to a different surface.

Offset - This offsets the plane in the defined Normal direction by the value entered (in measurement routine units).

Using the Simulate Pointcloud Function

The **Simulate Pointcloud** function allows you to create and view the pointcloud from the **Scan** dialog box (Linear, Freeform, and so on) when the CMM is in Offline mode.

Using the laser probe orientation, field of view, and scan settings, the software projects the laser lines onto the CAD model. This way you can see if the simulated pointcloud is acceptable and make changes if needed for an individual scan. PC-DMIS holds the simulated points in a COP.

To control the speed of the simulated laser scan, you can adjust the settings found on the **Animation** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**). For details, see "Using the Animation Parameters for Pointcloud Simulation".

To define the active sensor tip and scan speed, follow the "Getting Started" chapter. If you like, you can pre-define the laser width and density of the scan from the **Measure Laser Probe** dialog box when you define the sensor. To access this dialog box, open the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), and then click **Measure**. For details on laser probe measurement options, see "Measure Laser Probe Options".

Define the scan path properties from any **Scan** dialog box (linear, freeform, and other properties). You can also define the laser width and density settings from the same dialog box. For details, see "Scan Zoom States (for HP-L Sensors)".

To display the simulated pointcloud in the Graphic Display window, click the **Simulate** button from any **Scan** dialog box. You can also simulate the pointcloud when you execute the scan from the Edit window in Offline mode.

After you create the scans, you can execute the entire offline measurement routine and display all scans in different probe orientations. This allows you to check if you can extract scanned Auto features (for example) based on the scan settings.

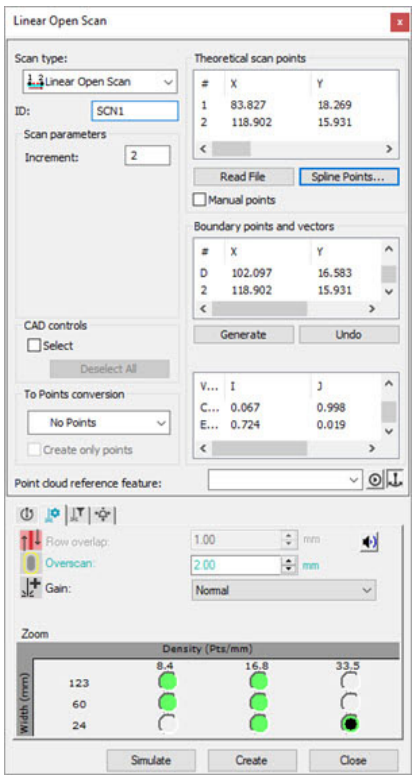


Warning: If the CMM is online and you press the **Simulate** button on the **Laser Scan** dialog box (Freeform, Linear Open, etc.), the software immediately drives the machine and scans online. To prevent injury, make sure you are clear of the machine before you press the **Simulate** button.

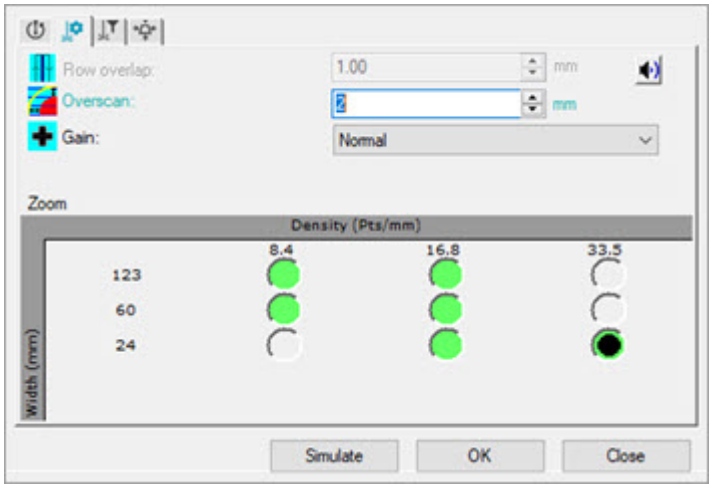
Example Using the Simulate Pointcloud Function

For example, to use the Simulate pointcloud function on a linear open scan:

1. Create a COP (**Insert | Pointcloud | Feature**). For details on pointcloud features and creating a COP, see the "Using Pointclouds" chapter.
2. Set the scan speed. For details, see "Getting Started".
3. Open the **Linear Open Scan** dialog box (**Insert | Scan | Linear Open**).

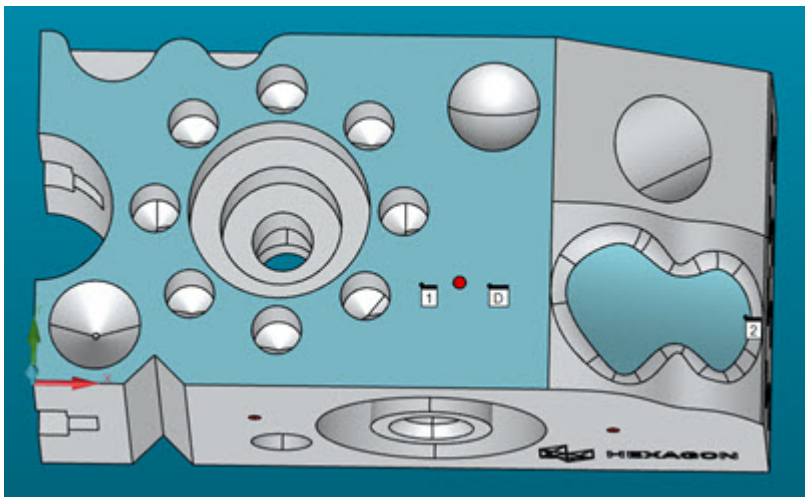


4. In the **Scan parameters** section, set the **Increment** value.
5. At the bottom of the dialog box, click the **Laser Scan Properties** tab, and set these options:
    - Type the **Overscan** value.
    - Select the **Gain** option from the list.
    - Select the stripe **Width** and scan **Density** setting.



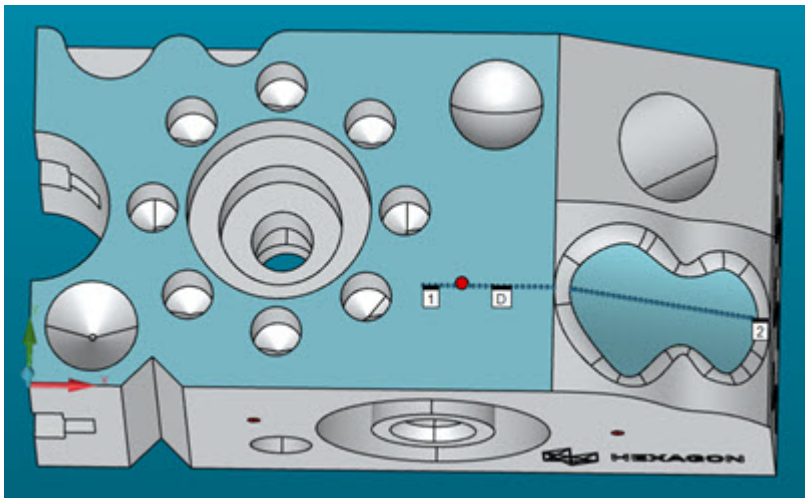
Laser Scan Properties tab

6. In the Graphic Display window, click the three points on the CAD model to define the boundary points and vectors.



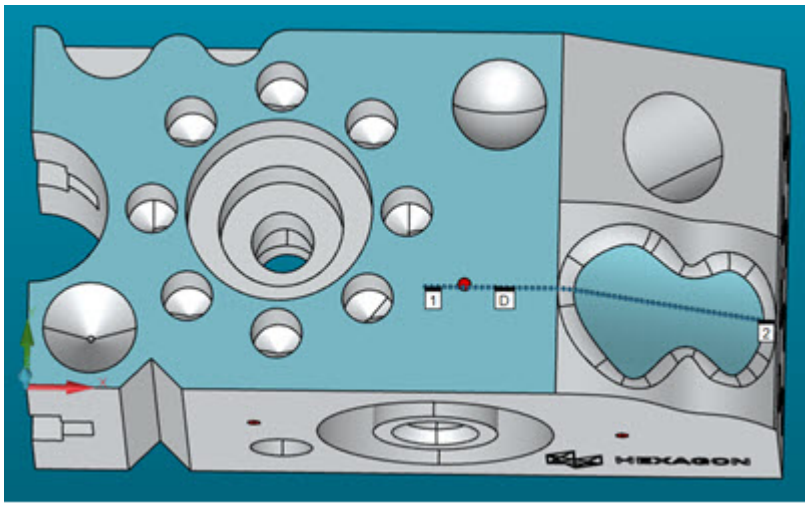
Example showing the three points to set up the scan

7. From the **Boundary points and vectors** section, click **Generate**.



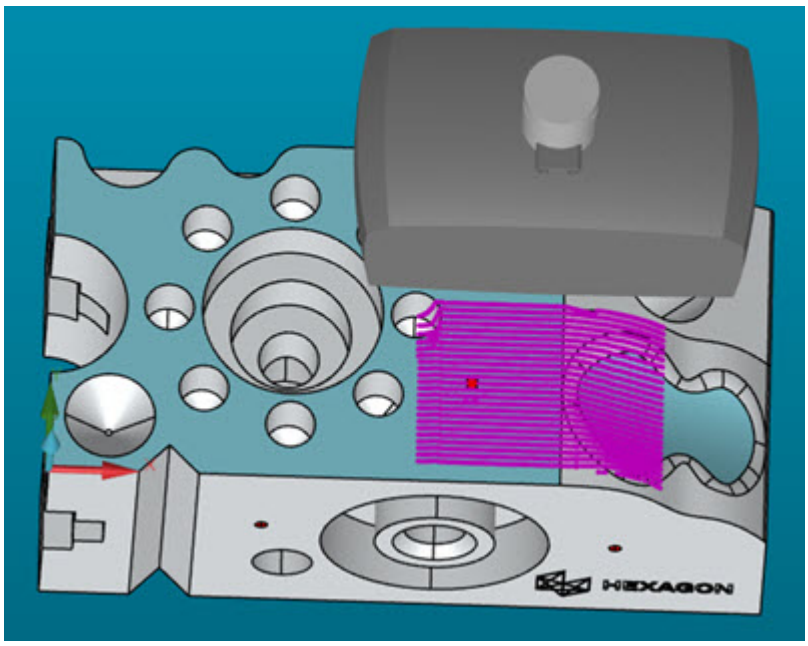
Example showing a linear open scan generated

8. From the **Theoretical Scan Points** section, click **Spline Points**.

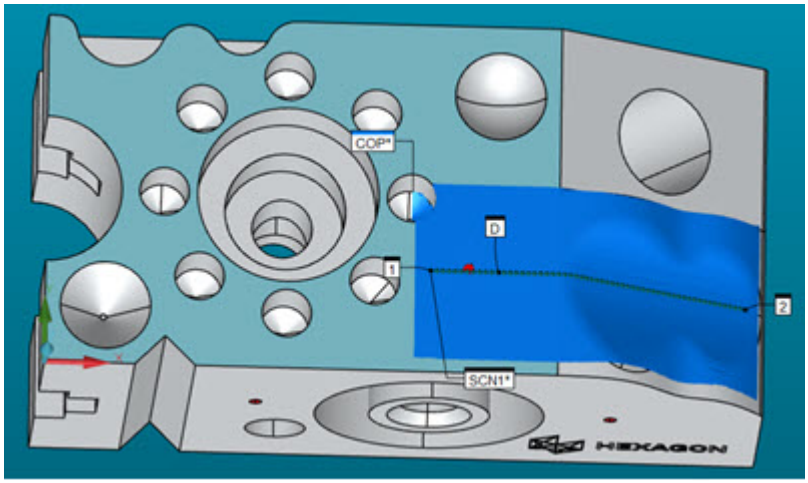


Example showing a linear open scan splined

9. To show the simulated pointcloud based on the current probe orientation (active tip) and laser scan settings, click the **Simulate** button.



Example showing the pointcloud simulation in progress



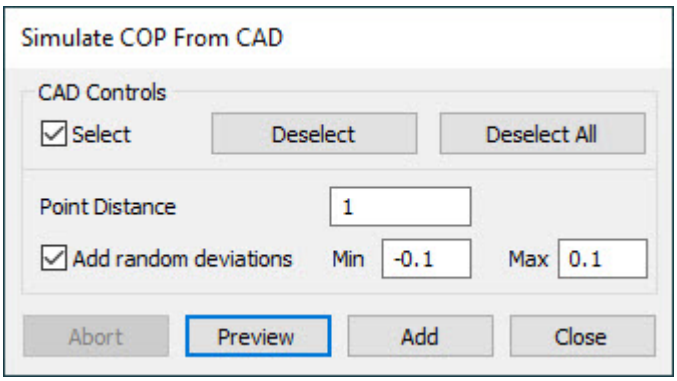
Example showing the pointcloud simulation completed

If needed, you can make changes to the scan and simulate it to check the results.

10. When everything looks correct, click the **Create** button to implement the scan in your measurement routine.

Simulate COP from CAD

You can use the **Simulate COP from CAD** dialog box to create a theoretical pointcloud on your CAD model by simply selecting surfaces. You must have a CAD model imported into your measurement routine.



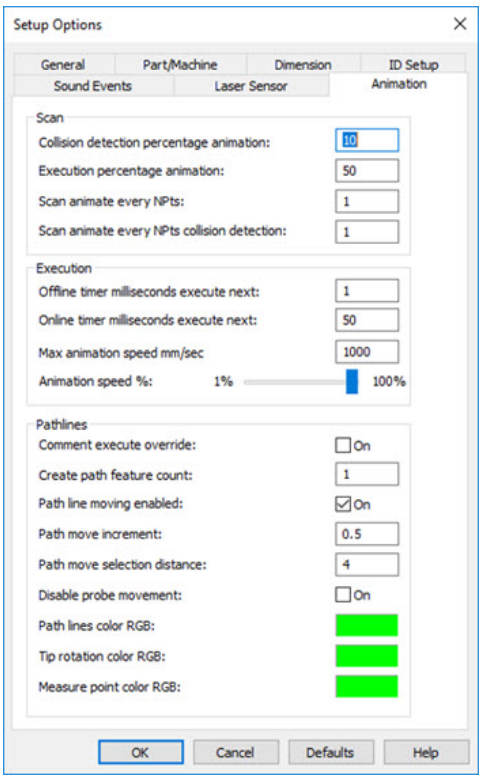
Simulate COP from CAD dialog box

To do this:

1. Click the **Simulate COP from CAD** button on the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**).
2. Click the **Select** check box from the **CAD Controls** area of the dialog box. This allows you to select the surfaces that you want to generate the theoretical pointcloud from.
3. Type a value for the **Point Distance**. This value defines the minimum distance between neighboring points. PC-DMIS uses this value to create the pointcloud.
4. You can add in random deviation errors to make your theoretical pointcloud more realistic. To do this, select the **Add random deviations** check box and type in the **Min** and **Max** deviation values.
5. Click **Preview** for an initial view of the theoretical pointcloud. Click **Add** to create the theoretical pointcloud or click **Close** to close the dialog box without creating the pointcloud.

Using the Animation Parameters for Pointcloud Simulation

You can control the speed of the simulated laser scan in the **Scan** and **Execution** areas on the **Animation** tab of the **Setup Options** dialog box (**Edit | Preferences | Setup**, or press the F5 key). For details, see "Setup Options: Animation Tab" in the PC-DMIS Core documentation.



Setup Options - Animation tab

Scan Area

Scan animate every NPts - This value determines the number of scan path points that PC-DMIS uses for the animation.

- For pointcloud simulation, if you type a value of 1, the software uses every scan point. This results in a smoother animation.
- If you use a larger value (for example 10) for pointcloud simulation, the laser scanner probe moves from point 1 to point 10 and immediately shows all the purple pointcloud stripes between those scan path points. The result is a faster but less-smooth animation.

The default value is 50.

You can also set this value in the PC-DMIS Settings Editor. For details, see "ScanAnimateEveryNPts" in the PC-DMIS Settings Editor documentation.

Execution Area

For Pointcloud simulation, the values in this area are typically set to the maximum values.

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 where the animation renders too slowly. To increase the duration between redrawn views of the animation, increase this value. This causes the software to draw fewer animation steps.



You can also set this value in the PC-DMIS Settings Editor. For details, see "MaxAnimationSpeed" in the PC-DMIS Settings Editor documentation.

Animation speed % - The slider allows you to adjust the actual percentage of the **Max animation speed mm/sec** value PC-DMIS uses.

Pointcloud Operators

The Pointcloud operator commands listed below perform different operations on pointcloud (COP) commands and other pointcloud operator commands. The software defines the units for these commands by the measurement routine.



Versions earlier than PC-DMIS 2014 used a COPOPER keyword prior to the operator command. This COPOPER command is no longer available, and the commands now use a COP prefix. For example, the Filter operator is now `COPFILTER` .

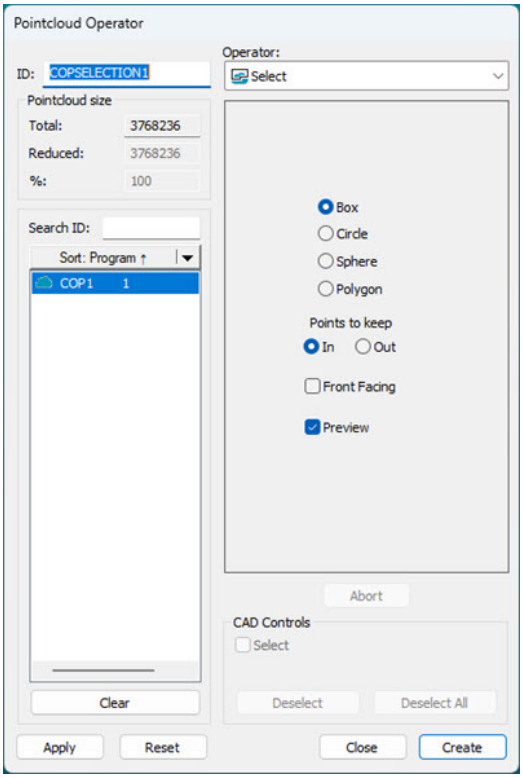
You can add pointcloud operator commands into your measurement routine in any of these ways:

- Select the **Insert | Pointcloud | Operator** menu item.
- Select the **Insert | Pointcloud** submenu and then select either **Cross Section**, **Surface Colormap**, **Point Colormap**, or **Thickness Colormap**. With these options, you can add basic pointcloud commands that alter the display of pointclouds in the Graphic Display window.
- **Operation | Pointcloud** - This submenu allows you to alter the number of points that PC-DMIS includes in COP commands. Items included in this submenu are: **Clean**, **Empty**, **Filter**, **Purge**, **Reset**, and **Select**.
- Manually type pointcloud operator commands into the Edit window. If the cursor is on the command in the Edit window, and you press **F9**, the **Pointcloud Operator** dialog box opens.
- From the **Pointcloud** toolbar, click the appropriate **Pointcloud Operator** button to open the associated **Pointcloud Operator** dialog box. The software applies the pointcloud operator to the COP.



You must be licensed with the **COP** option to use pointcloud operator commands.

Manipulating Pointcloud Operators



Pointcloud Operator dialog box

The **Pointcloud Operator** dialog box is displayed by selecting **Insert | Pointcloud | Operator** from the main menu. The dialog box contains the following elements:

ID - Contains a unique identity of the pointcloud operator command being edited.

Pointcloud size - This area contains the **Total** size of the pointcloud operator selected in the list box. The **Reduced** size and the percentage (**%**) of reduction in size are also shown.

Command List - The list of commands on the left shows the COP or pointcloud operator commands that send data to the pointcloud operator command in the **ID** box. The Command List section also has these two functions:

Search ID - If there's a long list of operators defined, you can search using the **Search ID** box to locate specific operators in the list. When you start to enter the operator's ID into the box, the list automatically filters based on your entry.

Sort - The **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine**, or **Time**. Select the option from the list and then click the **Sort** button.

Apply - Applies the operator to the COP or pointcloud operator commands selected.

Reset - Restores all the data stored in a COP command.

CAD Controls - Lets you apply the operation to selected CAD elements. See the "CAD Controls" topic, which describes scanning in more detail.

Operator - This list shows the operator commands that you can select and apply to pointcloud or other operator commands. Depending on the type of operator selected, different options become available in the dialog box. Refer to the following operator types for details:

Pointcloud Thickness Colormap

The Thickness Colormap allows you to show and measure the part thickness as a colormap using only the Mesh or Pointcloud (COP) data object. You can also compare the measured thickness to the nominal CAD model thickness.

The measured data object must have data on two opposing sides which have the opposite normal orientations to use this function.

To create a Pointcloud Thickness Colormap, click the **Pointcloud Thickness Colormap** button  from the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**) to open the **Pointcloud Operator** dialog box. You can also do this from the **Thickness Colormap** menu option (**Insert | Pointcloud | Thickness Colormap**).

When using a Pointcloud, the data must have XYZIJK values or stripe information. For details, see "Example Pointcloud File Formats for Thickness Colormaps" in this documentation.

When you perform a thickness colormap of the data object (Pointcloud or Mesh), PC-DMIS calculates the measured thickness up to a **Max Thickness** value. The software does not evaluate any data value greater than the **Max Thickness**.

When the measured data is aligned to a CAD model, you can choose to create a thickness colormap which shows the deviation of the measured thickness in comparison to the nominal CAD model thickness.

If you use a large **Max Thickness** value on a large data object, it can result in a longer processing time.

You can create the following thickness colormaps:

- Measure part thickness using either a Pointcloud or Mesh data object

- Create Thickness Colormap on specific CAD surfaces when you mark the **Select** check box in the **CAD Control** area of the dialog box and then select the CAD surfaces in the Graphic Display window.
- Compare to CAD thickness colormap, which shows the deviation of the Pointcloud or Mesh data object thickness in comparison to a CAD model.

Show / Hide Colormaps

You can show or hide colormaps in the Graphic Display window in different ways. When hidden, PC-DMIS does not show the colormaps in the Graphic Display window as you move through the Edit window.

The **Activate Colormap** button has two states: Enabled and Disabled. From the **Graphics Items** toolbar or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**), click the **Activate Colormaps** button () so it's in the enabled state (). PC-DMIS now actively shows colormaps in the Graphic Display window.

To hide the colormaps in the Graphic Display window, click the **Activate Colormaps** button again so it's in the disabled state (). You can also select **None** from the **Colormaps** list to disable colormaps.

To show the colormaps:

- Click the **Activate Colormaps** button so it's in the enabled state. When you enable this button, PC-DMIS displays the colormaps in the Graphic Display window based on the cursor position in the Edit window.
- Select a colormap from the **Colormaps** list.
- When you apply or execute a colormap, PC-DMIS automatically sets the **Activate Colormaps** button to the enabled state.



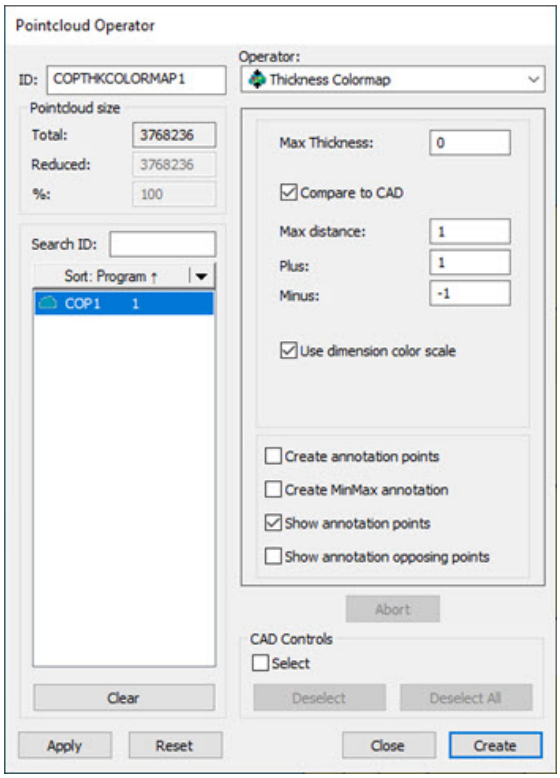
When the cursor is on a Mesh, Point, Surface or Thickness colormap in the Edit window, the active colormap appears in the Graphic Display window. PC-DMIS also displays the **Colormap ID** in the **Colormap** combo box.

If your cursor is above all the colormaps in the Edit window, PC-DMIS does not show any colormap in the Graphic Display window, and sets the **Colormap** combo box to **None**.

Measured Pointcloud Thickness Colormap

To measure a Pointcloud Thickness Colormap:

1. Click the **Pointcloud Thickness Colormap** button  from the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**) or click the menu option (**Insert | Pointcloud | Thickness Colormap**) to open the **Pointcloud Operator** dialog box for the Pointcloud Thickness Colormap operator.



2. Select a Pointcloud (COP) data object from the list.

> When you measure the thickness of a Pointcloud data object, you cannot select a method type. PC-DMIS automatically uses the Sphere method.

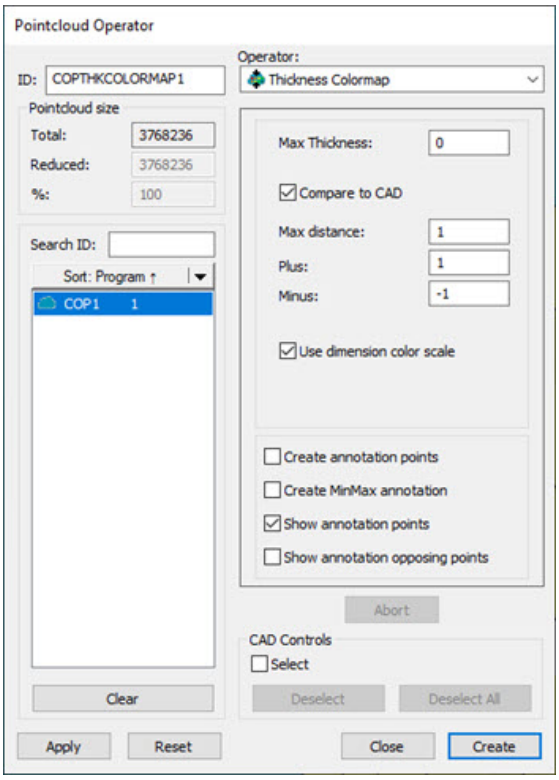
3. Enter the value for **Max Thickness**. The software does not evaluate any data value greater than the **Max Thickness** value.
4. Click **Apply**.
5. Create Annotations. For details, see "Thickness Colormap Annotations".
6. Click **Create**.

Compare to CAD Pointcloud Thickness Colormap

The **Compare to CAD** thickness colormap processing can be very time-consuming when working with a large pointcloud (greater than one million points). We recommend that you filter the pointcloud prior to performing the **Compare to CAD** operation. For details on filtering a pointcloud, see "FILTER" in this documentation.

You can create a thickness colormap of a Pointcloud data object in comparison to a CAD model. In this case, from the **Pointcloud Operator** dialog box, select the **Compare to CAD** check box.

When you select CAD faces for a Thickness Colormap compared to CAD, PC-DMIS uses both sides of the material to compute the thickness; however, PC-DMIS only colors the data for the selected faces.



PC-DMIS computes the deviation of the data object thickness in comparison to a CAD model.

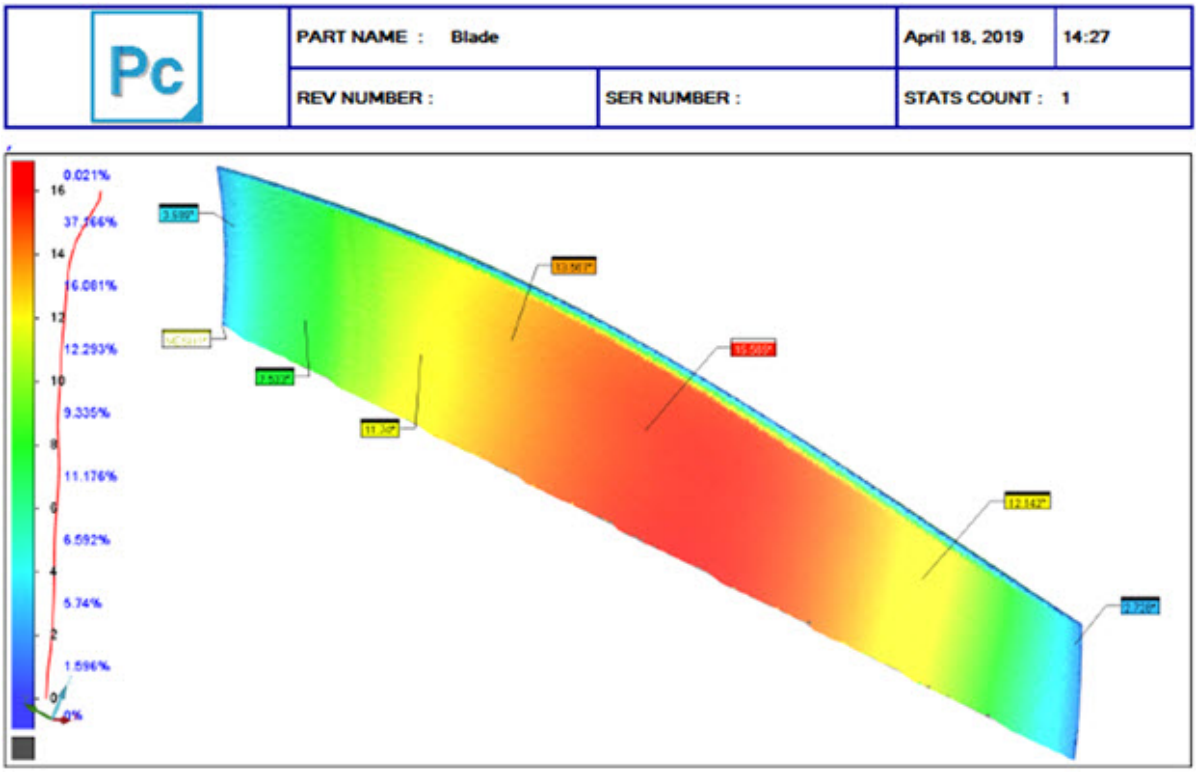
The software colormaps the Pointcloud data object used for the comparison to CAD to show the deviations.

To do this:

1. From the **Pointcloud Operator** dialog box, select **ThicknessColormap** from the **Operator** list.
2. Choose the corresponding Pointcloud data object.
3. Enter the **Max Thickness**. The software does not evaluate any data value greater than this value.
4. Click **Compare to CAD** and enter the appropriate tolerance values in the **Plus Tol** and **Minus Tol** boxes. You must use a minus sign when you enter a negative number.
5. Enter the **Max distance**. PC-DMIS uses the data within this distance from the CAD model for the colormap.
6. Click **Apply**.
7. Create annotations. For details on how to create annotation points for the Thickness Colormap operator, see "Thickness Colormap Annotations".
8. Click **Create**.

Show the Thickness Colormap in the Report

To show the Thickness Colormap in a report, select **Insert | Report command | Snapshot**. PC-DMIS inserts a snapshot of the colormap image when it creates the report.



Example of a thickness colormap report

Thickness Colormap Annotations

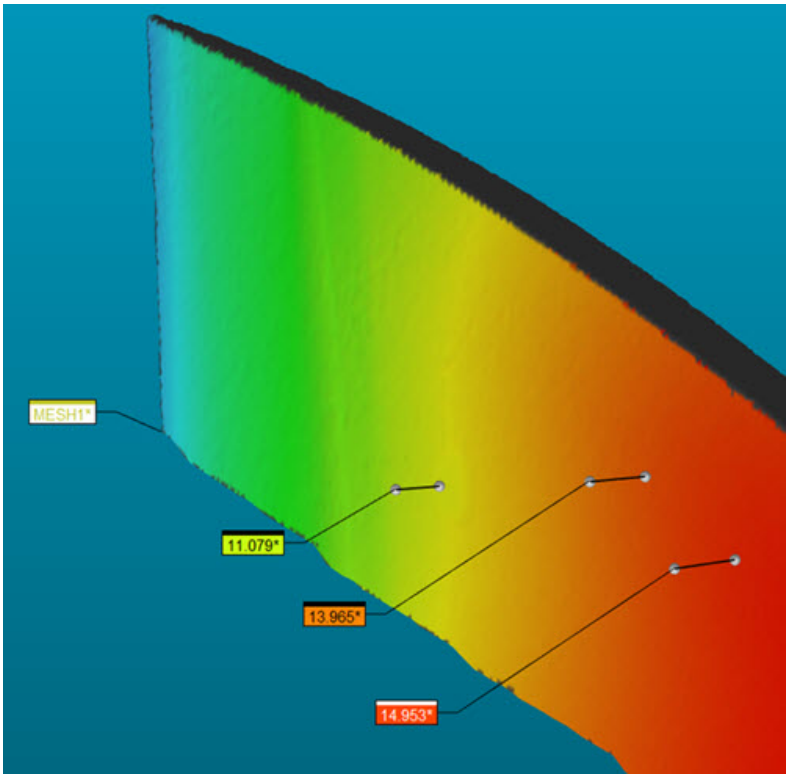
You can create annotation points, Min/Max annotations and show/hide annotations for Thickness colormaps, similar to Surface Colormaps.

- When you create annotations for a Pointcloud or Mesh data object (not compared to a CAD model), the annotation shows the measured thickness.
- When you create annotations for a Pointcloud or Mesh which has been compared to a CAD model, the annotations show the thickness deviation between the measured thickness and the CAD.

Show Annotation Opposing Points

To understand how PC-DMIS measures the annotation point, you can have PC-DMIS display the initial clicked point, the point on the opposite side of the part thickness, and the connecting line in the Graphic Display window. To do this, click the **Show annotation opposing points** check box.

You can change the size of the annotation points and the thickness of the connecting line from the **Pointcloud** area of the **OpenGL** tab on the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.



Thickness colormap example with the Show annotation opposing points check box selected

Example Pointcloud File Formats for Thickness Colormaps

When you use a pointcloud, the data must have XYZIJK values or stripe information.

Example file formats are shown below:

```
20.91911 -3.91231 6.62312 0.52816 -0.84145 -0.11401

21.09812 -3.96453 6.52849 0.48867 -0.86438 -0.11851

21.98763 -4.04430 6.50748 0.47940 -0.88303 -0.09803

22.49231 -4.05894 6.51137 0.50725 -0.85229 -0.12762

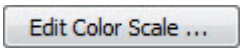
22.89023 3.93331  6.52312 0.52616 -0.85145 -0.12401
```

Example pointcloud file in XYZIJK format

```
L0##1##1##0.724029##-0.499422##0.475746
827.932922 34.322559 186.829498
827.927063 34.331051 186.841080
827.922791 34.338451 186.853577
827.922607 34.343029 186.868881
827.924866 34.345963 186.885864
827.927795 34.348576 186.903214
827.934082 34.353867 186.937988
827.942688 34.362518 186.989517
827.953796 34.373577 187.058304
827.969788 34.389599 187.161560
827.992676 34.409428 187.300430
828.029541 34.437286 187.510300
828.089600 34.476681 187.827393
828.137268 34.509426 188.090515
828.191040 34.551125 188.403336
828.259766 34.602585 188.785507
828.335510 34.659737 189.218796
828.387390 34.701157 189.529175
828.455322 34.758785 189.940521
828.519897 34.820339 190.347870
828.587646 34.881676 190.772919
828.625549 34.920185 191.025818
828.665955 34.975124 191.340225
```

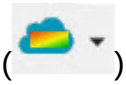
Example pointcloud file with stripe information

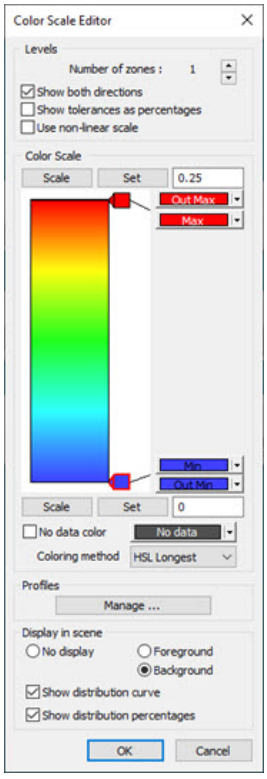
Edit the Color Scale



The **Edit Color Scale** button is available on the **Pointcloud Operator** dialog box for the Point Colormap and Surface Colormap operators. It allows you to change the color scale for these operators. By default, the Min/Max values of the scale are set to the +/- tolerance values of the colormap. You can save different color bars and then recall them with this function.

To begin:

1. From the **Pointcloud** toolbar, select **Pointcloud Point Colormap** () or **Pointcloud Surface Colormap** () to open the **Pointcloud Operator** dialog box for the operator.
2. Click the **Use dimension color scale** check box to clear it and display the **Edit Color Scale** button.
3. Click the **Edit Color Scale** button to display the **Color Scale Editor** dialog box:



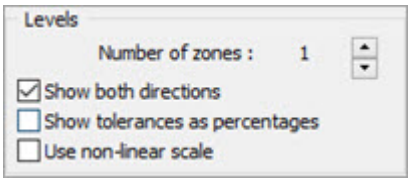
Color Scale Editor dialog box

The following areas of the dialog box are described.

- **Levels** area
- **Color Scale** area

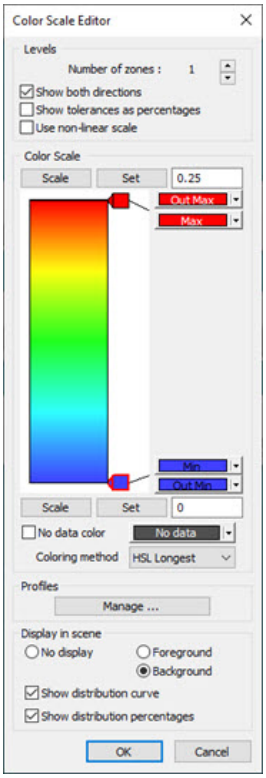
- **Profiles** area
- **Display in scene** area

Color Bar Levels Area



Levels area of the Color Scale Editor dialog box

Number of zones - This setting allows you to change the number of color zones the software displays in the color bar. A setting of 1 displays the gradient view shown below:



Color Scale Editor dialog box

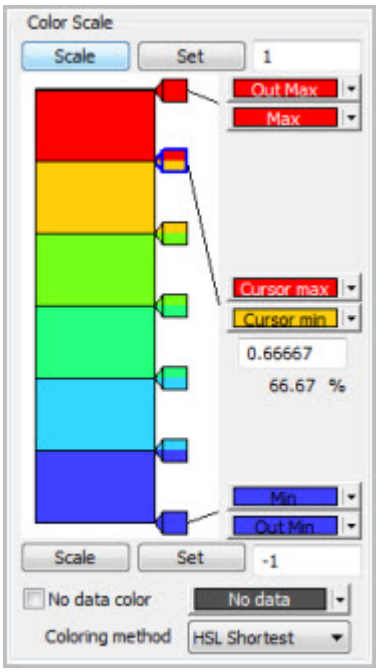
Click the up and down **Levels** arrows to change the number of tolerance zones. You can also click any of the current zones to create a new zone at that location.

Show both directions check box - Select this check box to enable the **Min** value's **Scale** and **Set** controls. If you do not select this check box, the **Min** value's **Scale** and **Set** controls are disabled. The **Min** value in this case is the negative of the **Max** value.

Show tolerances as percentages check box - If you select this check box, the software displays the Color Scale with percent values instead of tolerance values.

Use non-linear scale check box - Select this check box to display a non-linear colormap scale. For details, see the "Non-linear Colormap Scale" topic in this documentation.

Color Scale Area



Color Scale area of the Color Scale Editor dialog box

Color Scale section - This section determines the tolerance zones and the colors associated with the measured values in relation to the respective tolerances. The **Scale** and **Set** buttons change the Max and Min tolerance values with the following differences:

Scale button - If you click this button, the software appropriately scales the intermediate zone values designated by the tolerance markers around the new Max and Min values.

1. Enter a new Max or Min value and then click **Set**. If you change the Max and Min values on the color bar, this also changes the Plus and Minus tolerance values on the colormap.
2. Click the respective **Scale** button. All the zones in the color bar appear the same, except the software appropriately scales the values of each marker around the new Max and Min values.

Set button - Click this button to change the upper value of the highest zone, or the lower value of the lowest zone. The intermediate zone values designated by the tolerance markers remain the same.

1. Enter a new Max or Min value.
2. Click the respective **Set** button. PC-DMIS changes the corresponding Max or Min zones. All intermediate zone values remain the same.

To change Zone values, click and drag one of the zone markers. You can also enter the Zone values. To enter new zone values:

1. When you click a zone marker, PC-DMIS displays a leader line from the marker to the selected zone and a field appears.
2. Enter an appropriate value in the field then click outside the field for the value to take effect.

No data color check box - Select this check box to set the color where no data exists based on the colormap Max Distance. To define the color for this option:

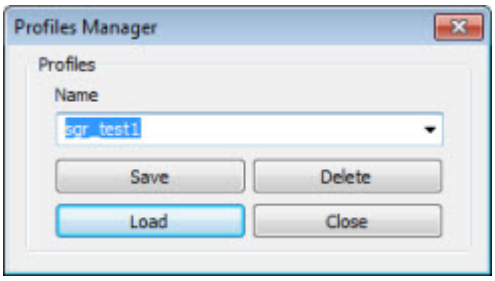
1. Click the drop-down arrow to the right of the check box to display the standard color picker dialog box.
2. Select the color for this option and click **OK**.
3. Click the check box to select it and apply this option to your surface colormap.

Coloring method - The drop-down list provides pre-defined color bar color schemes you can select. Click the drop-down arrow to display the list and select the color scheme you want to apply.

Color Bar Profiles Area

Use the **Profiles** area of the **Color Scale Editor** dialog box to manage color bar schemes.

Click the **Manage** button to open the **Profiles Manager** dialog box.



Profiles Manager dialog box

The dialog box has these options:

- If this is a new color scheme, type a unique name for the color scheme in the **Name** box and click **Save**. The software saves the current color bar profile under the name entered.
- To load a color scheme profile, select one from the **Name** list and click **Load**.

- To delete an existing profile, select one from the **Name** list and click **Delete**. The software permanently deletes the profile - this cannot be undone so use caution when you delete a color scheme.

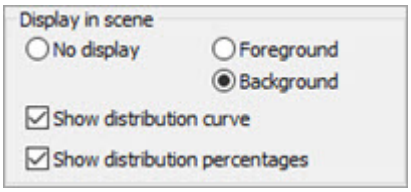


You can also begin typing a profile name in the **Name** field to filter the list based on your entry.



PC-DMIS saves the files with a .cbr extension in the same folder as the measurement routines.

Color Bar Display in Scene Area



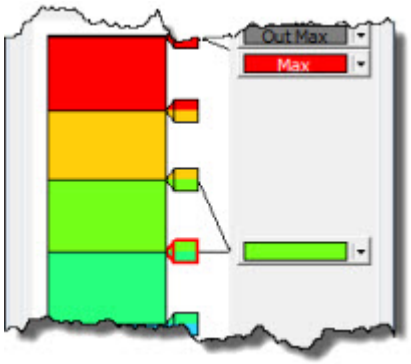
Display in scene area of the Color Scale Editor dialog box

The **Display in scene** area of the **Color Scale Editor** dialog box defines how the color scheme appears in the Graphic Display window. The options are:

- **No Display** - The Color Bar does not appear in the Graphic Display window.
- **Foreground** - The Color Bar appears on top of the CAD objects in the Graphic Display window.
- **Background** - The Color Bar appears behind the CAD objects in the Graphic Display window.
- **Show distribution curve** check box - When you select this check box (default), the software displays the distribution curve histogram layered on top of the color scale data values. The curve provides a visual indicator of the colormap deviations within the tolerance zones.
- **Show distribution percentages** check box - When you select this check box (default), the software displays the percentage values along with the color scale data values. This shows the percentage of deviation within the tolerance zones.

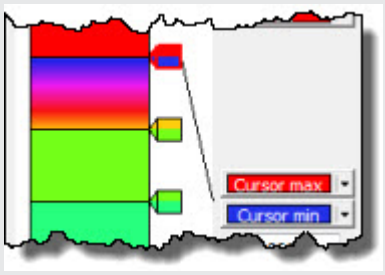
Changing a Zone's Color

1. Click the Max tolerance marker  for the particular zone, then press the Ctrl key on your keyboard and click the Min tolerance marker for the same zone.

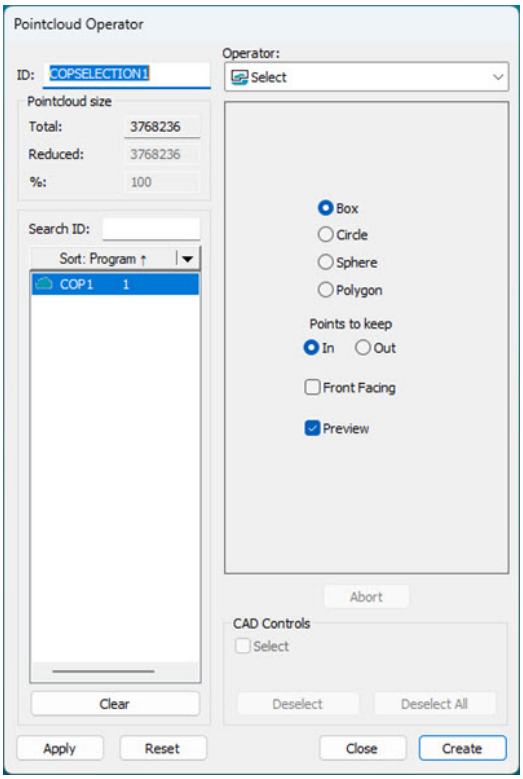


2. Once selected, click the drop-down arrow to display the standard color-picker dialog box.
3. Select the new color and click **OK**. The software changes the color for the selected zone to the new color.

If you only change the Max or Min value of a zone, PC-DMIS only changes that zone's color to a gradient scheme. For example, if you change just the Max color of a zone, the zone's gradient color scheme is based on the new Max color selected and on the current color of the Min value as shown below.



SELECT

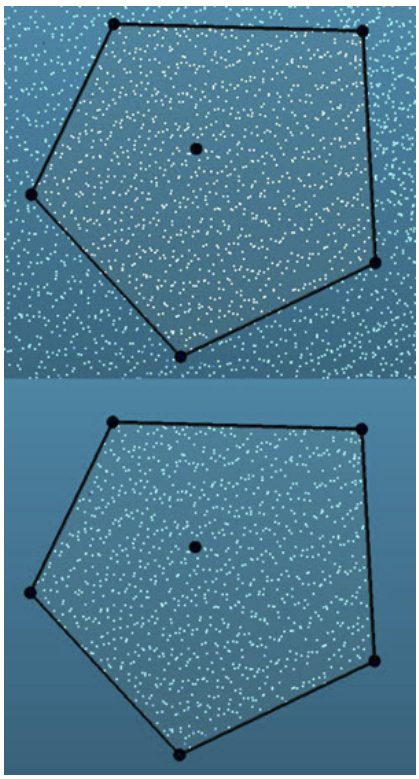


Pointcloud Operator dialog box - Select Operator

To apply the SELECT operation to a pointcloud, from the **Pointcloud** toolbar, click **Select Pointcloud** ( ), or select **Operation | Pointcloud | Select**. By default, the software uses the **Polygon** option when you click the **Select Pointcloud** button.

The SELECT operation selects a subset of data contained in a COP command.

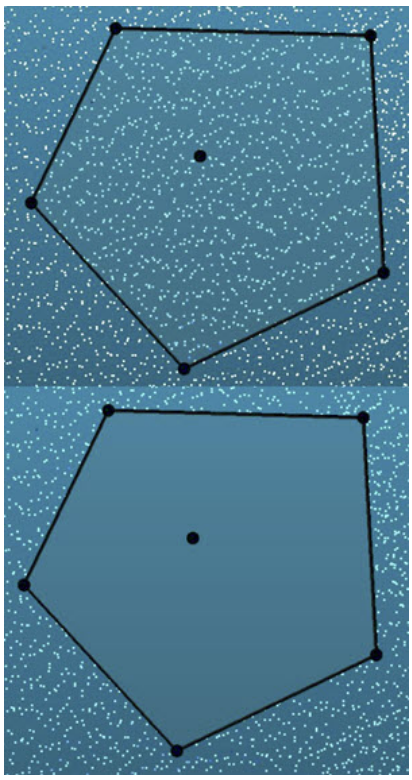
- PC-DMIS keeps only a subset of the pointcloud data strictly inside the polygon selection when you select the **In** option.



Pointcloud selection showing data using the In option:

The top Image is the initial selection.
The bottom image is the result after you click the Apply button.

- PC-DMIS keeps only a subset of the pointcloud data strictly outside the polygon selection when you select the **Out** option.



Pointcloud selection showing data using the Out option:

The top Image is the initial selection.
The bottom image is the result after you click the Apply button.

To select a region of points:

1. From the **ID** list of commands, select the Pointcloud ID that you want to apply the selection to.
2. From the **Pointcloud Operator** dialog box, choose a selection method:



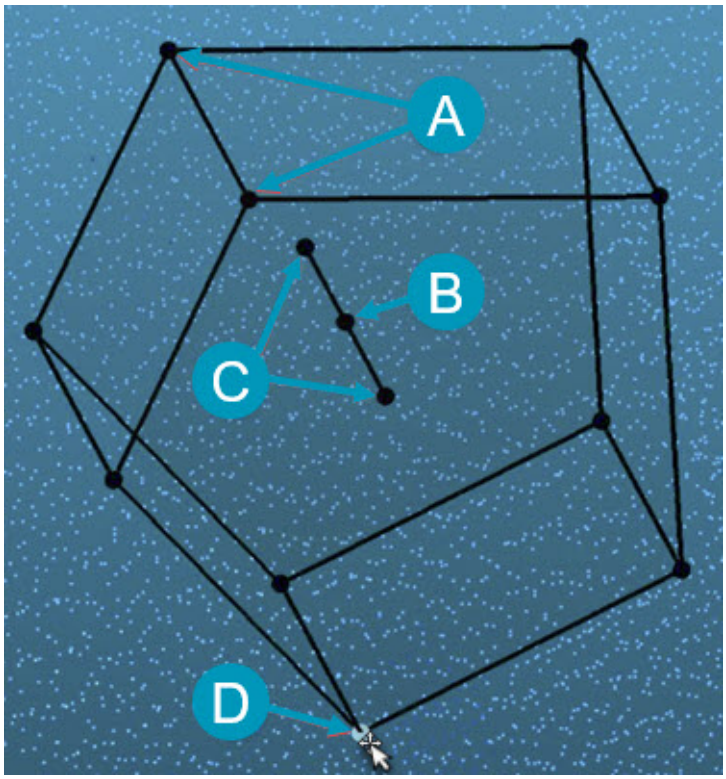
Use Ctrl + left-mouse click when you make the selection. This snaps the center of the selection onto the Pointcloud.

Also, you may find it helpful to turn off the **Show Graphic Window as Solid** option from the **Graphic View** toolbar (**View | Toolbars | Graphic View**) so you can see the pointcloud.

Selection Method	Points Needed
Box	Click and drag to define a rectangular shape.
Circle	Click to set the center of the circle and then drag to define the circle size.
Sphere	Click to set the center of the selection sphere and then drag to define the spherical shape.
Polygon	Click to define the vertices of the polygon. To close the polygon selection, double-click your left mouse button or press the End key.

3. In the Graphic Display window, click and drag an area to select a portion of the pointcloud. The type of area you select depends on the selection method made in Step 2.
4. You can change the outline, position, and depth of the selection:
    - To change the outline of a box, circle, or sphere, drag an edge to a new position. For a polygon, drag a vertex.
    - To rotate a box selection, drag a corner vertex to a new position.

- To move a selection's position, drag the center control point to a new position.
- By default, the box, circle, and polygon selections are 3D, extending through the pointcloud, normal to the selection's view. You can click and drag one of the center depth points to change the depth of the selection.



Polygon region showing:

A. *Region boundary points*

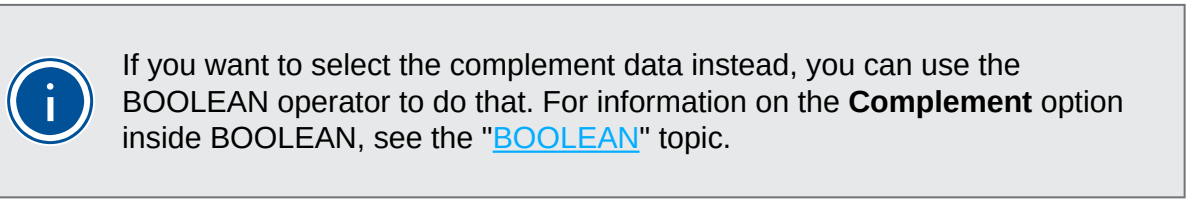
B. *Region center control point*

C. *Region center depth points*

D. *Hover over a vertex on a polygon selection (or an edge for box, circle or sphere selections) until it is highlighted, and then click and drag it to change the outline of the selection*

5. If you want to keep the points inside the selection, from the **Pointcloud Operator** dialog box, click **In**. If you want to keep the points outside the selection, click **Out**.
6. If you want to keep only the points that are front facing in the selection view, click the **Front Facing** check box. For details, see "Pointcloud Select - Front Facing".
7. The software selects the **Preview** check box by default. This allows you to see the Pointcloud when you select a region. You may want to clear the check box so the selected Pointcloud data are not highlighted. This may be helpful if you have a very large Pointcloud object and showing the selected pointcloud data affects performance.

8. After you click the necessary points in the Graphic Display window, click the **Apply** button to perform the selection. PC-DMIS displays the points inside or outside the selected domain in the Graphic Display window.
9. When you're finished, click **Create**. PC-DMIS inserts a `COP/OPER,SELECT` command into the Edit window.



Click **Create** to insert a `COP/OPER,SELECT` command into the Edit window.

PC-DMIS creates the Pointcloud SELECT command in the CAD coordinate system. Click the links below to see descriptions of the values for each selection method.

COPSELECT Box Example

```
COPSELECTION1=COP/OPER,SELECT,BOX,INSIDE=YES,FRONT_FACING=NO,
    <77.136,-0.01,-25.715>,<0,-1,0>,<1,0,0>,17.104,28.514,0,SIZE=110280
```

Where:

77.136,-0.01,-25.715 = The selection's XYZ Center Location values.

0,-1,0 = The selection's View I,J,K values.

1,0,0 = The selection's Major Direction values.

17.104 = The selection's Width value.

28.514 = The selection's Length (Major Direction) value.

0 = The selection's Depth value.

COPSELECT Circle Example

```
COPSELECTION1=COP/
OPER,SELECT,CIRCLE,INSIDE=YES,FRONT_FACING=YES,
    <0,80.045,-70.337>,<1,0,0>,22.047,15.143,SIZE=3388,REF=COP1,,
```

Where:

0,80.045,-70.337 = The selection's XYZ Circle Center Location values.

1,0,0 = The selection's View I,J,K values.

22.047 = The circle's Diameter value.

15.143 = The selection's Depth value.

COPSELECT SPHERE Example

```
 COPSELECTION3=COP/
OPER,SELECT,SPHERE,INSIDE=YES,FRONT_FACING=YES,

     <-95.864,149.866,-9.443>,56.185,SIZE=2278666,REF=COP1,,
```

Where:

-95.864,149.866,-9.443 = The sphere's XYZ Center values.

56.185 = The sphere's Diameter value.

COPSELECT POLYGON Example

```
 COPSELECTION4=COP/
OPER,SELECT,POLYGON,INSIDE=YES,FRONT_FACING=NO,10.709,

     <0,2.889,-2.864>,

     <0,27.35,-2.864>,

     <0,28.586,-98.603>,

     <0,2.148,-98.356>,

     <0,1.901,-68.006>,

     <0,22.161,-68.006>,

     <0,21.667,-32.72>,

     <0,3.136,-32.967>,

     <0,3.136,-32.967>,

     SIZE=2278666,REF=COP1,,
```

Where:

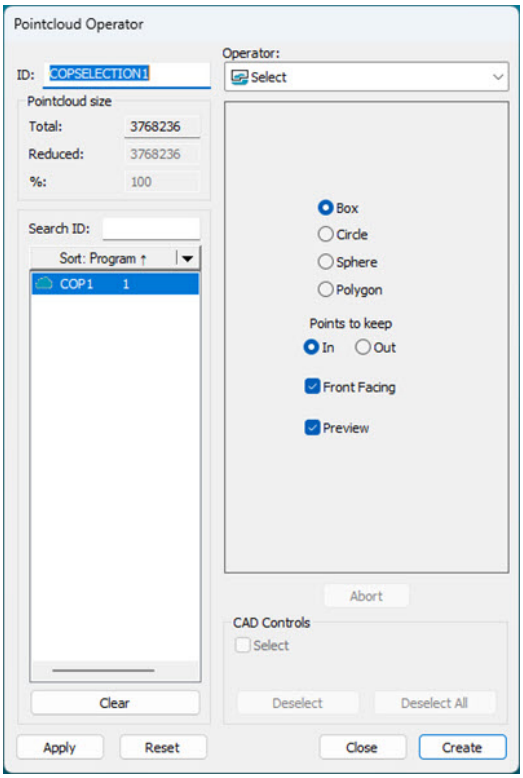
10.709 = The selection's Depth value.

The Depth value is followed by nine sets of XYZ coordinate values of the Polygon's Vertices in the Selection View.



A 0 (zero) depth value means infinite depth, cut through the entire pointcloud in the selection view.

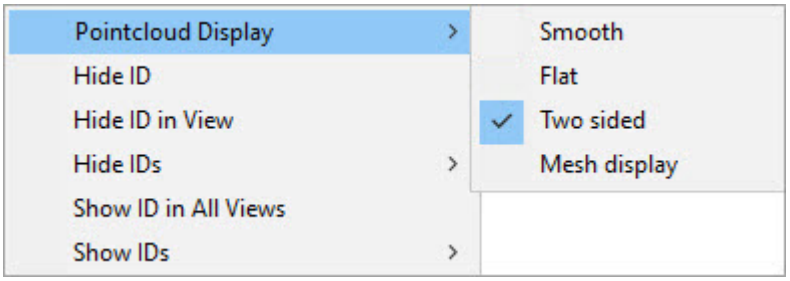
Pointcloud Select - Front Facing



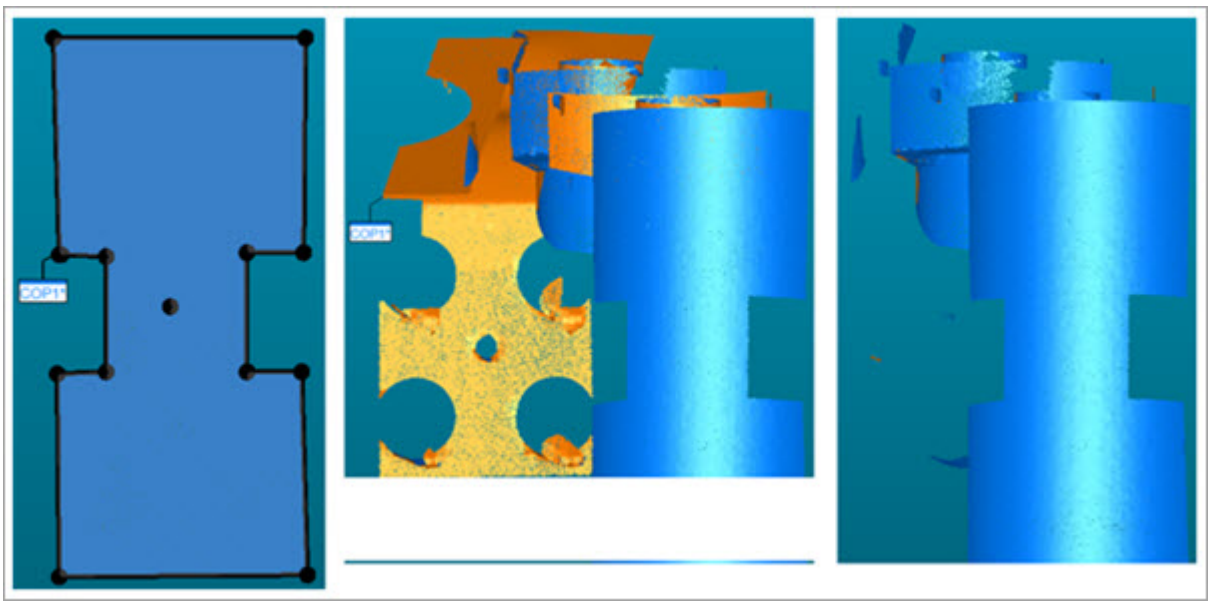
Pointcloud Operator dialog box - Select Operator with Front Facing option selected

Click the **Front Facing** check box on the **Pointcloud Operator** dialog box if you only want to select the points that are front facing in the selection view. When you disable this check box, PC-DMIS selects the points that are in both the front and rear facing views.

To see the front and rear faces as contrasting colors, right-click the pointcloud in the Graphic Display window and select **Pointcloud Display | Two sided**.

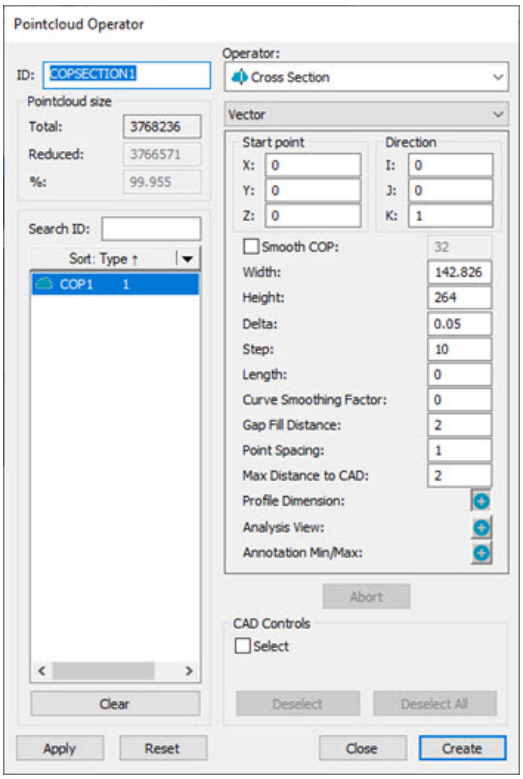


The pointcloud must have stripe information or IJK values for each point to use this option.



Examples of Selection View (left), Selection with Front Facing disabled (middle), Selection with Front Facing enabled (right)

CROSS SECTION



Pointcloud Operator dialog box - Cross Section Operator

The CROSS SECTION operation generates a subset of polylines determined by the defined intersection of a set of parallel planes with the COP or Mesh object. The set of planes is defined by the start point, direction vector, step distance between the planes, and length. The number of planes is determined by the **Step** distance divided into the **Length** plus one.

You can use the legacy profile dimension to evaluate the CROSS SECTION operator. Geometric Tolerance commands do not support pointcloud or mesh operators.

To apply the CROSS SECTION operation to a pointcloud, click **Cross Section Pointcloud** () on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Cross Section**.

From the **Pointcloud** or **QuickCloud** toolbar, click the **2D Section Slide Show** button  to display cross sections in the 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

The drop-down list under the **Operator** list contains these options: **Vector**, **Axis**, **Curve**, and **2 Points**. For details on how the **Curve** function works, see the "Creating a Cross Section along a Curve" topic. For details on the **2 Points** option, see the "Creating a Cross Section between 2 Points" topic.

The CROSS SECTION operator uses the following options:

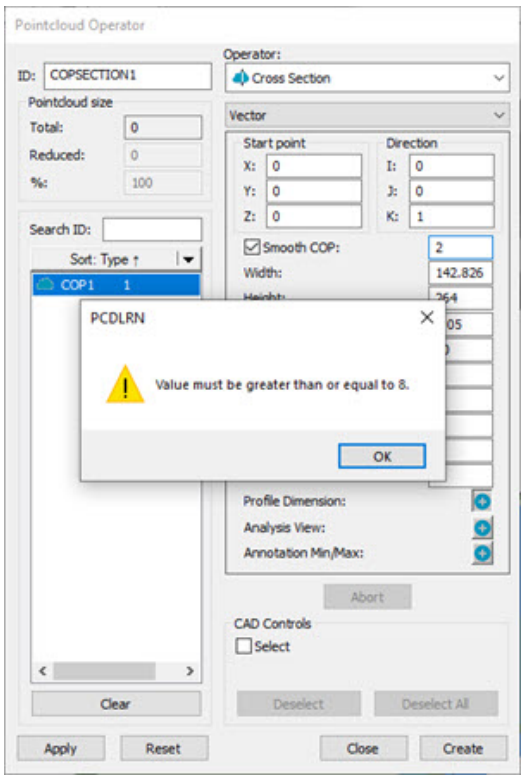
- **Start point**: This value indicates the coordinates of a point that belongs to the first plane that cuts the pointcloud.

  The software displays the start point as a blue ball in the Graphic Display window. You can use the ball as a handle to drag to a new location. You define the start point by your first click in the Graphic Display window. In the actual Edit window command, the start point value is held in the START PT parameter.

- **Direction** (applies only to the **Vector** and **2 Points** option): This value indicates the direction of the normal vector. You define the direction by your second click in the Graphic Display window. In the actual Edit window command, the **Direction** value is held in the NORMAL parameter.

- **Axis** (applies only to the **Axis** option): Use this option to create a cross section along the X, Y, or Z axis. Select the desired axis (the default is X), set a start point in the Graphic Display window, and set an end point. The section plane cuts the part at a given step value over the length of the cross section.

- **Smooth COP** check box: Select this check box to optimize the point positions in order to smooth the pointcloud. The value you enter in the **Smooth COP** box represents the number of neighboring points to consider within a defined area that PC-DMIS considers when creating the cross section. The higher the value, the more neighboring points are included. The minimum valid value is 8. If you enter a value less than 8, PC-DMIS shows an error message:



- **Width**: This value indicates the width of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.
- **Height**: This value indicates the height of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.
- **Delta**: This value indicates the maximum distance from the plane for a point to be considered part of the cross section. In the actual Edit window command, the **Delta** value is held in the TOLERANCE parameter. The **Delta** property is only available when a COP object is selected.
- **Step**: This value indicates the distance between the planes. In the actual Edit window command, the step value is held in the INCREMENT parameter.

> If the value in the **Step** box is greater than the value in the **Length** box, then only one section cut is created at the start point.

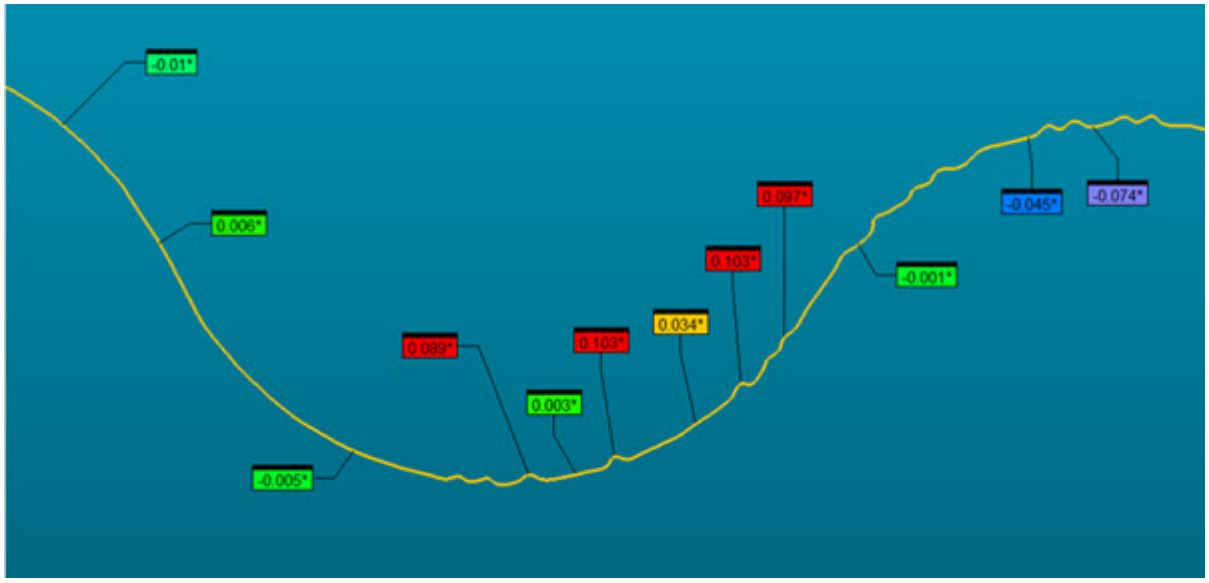
- **Length**: This value indicates the maximum distance between the first and last plane. The software displays the length value in the **Length** parameter of the dialog box. PC-DMIS displays it as a purple line in the Graphic Display window.
- **Curve Smoothing Factor**: You can use the **Curve Smoothing Factor** to apply a Least Square fit with a smoothing constraint to an ordered set of points. There is a trade-off when using the **Curve Smoothing Factor**, since, as the roughness of the spline is smoothed, the points move from their original position. Consequently, you should use **caution** when you apply the smoothing factor since it moves or changes

the data. This option may be useful for smoothing points that you consider to be "noise".

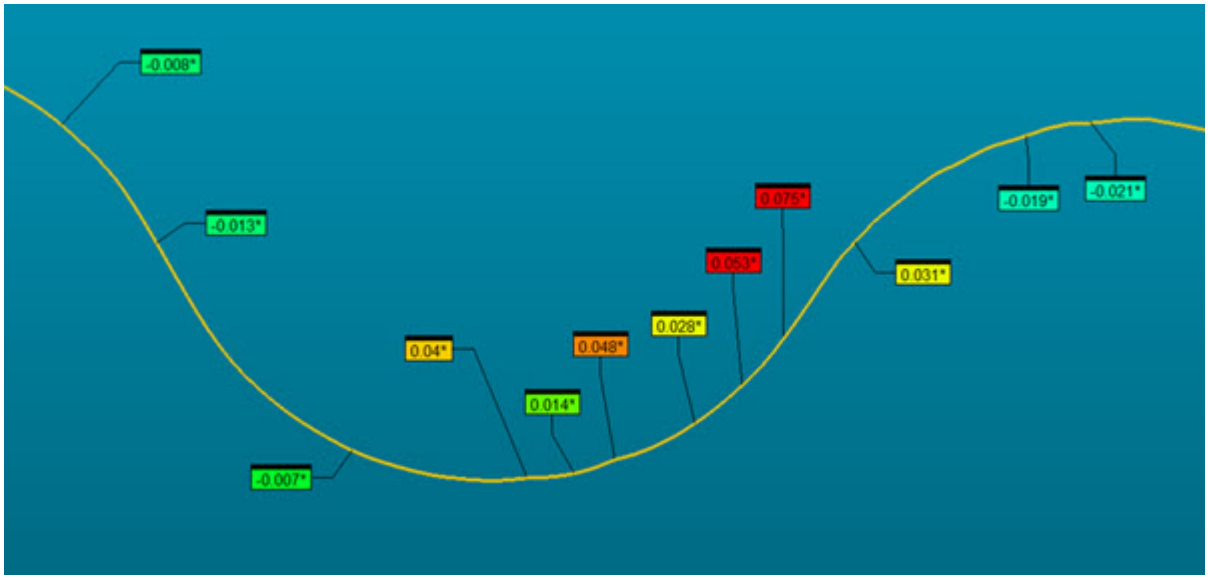
There is no upper limit to the **Curve Smoothing Factor** value. However, the more you increase this value, the less effect it has on your data until any noticeable change is seen. When you use a very large smoothing factor, changes occur to the original shape of the measured cross section polyline.

A value of 0 (zero) applies a very small internal smoothing factor.

You should test the **Curve Smoothing Factor** option with a starting value between 0.1 and 0.25. Then, depending on the result, increase the value to 0.5, 1 or 2 and re-test until you achieve the desired result.



Example with the Curve Smoothing Factor set to 0 (zero)



Example with the Curve Smoothing Factor set to 0.5

- **Gap Fill Distance**: This value defines the maximum gap distance along the yellow measured polylines of a cross section. If gaps equal to or smaller than this value appear, PC-DMIS fills in the gaps with calculated points. You can also set this value in the PC-DMIS Settings Editor. For details, see the "`CrossSectionMaximumEmptyLength`" topic in the PC-DMIS Settings Editor documentation.

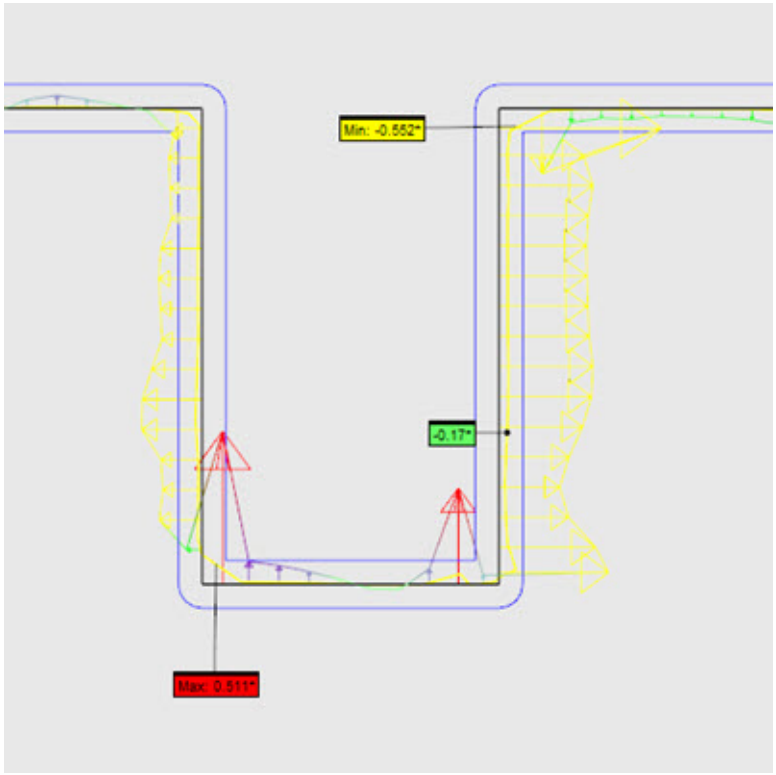
- **Point Spacing**: This value is used only when the `CrossSectionCopCadCrossSectionDrivenByCad` entry is set to 1 (True). This value is the step used along the CAD polylines to look for the best interpolated COP point. For greater accuracy, or if the CAD model is very small, you can set this value to a smaller value.

  You can also set this value in the PC-DMIS Settings Editor. For details, see the "`CrossSectionCopCadCrossSectionStep`" topic in the PC-DMIS Settings Editor documentation.

- **Max Distance to CAD**: This value defines the maximum distance of the pointcloud data relative to the nominal CAD model. The default value is 2 mm. If the scanned part deviates more than the **Max Distance to CAD** value from the CAD model, the software may not compute the yellow measured cross section. You can adjust this value to account for large deviations of the scanned data relative to the CAD model.

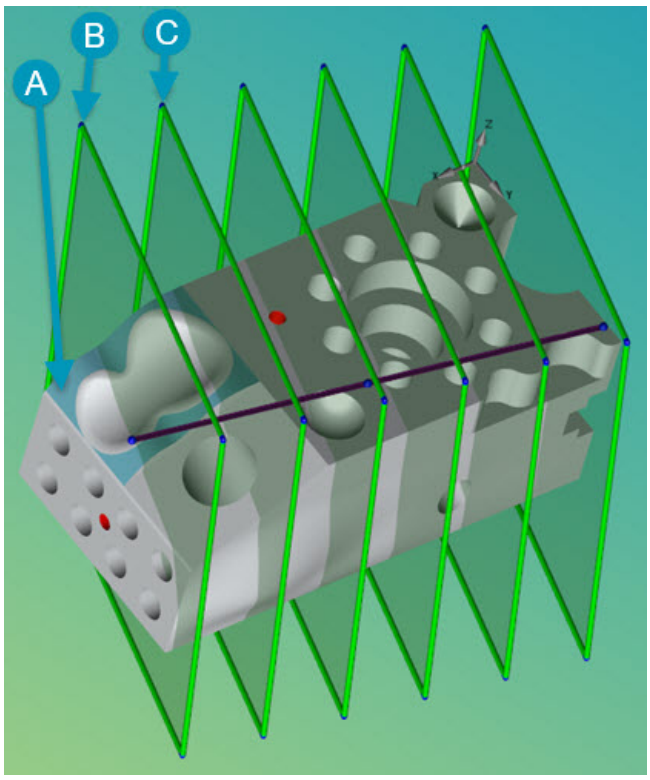
- **Profile Dimension**: Click the **Add** button to create a new profile dimension for each cross section. For details on Profile Dimension, see "Dimensioning Profile - Line or Surface" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

- **Analysis View**: Click the **Add** button to create the `ANALYSISVIEW` command in the Edit window. For details on the `ANALYSISVIEW` command, see "Create Analysis View Command" in the "Inserting Report Commands" chapter in the PC-DMIS Core documentation.
- **Annotation Min/Max**: Click the **Add** button to create the minimum and maximum values in the form of annotation labels for the active cross section.

  PC-DMIS calculates the minimum and maximum points each time you execute the measurement routine.
- **CAD Controls**: Mark the **Select** check box to select CAD surfaces in the Graphic Display window. PC-DMIS filters out any cross sections that do not pass through the selected surfaces when you click **Create**.

  For example, if you selected surface A after the start and end points are defined, PC-DMIS only generates cross sections at B and C:



Example of a selected surface (A) limiting the cross sections to only (B) and (C)

Selected surfaces do not affect what you see when you click the **View** button.

When the cutting planes are visible in the Graphic Display window, you can manipulate them as follows:

- To resize the height and width of the cutting planes, drag a plane's edge handle to a new position.
- To rotate the set of planes around their axis, drag a plane's corner handle to a new position.
- To redefine the purple line's `START` or `END` definition, drag the first or last purple length line's blue point handle to a new position. While the direction is changing, the values in the dialog box and the number of planes in the Graphic Display window are updated. In Axis mode, the direction of the planes does not change.
- To move the set of planes, drag the purple length line's middle blue point handle.

> When a cross section is created or edited, the cutting planes appear in a transparent view as shown above.

Click **Create** to:

- Insert a `COP/OPER,CROSS SECTION` command for each plane into the Edit window. You can see this in the code below:

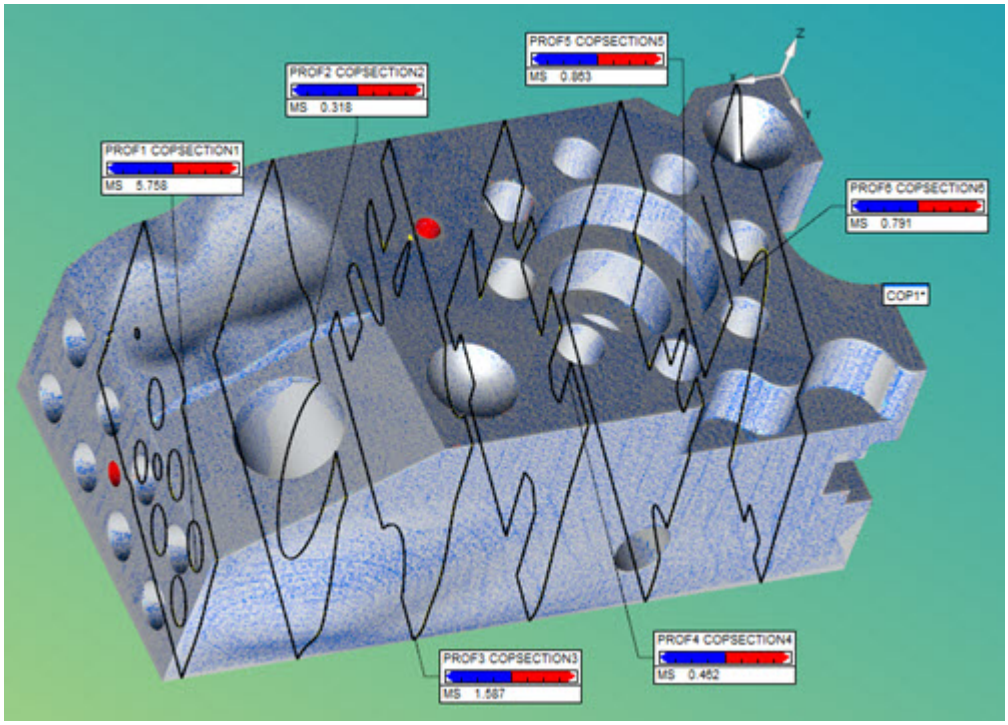
```
 COPSECTION3=COP/OPER,Cross
Section,TOLERANCE=0.05,WIDTH=117.715,HEIGHT=227.086,

     START PT = -6.439,60.097,6.276,NORMAL =
    0.9684394,-0.2221293,-0.1130655,SIZE=76

     REF,COP1,,
```

PC-DMIS represents the nominal CAD with black polylines, and the COP polylines with yellow polylines.

- Insert a label for each plane into the Graphic Display window as shown below:



Finished cross sections showing six planes

Defining the Cross Section by Typing Values

Use the **Pointcloud Operator** dialog box to type the values:

- **START PT**: This value specifies the cross section's starting point using the **Start Point X**, **Y** and **Z** boxes.
- **NORMAL**: This value specifies the cross section's vector using the **Direction I**, **J** and **K** boxes.
- **WIDTH**: This specifies the value of the cross section's width property in the **Width** box.
- **HEIGHT**: This specifies the value of the cross section's height property in the **Height** box.

- **TOLERANCE**: This specifies the value PC-DMIS uses to determine the maximum distance from the plane a point must be in order for PC-DMIS to consider it as part of the cross section in the **Delta** box.
- **INCREMENT**: This specifies the value between cutting planes in the **Step** box.
- **LENGTH**: This specifies the value between the first and last cutting planes in the **Length** box.
- **SMOOTHING TOLERANCE**: This specifies the tolerance value to refine the points associated with the generated cross section in the **Smoothing Tol** box.

Defining the Cross Section by Using the Graphic Display Window

To define some of the cross section parameters, click the CAD model in the Graphic Display window to select the **Start Point**. A pink line appears. Click a second point on the CAD model to determine the **Direction** vector and the **Length**.

Creating a Profile Dimension from the Graphic Display Window

When you double-click a cross section label, PC-DMIS creates a new profile dimension that evaluates the selected cross section.

Measuring a Radius on a Cross Section with the 2D Radius Gage

PC-DMIS provides the 2D Radius Gage to quickly measure the radius on a pointcloud cross section. For details, see "2D Radius Gage Overview".

2D View of Cross Sections

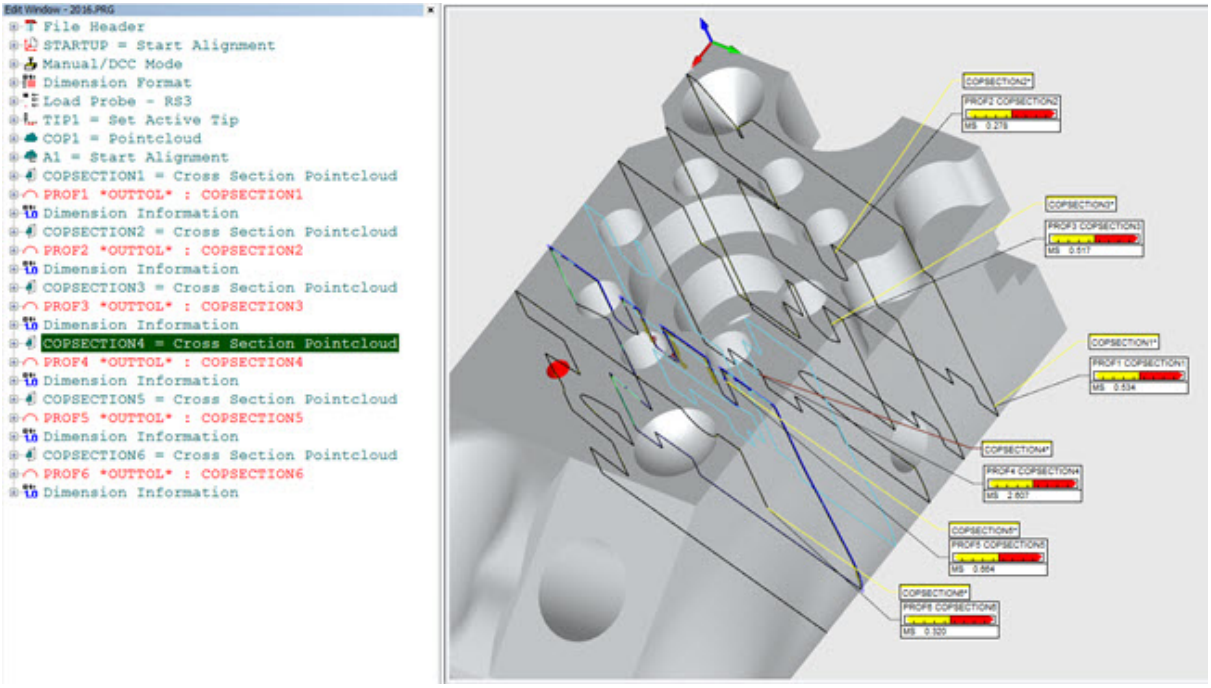
Once you define a cross section, each section can be individually displayed in a 2D view. The view is normal to the cross section. Any annotation points created on the cross section appear in the 2D view.

To view the cross sections in 2D, from the **Graphic Modes, Mesh**, **Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **2D Section Slide Show** button ().

For more information on the 2D Section Slide Show, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

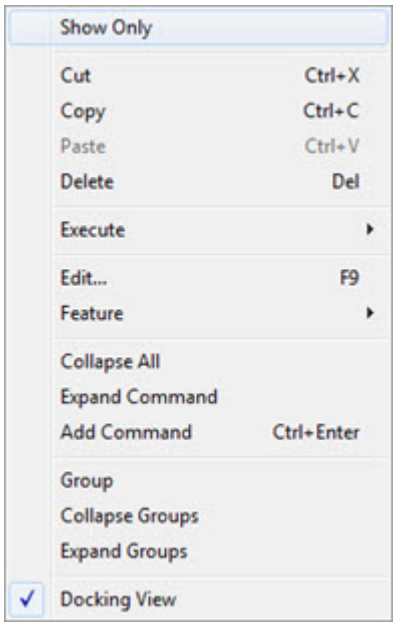
How to view the cross sections in 2D from the Edit window.

a. From the Edit window, click the cross section you would like to view in 2D. The selected section appears in light blue in the Graphic Display window.



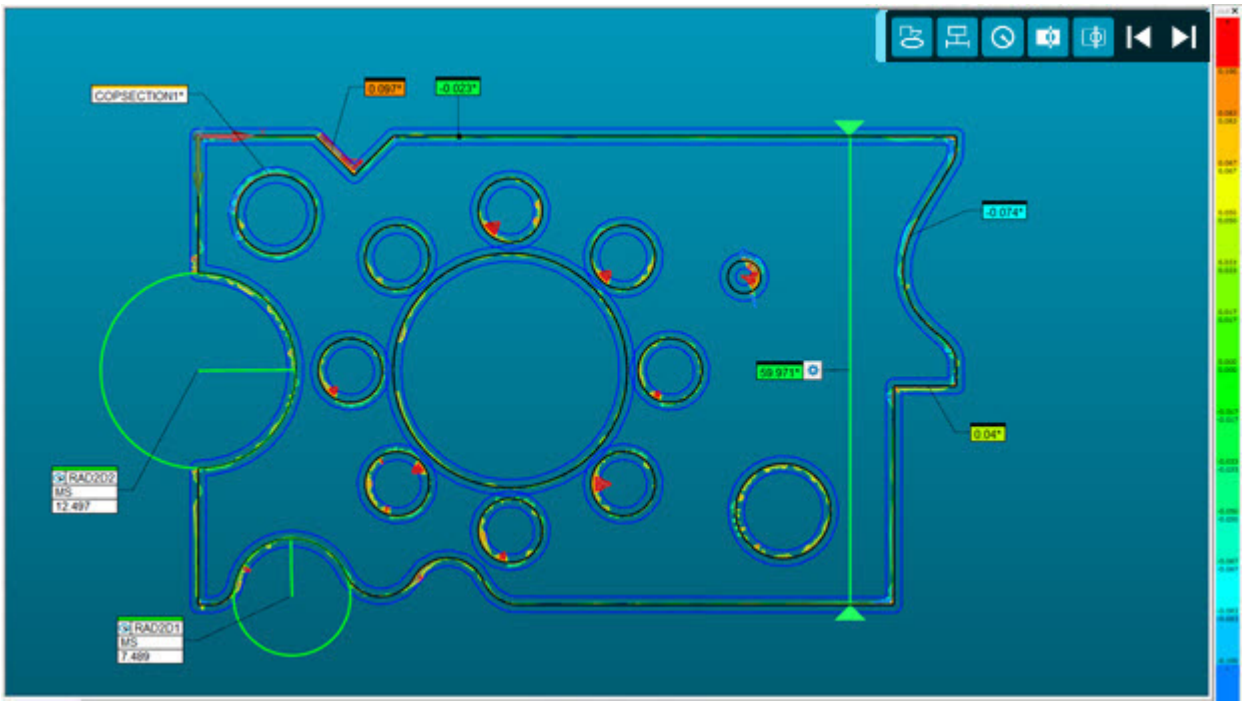
Example of a selected section of a cross section

b. Right-click the selected cross section to display the Edit window's pop-up menu.



c. Click the **Show Only** option to display the 2D view of the selected cross section. When you enable the option, PC-DMIS displays a check mark to the left of it.

When in 2D view, the **Cross Section Graphic Control** toolbar is available.



Example of a section view normal to the cross section

As you hover and move the cursor over the cross section in the Graphic Display window, the labels appear and update in real time. Click any point on the cross section when in 2D view to create an annotation label for that location.

The **Cross Section Graphic Control** toolbar is a floating toolbar that you can position anywhere in the Graphic Display window.



Cross Section Graphic Control toolbar

The buttons from left to right perform these functions:

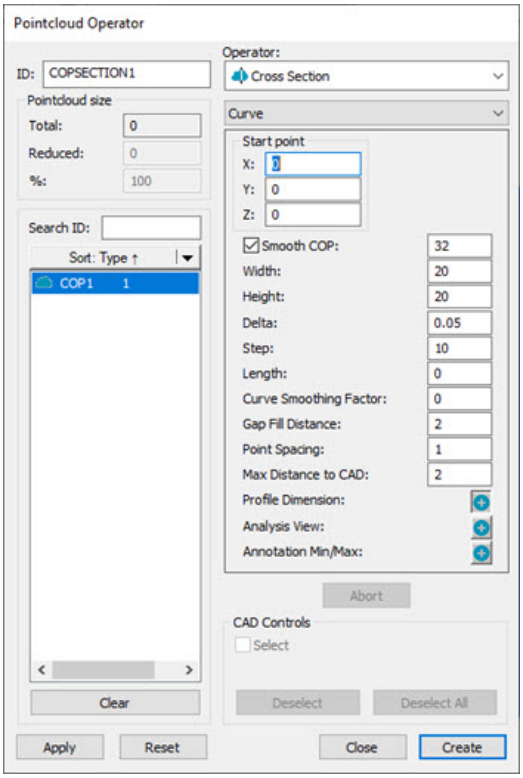
- **Show/Hide annotations** - Shows and hides the annotations.
- **Show/Hide distance gages** - Shows and hides the distance gages.
- **Show/Hide 2D radius gages** - Shows and hides the 2D radius gages.
- **Show/Hide nominal polylines** - Shows and hides the nominal polylines.
- **Show/Hide measured polylines** - Shows and hides the measured polylines.
- **Show the previous 2D Section** - From the currently-selected cross section in the Edit window, each time you click this button, the software displays the previous cross section through to the first cross section.

- **Show the next 2D Section** - From the currently selected cross section in the Edit window, each time you click this button, the software displays the next cross section through to the last cross section.

Click the **Show the previous 2D Section** or the **Show the next 2D Section** button to toggle backward or forward to view the cross sections in a slide show pattern. For details, see the "Cross Section Slide Show" section in the "Show and Hide Cross Section Polylines" topic.

Creating a Cross Section along a Curve

You can create a cross section along a curved feature with the **Curve** function of the **Pointcloud Operator** or **Mesh Operator** dialog box. PC-DMIS creates the cross section normal to the CAD curve.



Pointcloud Operator dialog box - Cross Section Operator, Curve function selected

To create a cross section along a curve:

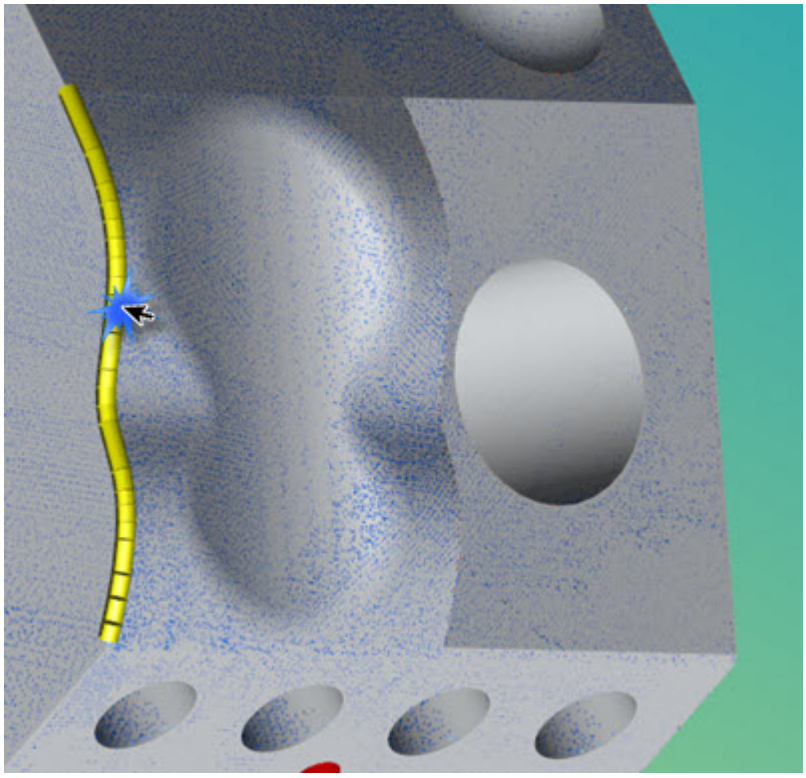
1. For cross sections created with a COP as the input, click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.

   For cross sections created with a Mesh as the input, click **Insert | Mesh | Operator** to display the **Mesh Operator** dialog box.

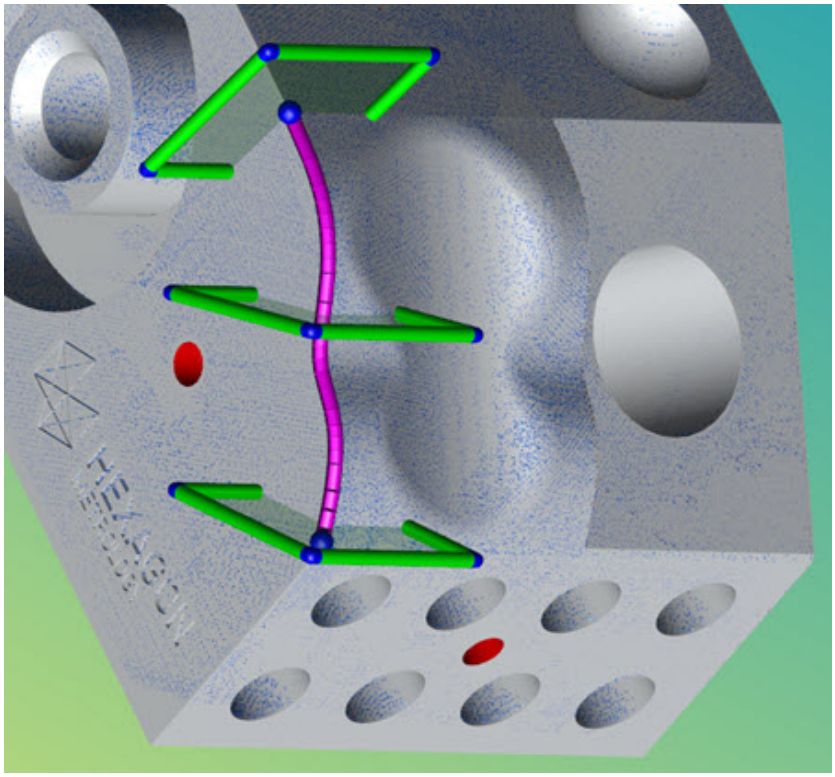
2. From the **Operator** list, select the **Cross Section** operator.

3. Under the **Operator** list, from the drop-down list, select the **Curve** function.

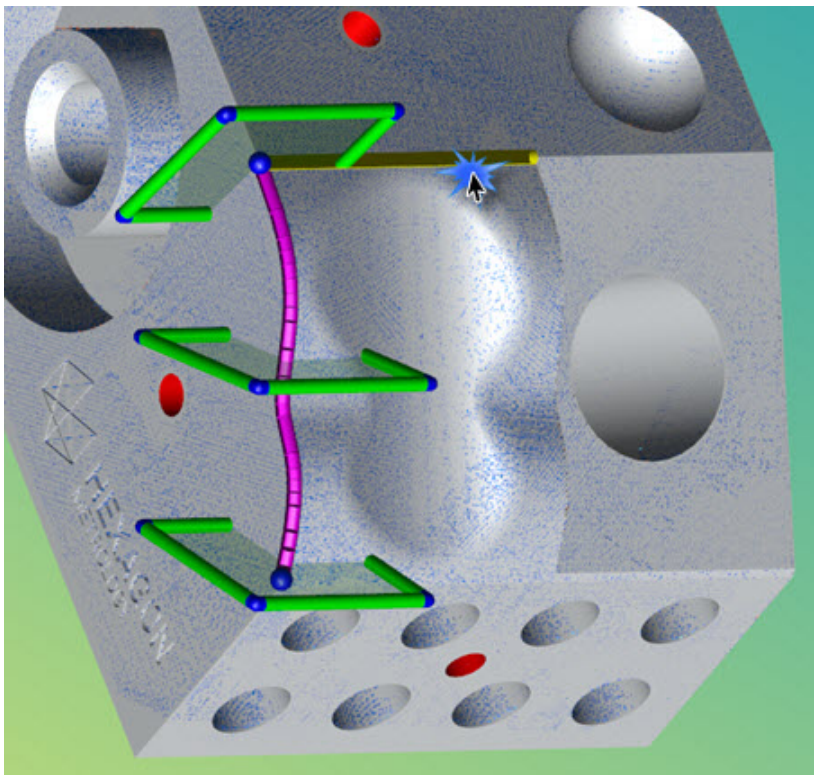
4. In the Graphic Display window, point to any curved feature. PC-DMIS automatically detects and highlights the curve:



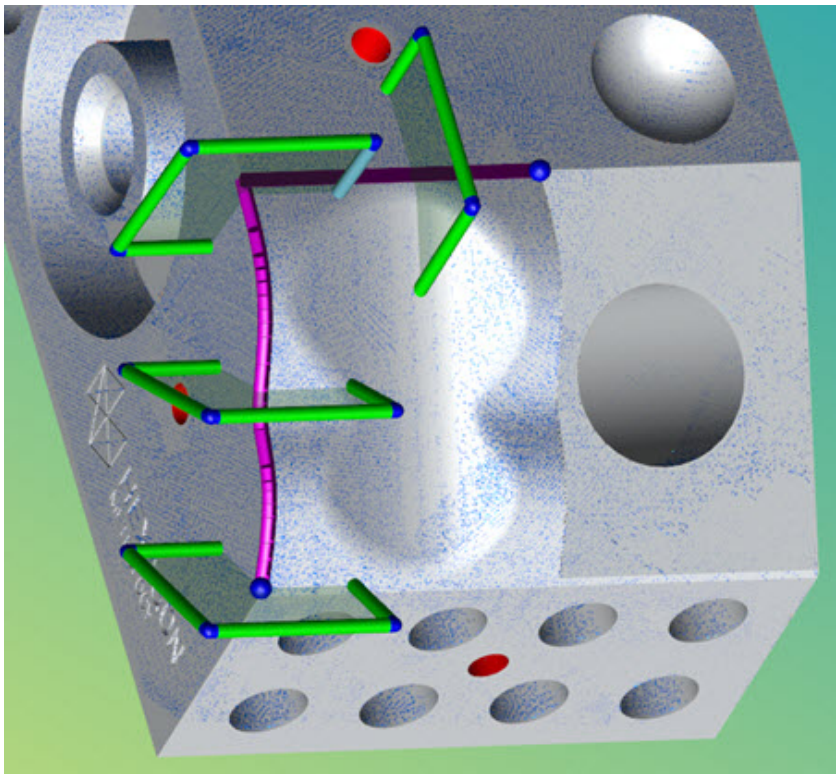
5. Click the highlighted edge that you want to create cross sections on. PC-DMIS automatically generates the cross sections:



Hold down the Ctrl key while pointing to the next edge to select multiple contiguous edges:



Click the edge to select it:

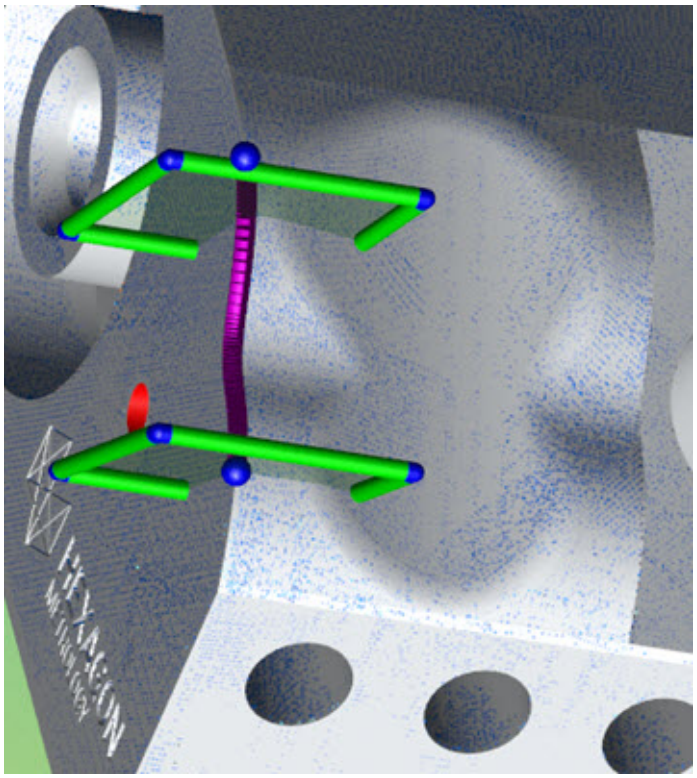


Select as many edges in this manner as required.

To de-select an edge, press the Ctrl key, point to the first or last edge (it turns red) and then left-click it.

To de-select all edges, click the **Reset** button.

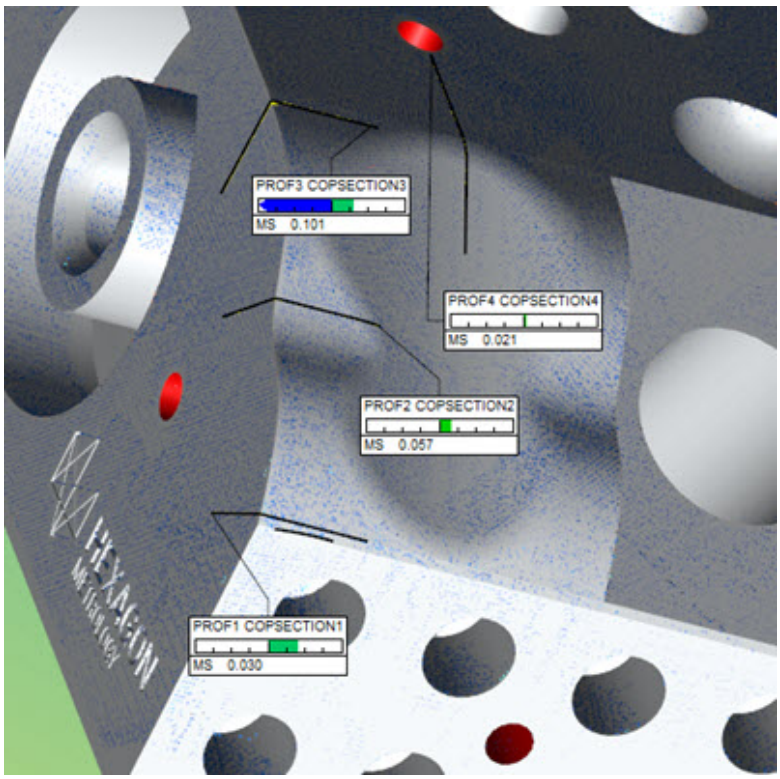
6. Drag the **Start** or **End** points (the blue ball handles) of the curve length line (the purple line) to define only a portion of the curve. If the updated section is too short, click the **Reset** button to cancel and repeat from step 3.



Curve length line (purple line) with Start and End points (blue ball handles)

The dialog box values update automatically when you make changes to the **Start** or **End** points of the defined cross section.

7. When you are done, click **Apply** to create the polylines. Click **Create** to generate the cross sections in the Edit window.



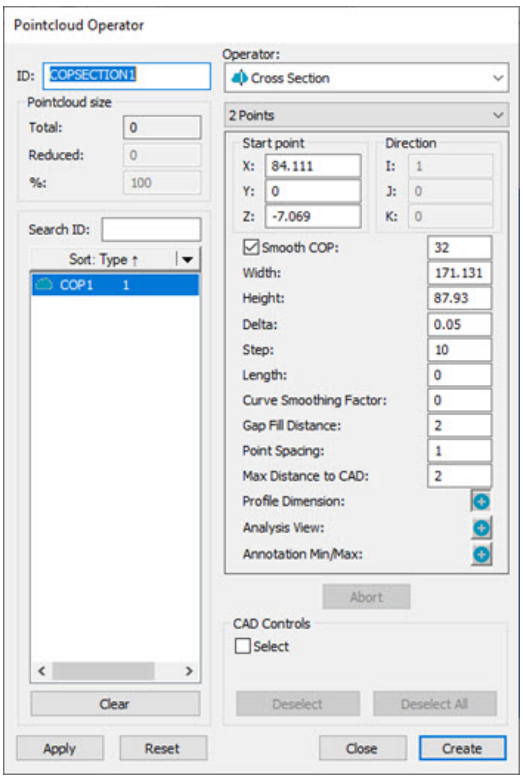
The black polylines represent the nominal CAD. The yellow polylines represent the measured polyline.

Smoothing the Cross Section along the Curve

You can smooth the cross section that PC-DMIS created along a curve with the **Curve Smoothing Factor** option in the **Pointcloud Operator** or **Mesh Operator** dialog box. For details, see the "**Curve Smoothing Factor**" description in the "Cross Section" topic.

Creating a Cross Section between 2 Points

You can create a cross section between two points with the **2 Points** function from the **Pointcloud Operator** or **Mesh Operator** dialog box.



Pointcloud Operator dialog box - Cross Section Operator, 2 Points function selected

PC-DMIS creates the 2 Points cross section between two selected points and orients it normal to the current Graphic View. The cross section's purple **Length** line is perpendicular to the line defined by the two points you select. It is created at the midpoint of this line and defaults to 0 (zero).

To create a cross section between two points:

1. For cross sections created with a COP as the input, click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.

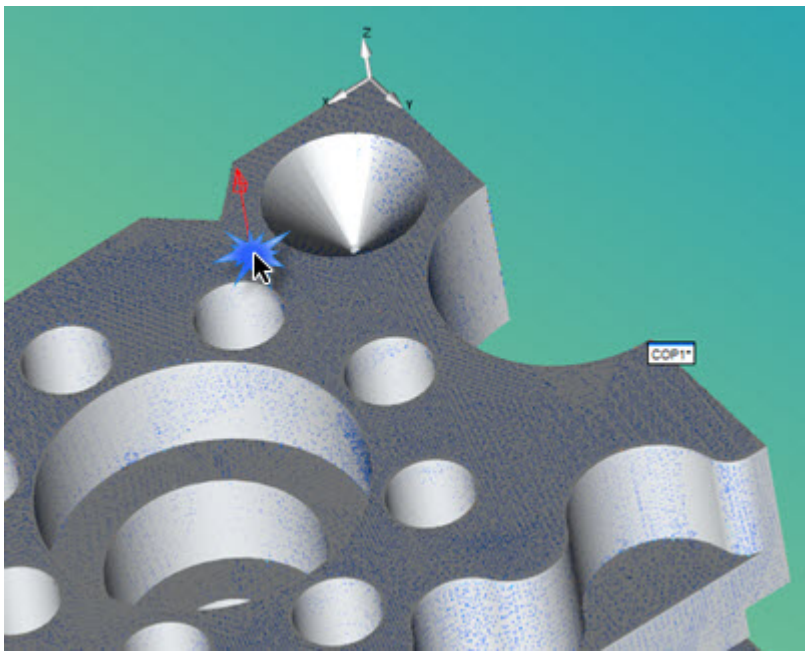
   For cross sections created with a Mesh as the input, click **Insert | Mesh | Operator** to display the **Mesh Operator** dialog box.

2. From the **Operator** list, select the **Cross Section** operator.

3. Under the **Operator** list, from the drop-down list, select the **2 Points** function.

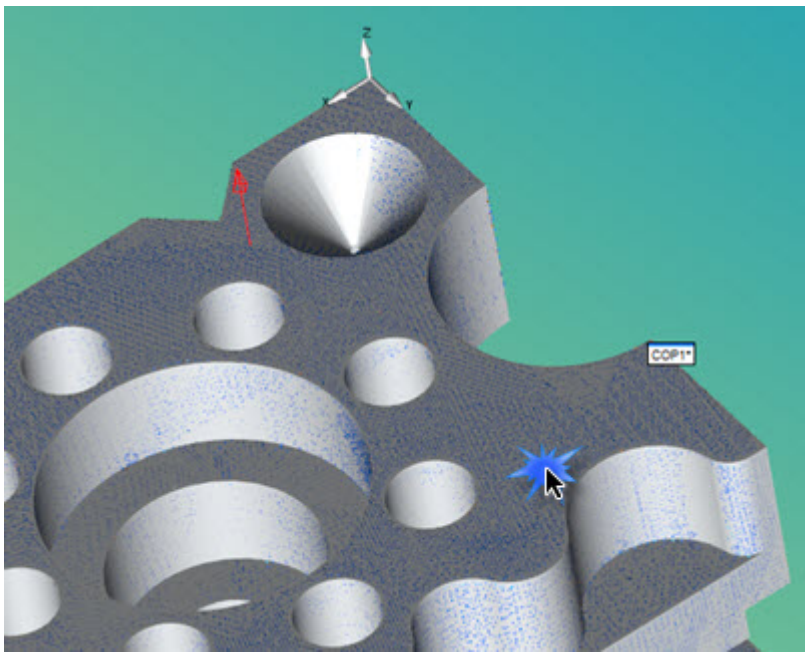
4. From the **QuickMeasure** or **Graphic View** toolbar, select the correct Graphic View for the cross section orientation. For details on the **QuickMeasure** toolbar, see "QuickMeasure Toolbar" in the PC-DMIS CMM documentation. For details on the **Graphic View** toolbar, see "Graphic View Toolbar" in the "Using Toolbars" section of the PC-DMIS Core documentation.

5. From the Graphic Display window, click where you want to define the cross section's first point:

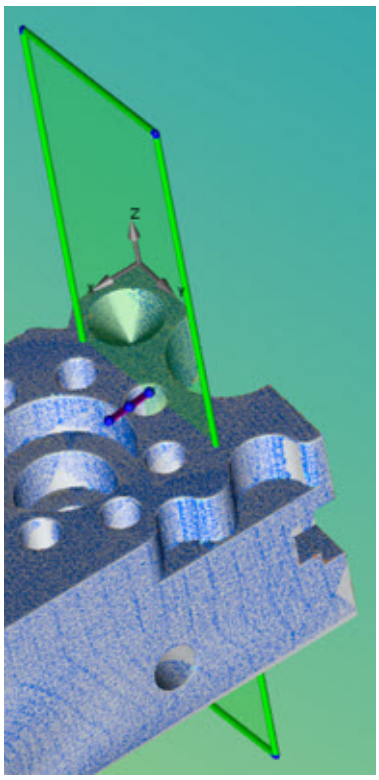


The point's vector appears as a red arrow normal to the selected surface.

6. From the Graphic Display window, click where you want to define the cross section's second point:



Once you select the second point, PC-DMIS displays the cross section:



7. Adjust the cross section's properties as needed.

Show and Hide Cross Section Polylines

You can show or hide cross section features that have been created.

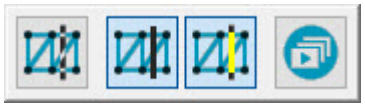
Show or Hide Cross Section Polylines from the Mesh, Pointcloud, or QuickCloud Toolbar

To show or hide cross section polylines:

1. From the **Mesh**, **Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar:



Pointcloud cross section drop-down toolbar



Mesh cross section drop-down toolbar

2. Click the **2D Section Slide Show** button  to display the 2D view of the cross sections in the Graphic Display window.

3. From the floating **Cross Section Graphic Control** toolbar in the Graphic Display window, click the appropriate button to perform the described action:



Show/Hide nominal polylines button - Click to hide or show the black nominal polylines.



Show/Hide measured polylines button - Click to hide or show the yellow measured polylines.

Cross Section Slide Show

The **2D Section Slide Show** button enables the floating **Cross Section Graphic Control** toolbar in the Graphic Display window. From the floating toolbar, use the **Show the previous 2D Section** and **Show the next 2D Section** buttons to display each cross section in the respective order. You can tell that the cross section slide show is enabled when the button appears to be pressed in  .



If the measurement routine contains both COPSECTIONS and MESHSECTIONS, the **Show the next 2D Section** and **Show the previous 2D Section** buttons allow you to navigate to the next section, either pointcloud or mesh.

Once you enable the cross section slide show, from the floating toolbar, click **Show the previous 2D Section** and **Show the next 2D Section** to display individual cross sections in 2D view (Show Only view):

1. From the **QuickCloud** toolbar, click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.

2. Click the **2D Section Slide Show** button. The software displays a 2D view of the cross section and the **Cross Section Graphic Control** floating toolbar. You can reposition the toolbar anywhere in the Graphic Display window. The floating toolbar contains these buttons you can use to navigate through each cross section in 2D view in the Graphic Display window:



Show the previous 2D Section - Click to display the cross section *before* the currently selected one in the Edit window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle backwards until you reach the first cross section.

> If you do not select a cross section, the software selects the first one above the current cursor position in the Edit window. Consequently, nothing happens if there are no cross sections defined above the current cursor position. The same occurs if you select the *first* cross section in the list and you click this button.



Show the next 2D Section - Click to display the cross section *after* the currently selected one in the Edit window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle forward until you reach the last cross section.

> If you do not select a cross section, the software selects the first one below the current cursor position in the Edit window. Consequently, nothing happens if there are no cross sections defined below the current cursor position. The same occurs if you select the *last* cross section in the list and you click this button.

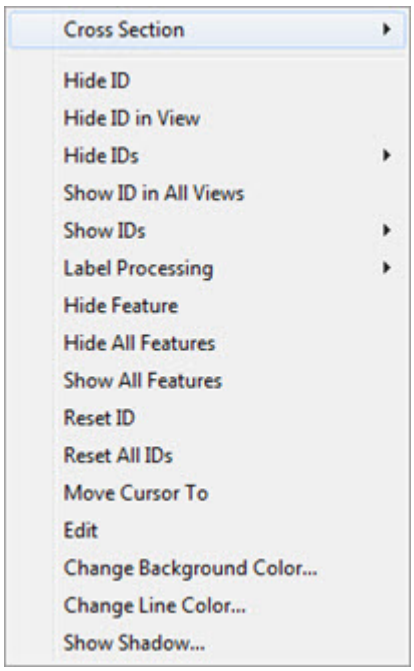
For details on the **Cross Section Graphic Control** floating toolbar, see "2D View of Cross Sections".

3. Click the **2D Section Slide Show** button a second time to exit the slide show and get the CAD graphic (3D view) back.

Show or Hide Cross Section Polylines from the Graphic Display Window

To hide cross section polylines from the Graphic Display window:

1. Right-click any cross section label in the Graphic Display window to display the pop-up menu.



2. Hover over the **Cross Section** option to display the **Cross Section** menu.

   If the measured and nominal cross section polylines are visible, the **Cross Section** menu has these options:

   If the measured and nominal cross section polylines are NOT visible, the **Cross Section** menu has these options:

   You may also have a mixture of the above options depending on the visible state of the polylines, such as:

3. Click the appropriate option to show or hide the associated polylines.

Measuring Cross Section Distances

Distances can be measured on 2D cross sections in the Graphic Display window. You must already have the cross sections created and be in cross-section 2D view. For details on how to view cross sections in 2D view, see "Show and Hide Cross Section Polylines".

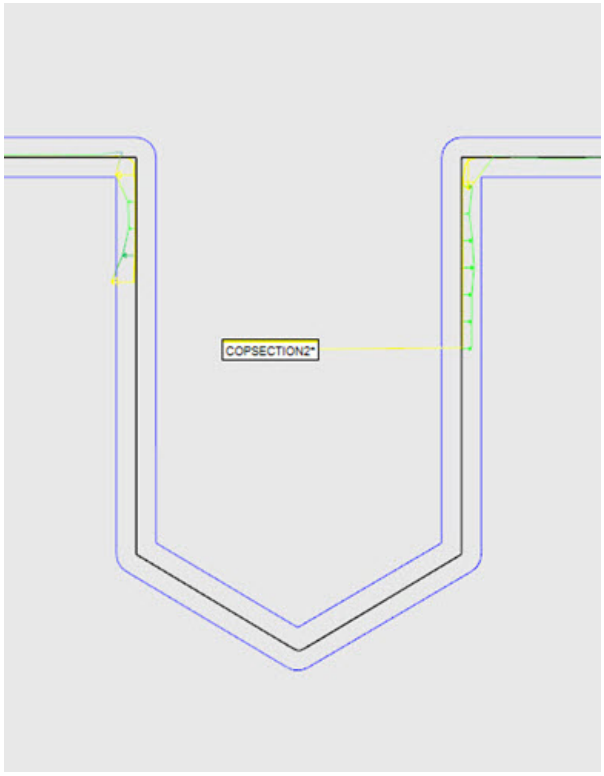
To create a cross-section Distance Gage:

1. From the **Pointcloud**, **QuickCloud**, or **Mesh** toolbar (**View | Toolbars**), click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.

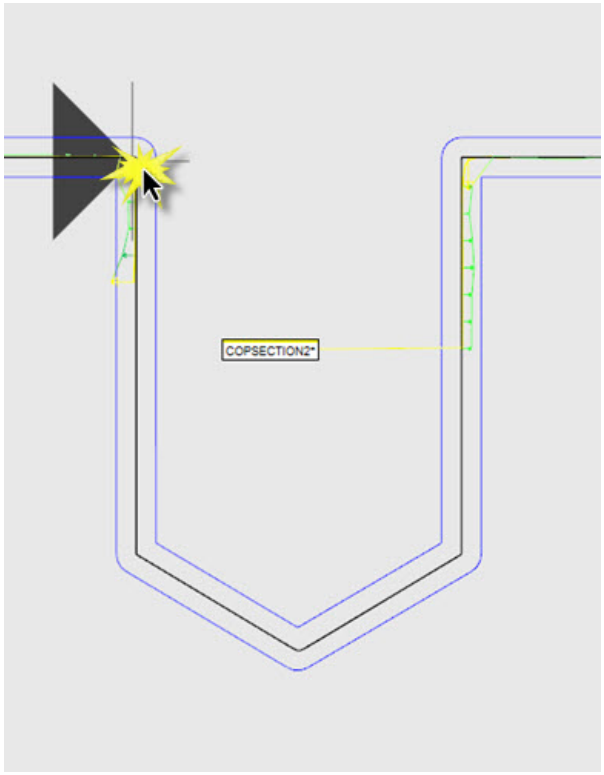


2. Click the **2D Section Slide Show** button ( ) to enter 2D view.

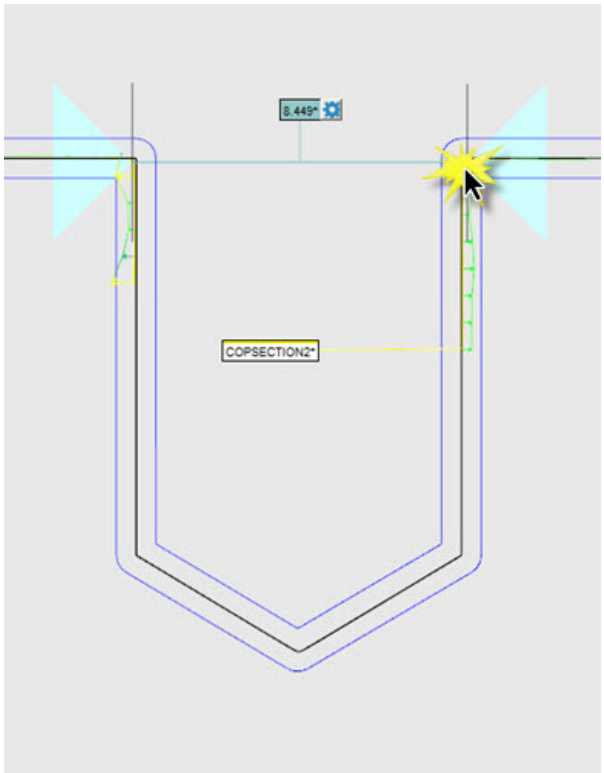
3. Click the **Show the previous 2D Section** or **Show the next 2D Section** button until the cross section is displayed in the Graphic Display window.



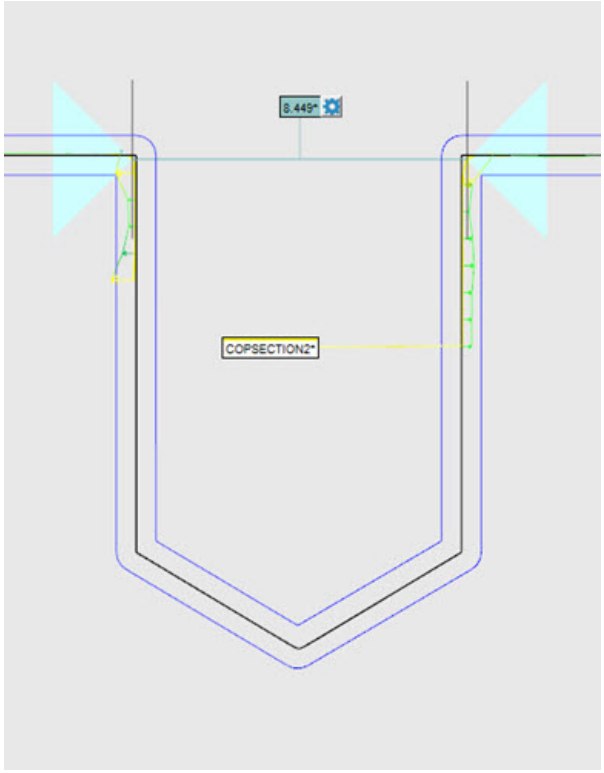
4. In the Graphic Display window, hover over the cross section, and then click and drag to display the start point.



5. Drag the cursor to the end point, and click to select it. The Distance Gage is calculated, created, and displayed in the 2D view with its associated label.



As you drag the cursor, the software intuitively detects if the start and end points are along an axis. If they are, the direction is recognized and constrained parallel to that axis.



Parallel Distance Gage example

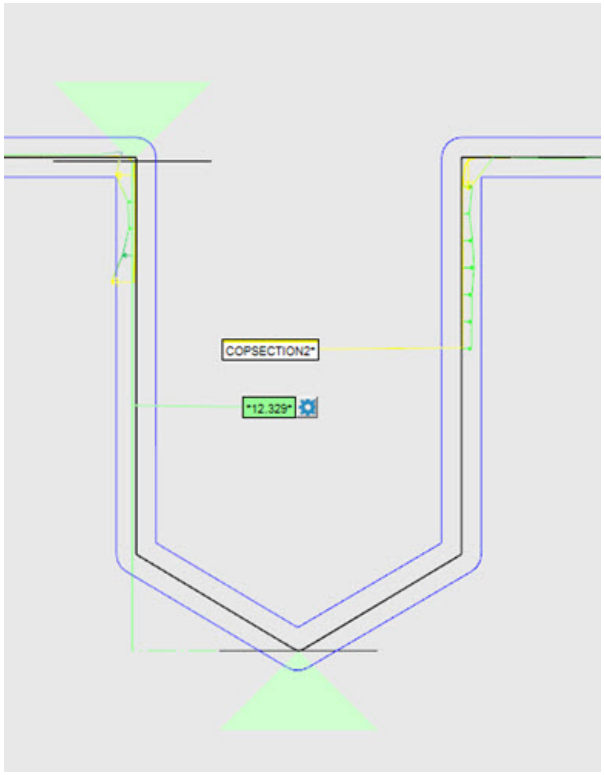
To create a Distance Gage parallel to the first side picked:

a. Press and hold the Shift key.

b. Click the start point, and drag and click the end point.

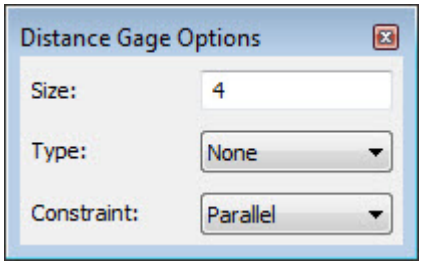
An example of this would be if the cross section was not created along the X, Y, or Z axis.

If the start and end points are offset from one side to another, the axis direction is still recognized. The distance, however, is calculated parallel but between the offset points.

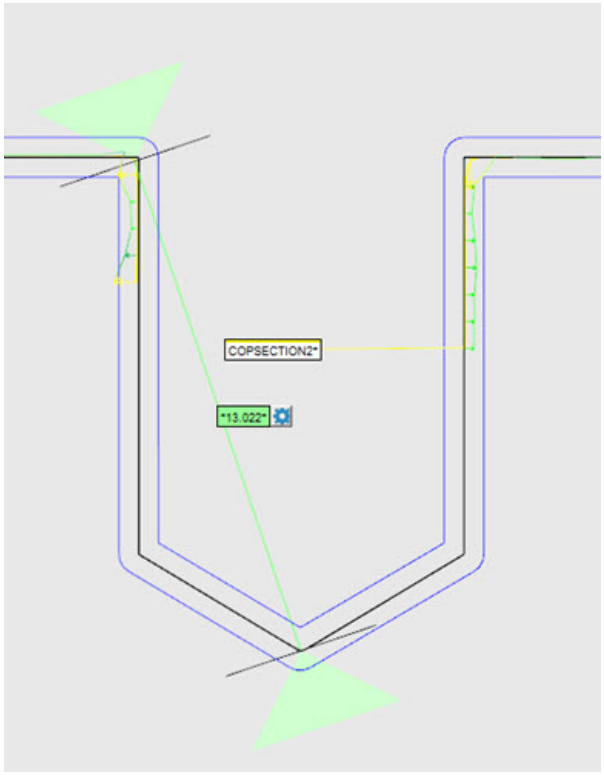


Offset Distance Gage example

6. To change the properties of the Distance Gage, click the **Distance Gage Options** button () on the label. The **Distance Gage Options** dialog box appears.



For example, if you do not want the Distance Gage calculated as an offset calculation, select the **Parallel** option from the **Constraint** list. Click the start and end points as before; the Distance Gage is calculated between the two points.



Example of a Distance Gage calculated with the Parallel Constraint option selected

7. Edit the properties of the Distance Gage:

   Size - If the **None** option is selected in the **Type** list, the **Size** value is used to determine the size of the start and end point icons in the Graphic Display window. If either the **Best Fit**, **Max Fit**, or **Min Fit** option is selected from the **Type** list, the **Size** value is used as described below. A size of 4 is the default.

   Type - Click the drop-down arrow to display these options:

   - **None** (default) - A Point-to-Point distance calculation between the closest cross section polyline points based on the selected start and end points.
   - **Best Fit** - A Least Square line is calculated based on all the yellow points within the first pick zone, defined by the **Size** value (the default is 4) and the selected start point. This is repeated for the second pick zone, defined by the **Size** value and the selected end point. The centroid of the first Least Square line is projected onto the measurement zone line. This is repeated for the centroid of the second Least Square line. The distance is between these two projected points.
   - **Max Fit** - Defined by the point farthest out in the first pick zone, defined by the **Size** value and the selected start point, and by the point farthest out in the second pick zone, defined by the **Size** value and the selected end point. The Max Fit points are projected onto the measurement zone line. The Max distance is between these two projected points.

- **Min Fit** - Defined by the closest point in the first pick zone, defined by the **Size** value and the selected start point, and in the second pick zone, defined by the **Size** value and the selected end point. The Min Fit points are projected onto the measurement zone line. The Min distance is between these two projected points.

If the **Type** option is changed, the measured distance is automatically recalculated, and the updated value is displayed based on the option selected.

Constraint - Select **None** (default) if you do not want to constrain it to any axis. Select the appropriate option to constrain the Distance Gage to the **X**, **Y**, or **Z** axis, or **Parallel** to calculate the distance parallel to the first selected side.

Creating a Distance Gage with and without Measured Points

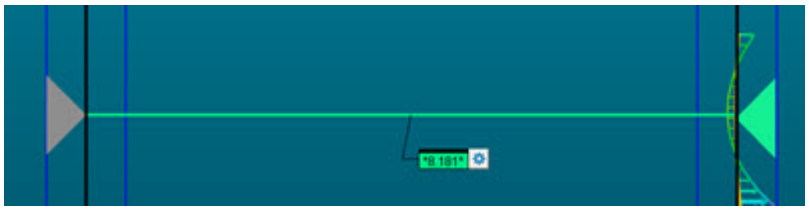
You can create a distance gage with or without measured points on either side of the gage as described below.

Case #1



Distance Gage created using measured points on both sides (indicated by colored arrows)

Case #2



Distance Gage created using measured points only on one side

In this case, PC-DMIS precedes the distance value with an asterisk. This indicates that one or more sides are not measured. The value shows the distance between the nominal (gray arrow side) and the measured side.

Case #3



Distance Gage created with no measured points on either side (gray arrows)

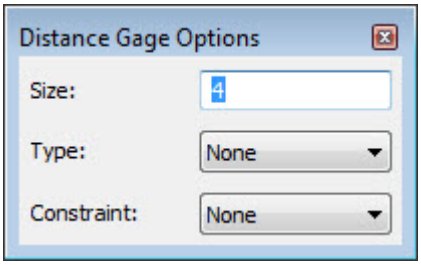
In this case, the distance gage shows the nominal value.

Creating a 3D Distance Gage

To create a 3D Distance Gage that is not constrained to any axis:

1. Press and hold the Ctrl key, hover over the cross section in the Graphic Display window, and click and drag to display the start point.
2. Continue to drag the cursor with the Ctrl key pressed to the end point location.
3. Click to select the end point and display the Distance Gage and its associated label.

The same functionality is available as described earlier for 2D Distance Gages. Click the **Distance Gage Options** button to view the **Distance Gage Options** dialog box. The **Constraint** option is set to **None**.

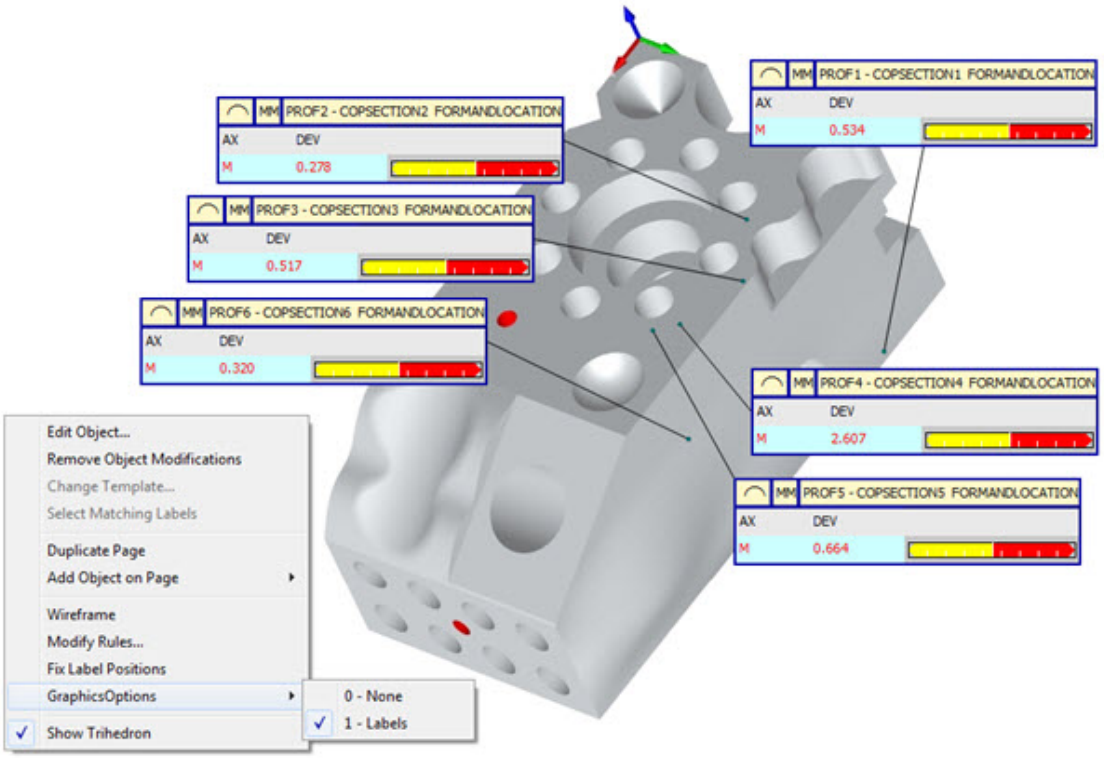


Viewing Cross Section Labels in Reports

You can view cross section Annotation and Distance Gage labels in reports in two ways:

Viewing labels from a report template that has a graphic image

1. From any report template that has a graphic image, right-click the image to display a pop-up menu.



2. Click **GraphicOptions** and then **1 - Labels** to display all the labels in your report. Click **0 - None** to hide all the labels.

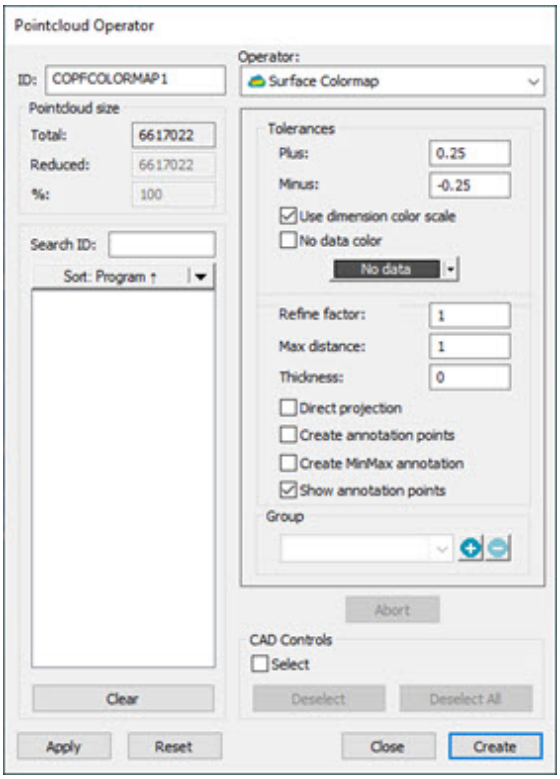
Viewing labels in the Report Graphical Analysis template from the Cross Section dialog box

1. Create the **Annotations** and **Distance Gage** items for your cross sections. For details on creating **Annotations**, see the "Cross Section" help topic. For details on creating **Distance Gage** items, see the "Measuring Cross Section Distance" help topic.

2. Create the Analysis View. For details on the `Analysis View` command, see the "Analysis View" description in the "Cross Section" help topic.

3. Click the **Graphical Analysis** option in the Report window (**View | Report**). The annotation and gage labels are automatically visible.

SURFACE COLORMAP



Pointcloud Operator dialog box - Surface Colormap Operator

The SURFACE COLORMAP operation applies a colored shading to the CAD model. PC-DMIS shades the model according to the deviations of the pointcloud compared to CAD. The model uses the colors from the **Edit Dimension Colors** dialog box (**Edit | Graphic Display Window | Dimension Colors**) and the tolerance limits from the **Upper tolerance** and **Lower tolerance** boxes. These two boxes are discussed below.

Show / Hide Colormaps

You can show or hide colormaps in the Graphic Display window in different ways. When hidden, PC-DMIS does not show the colormaps in the Graphic Display window as you move through the Edit window.

The **Activate Colormap** button has two states: Enabled and Disabled. From the **Graphics Items** toolbar or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**), click the **Activate Colormaps** button ( ) so it's in the enabled state ( ). PC-DMIS now actively shows colormaps in the Graphic Display window.

To hide the colormaps in the Graphic Display window, click the **Activate Colormaps** button again so it's in the disabled state ( ). You can also select **None** from the **Colormaps** list to disable colormaps.

To show the colormaps:

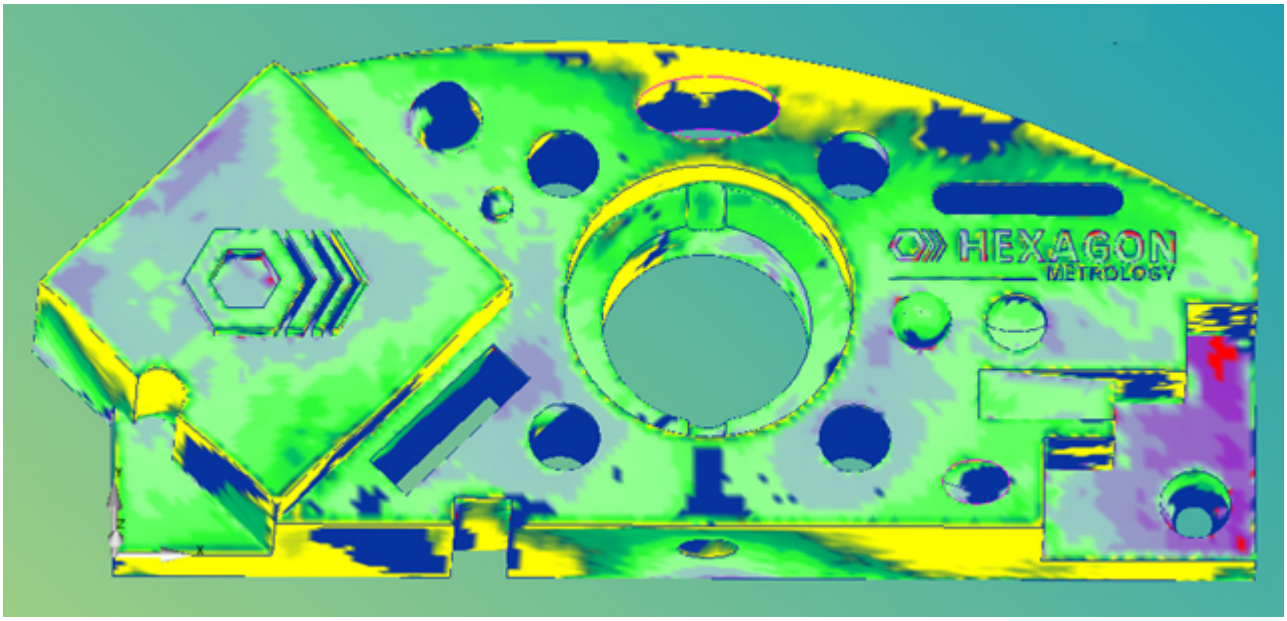
- Click the **Activate Colormaps** button so it's in the enabled state. When you enable this button, PC-DMIS displays the colormaps in the Graphic Display window based on the cursor position in the Edit window.
- Select a colormap from the **Colormaps** list.
- When you apply or execute a colormap, PC-DMIS automatically sets the **Activate Colormaps** button to the enabled state.

When the cursor is on a Mesh, Point, Surface or Thickness colormap in the Edit window, the active colormap appears in the Graphic Display window. PC-DMIS also displays the **Colormap ID** in the **Colormap** combo box.

If your cursor is above all the colormaps in the Edit window, PC-DMIS does not show any colormap in the Graphic Display window, and sets the **Colormap** combo box to **None**.

To view the color scale from the Dimension Colors Bar, select **View | Other Windows | Dimension Colors**.

To apply the SURFACE COLORMAP operation to a Pointcloud, on the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**), click the **Pointcloud Surface Colormap** button ( ), or select **Insert | Pointcloud | Surface Colormap**.

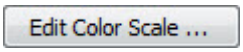


Example of a Surface Colormap applied to selected CAD elements

The SURFACE COLORMAP operator has these options:

Tolerances - Use this option to set the upper (**Plus**) and lower (**Minus**) tolerance values:

Use dimension color scale check box - When you select this check box, the software defines the color bar used for the Surface Colormap color properties by the **Dimension Colors Bar**. For details on the **Dimension Colors Bar**, see "Using the Dimension Colors Window (Dimension Colors Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

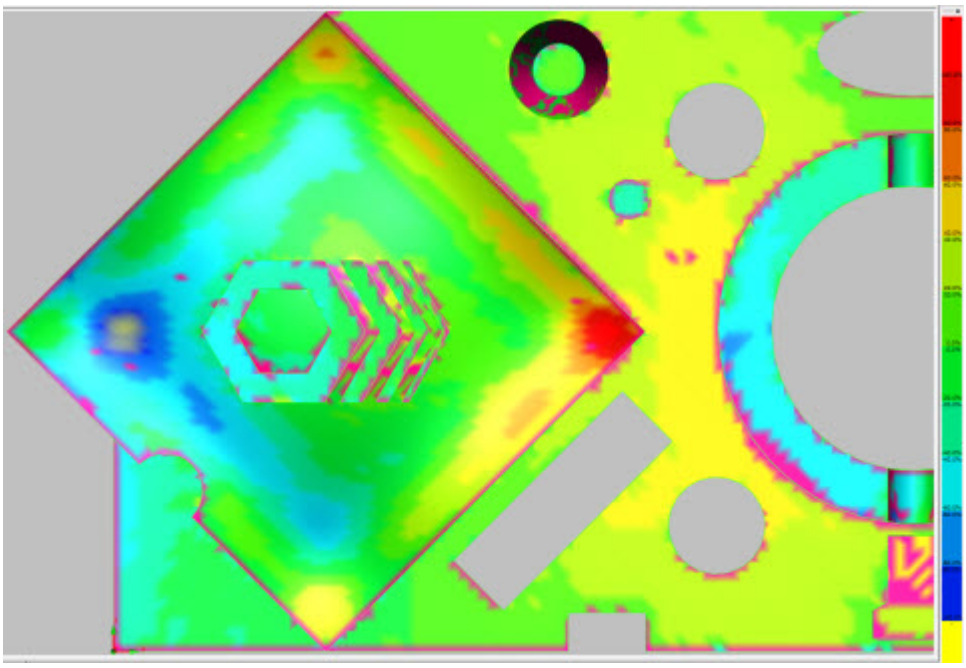


Edit Color Scale - When you clear the **Use dimension color scale** check box, PC-DMIS enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale, and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

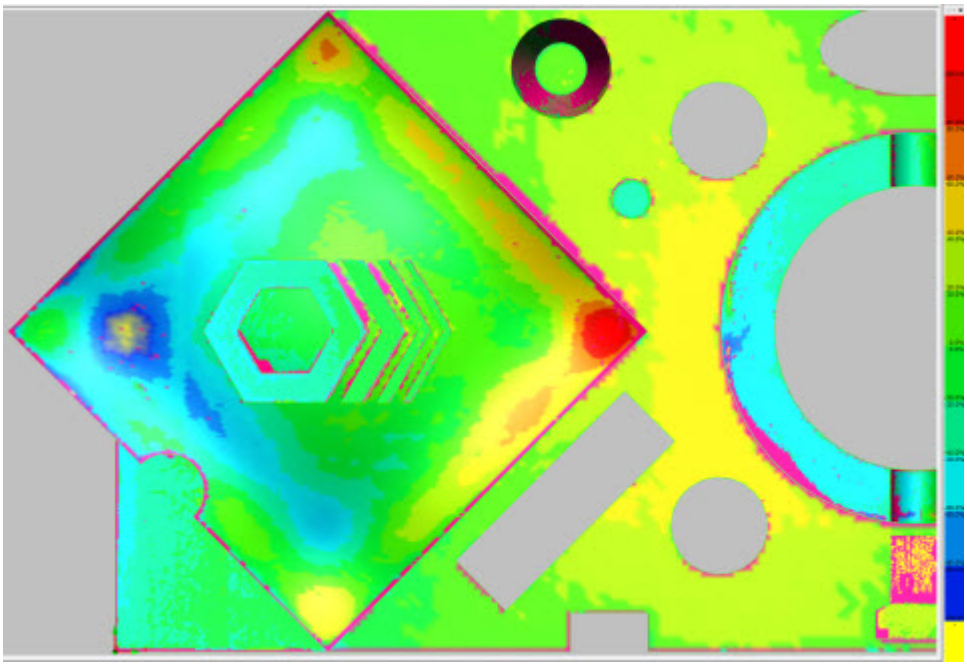
No data color check box - When you select this check box, the software maps the selected color to areas on the selected surfaces where no data is found.

Refine Factor - This option adjusts the accuracy of the Surface colormap. If you change this value, PC-DMIS draws a new and changed colormap. The underlying measured data does not change. The colormap tessellates the CAD model with an overlay of colored triangles. The vertices of each triangle are colored with the color that corresponds to its deviation from the pointcloud. The colors are taken from the dimensions color scale discussed above. You can use a smaller or larger refine factor value to respectively generate a finer or coarser tessellation. You may want to decrease the refine factor to obtain a smoothly shaded CAD with a more accurate deviation representation. However, a smaller refine value results in a larger number of triangles, thereby increasing the computation time and the size of the CAD model. For comparison, note that the number of triangles for a refine factor of 0.5, compared to a refine factor of 1.0, is about 4 times more; whereas a refine factor of 0.1 compared to 1.0, is about 100 times more.

Pointcloud COLORMAP example with a Refine Factor of 1:



Pointcloud COLORMAP example with a Refine Factor of 0.1:



Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

Thickness - This option adds a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

Direct projection - When you select this check box, PC-DMIS projects all the points of the pointcloud onto the tessellated CAD model. With the standard method, where the **Direct projection** check box is not selected, PC-DMIS searches for the closest point at each triangle vertex within a default angle.

PC-DMIS then computes the deviations for each tessellated triangle and colors the CAD model accordingly.

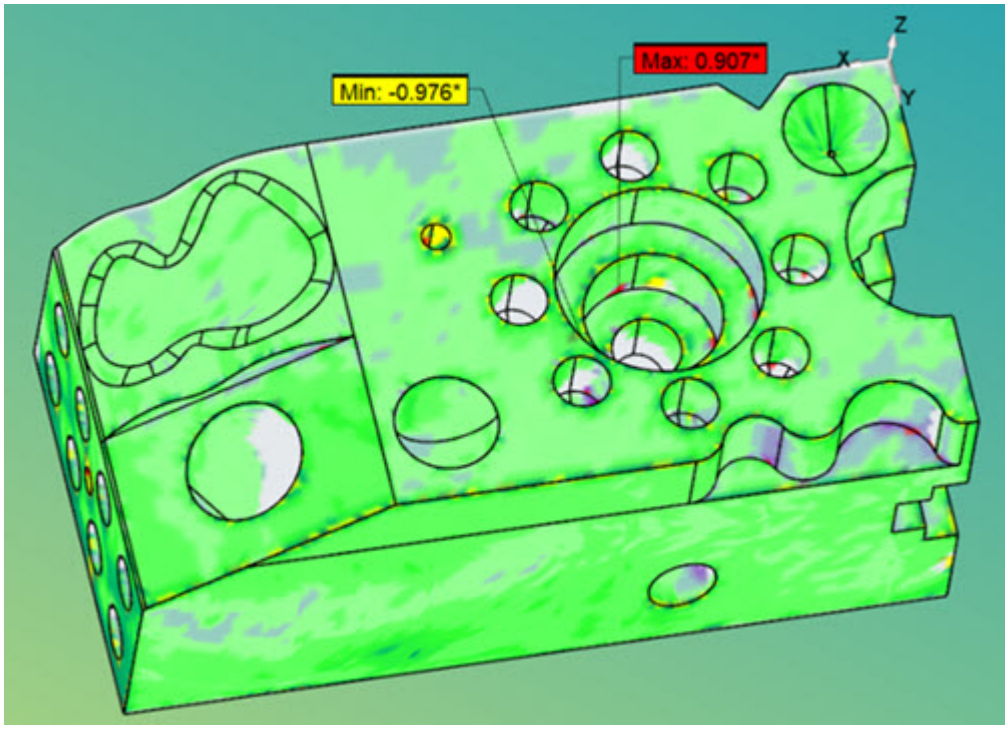
Create annotation points check box - Annotations are a way to display the deviation for a specific location on a surface colormap with its associated color. To create an annotation:

1. Select the **Create annotation points** check box. This clears the **Select** check box in the **CAD Controls** area and disables most of the options on the right side of the dialog box.

2. Select a point on the CAD surface in the Graphic Display window. PC-DMIS evaluates and creates an annotation label in the same background color as the COP deviation point with the deviation value. You can move the label around in the Graphic Display window as any other label.

> Once created, if you restart the measurement routine, or if you restart PC-DMIS and you reload the same measurement routine, the annotation labels remain in the same position and have the same characteristics.

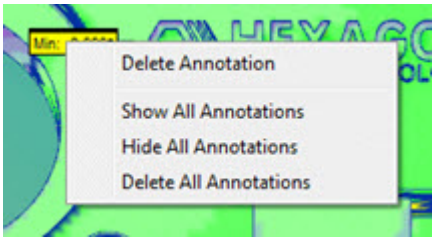
Create MinMax annotations check box - When you select this check box, the software creates the minimum and maximum values and displays them as annotation labels for the active COP Surface Colormap.



PC-DMIS calculates the minimum and maximum points each time you execute the measurement routine.

Show, Hide, or Delete Annotation Labels

To show, hide, or delete annotation labels, right-click one to display the pop-up menu and then select the appropriate option.



Delete Annotation - The software deletes the selected annotation label.

Show All Annotations - The software displays all annotation labels.

Hide All Annotations - The software hides all annotation labels.

Delete All Annotations - The software deletes all annotation labels.

Show annotation points check box - When you select this check box, the software displays all annotation points.

Group - You can use this option to create, modify, or identify Surface Colormap groups. For details, see "Method 2" in the "Apply COLORMAP to a CAD model with Multiple Surface Profile Tolerances" topic.

Click **Abort** to undo any calculations that were generated after you click the **Apply** button.

CAD Controls - This option lets you apply the operation to the CAD elements you select. For details, see the scan area of the "CAD Controls" topic.

Click **Create** to insert a `COP/OPER,SURFACE COLORMAP` command into the Edit window as shown below.

```
COPFCOLMAP2=COP/OPER,SURFACE COLORMAP,PLUS
TOLERANCE=0.25,MINUS TOLERANCE=-0.25,THICKNESS=0

    REF,COP1,,
```

Colormaps in the Report

For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

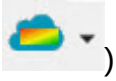
Apply COLORMAP to a CAD model with Multiple Surface Profile Tolerances

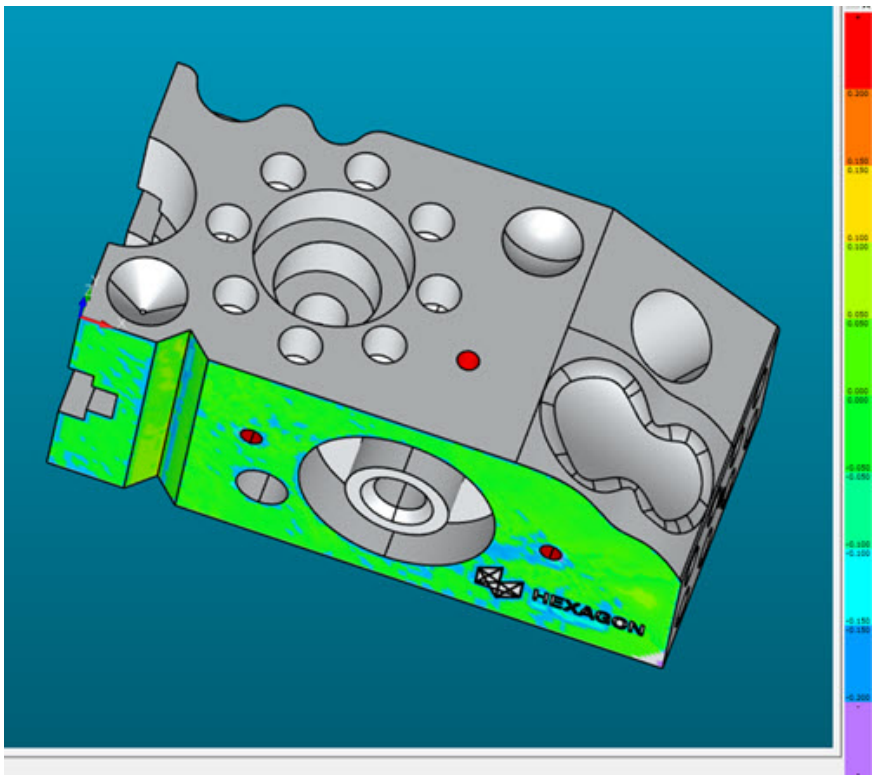
There are two methods to apply a Surface Colormap when the CAD model has multiple surface-profile tolerances.

Method 1

Create multiple Surface Colormaps, one for each tolerance or Surface Profile.

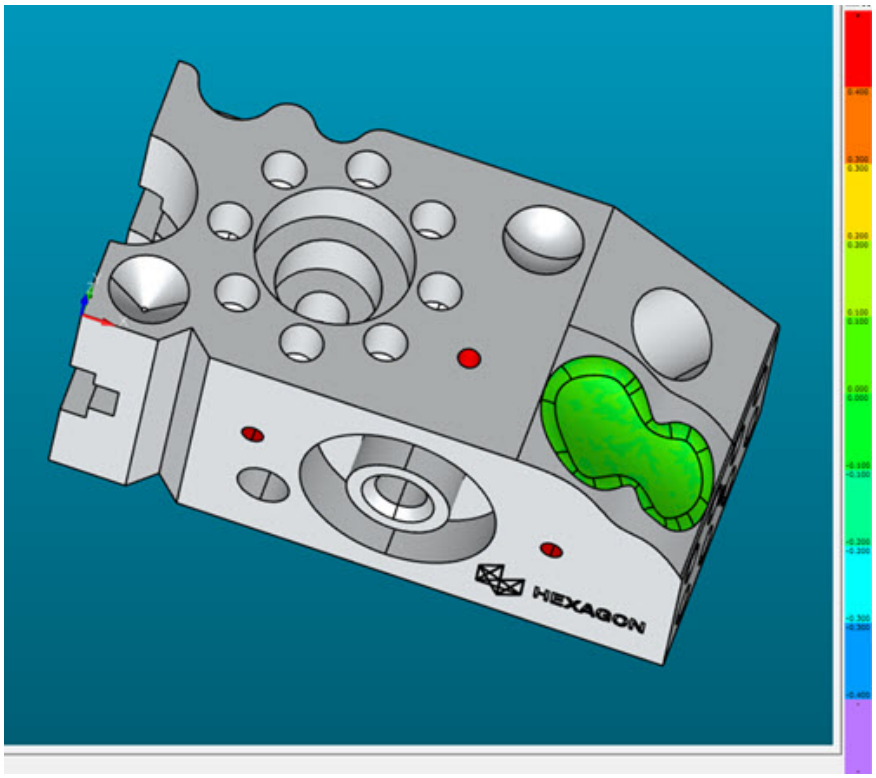
To create multiple Surface Colormaps, do the following:

1. From the **Pointcloud** toolbar, select the **Pointcloud Surface Colormap** button (). The **Pointcloud Operator** dialog box for the Surface Colormap appears.
2. Enter the tolerances.
3. Select the specific CAD surfaces. For details on selecting CAD surfaces, see "Working with CAD Surfaces" in the "Scanning Your Part" chapter in the PC-DMIS Core documentation.
4. Click **Apply** to apply the Surface Colormap to the selected CAD surface.



Example of the Surface Colormap applied to the first-selected CAD surfaces

5. Click **Create** to add the Surface Colormap in the Edit window.
6. Create a second Surface Colormap in the same manner for the next Surface Profile.

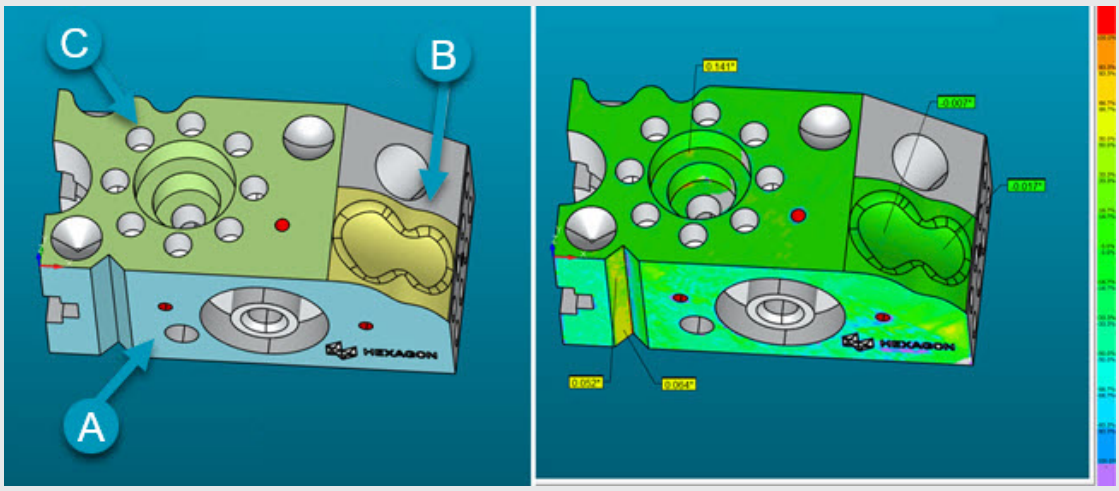


Example of the second Surface Colormap applied to the selected CAD surfaces

Method 2

You can create groups of selected CAD surfaces within a single colormap. Each group can have different tolerances and Surface Colormap parameters (Refine factor, Max distance, and Thickness). If the Surface Colormap has two or more groups, the software displays the color scale with percentages.

In this example, you can see that the left image shows three different grouped CAD surfaces. The image on the right shows surface color maps applied to each group, using the percentage of tolerances:



(A) - Group01 TOL +/- 0.1 mm

(B) - Group02 TOL +/- 0.2 mm

(C) - Group03 TOL +/- 0.3 mm

To create groups and apply different tolerances to the selected CAD surfaces within one colormap, do the following:

1. From the **Pointcloud** toolbar, select the **Pointcloud Surface Colormap** button ( ). The **Pointcloud Operator** dialog box for the Surface Colormap appears.
2. Enter the tolerance values and colormap parameters (**Refine factor**, **Max distance**, and other parameters).
3. From the **Pointcloud Operator** dialog box, select the **Select** check box in the **CAD Controls** area.
4. Click each of the CAD surfaces to be grouped. PC-DMIS highlights the surfaces with the group color as you click them. Click the **Deselect** button to remove the last highlighted surface from the group.

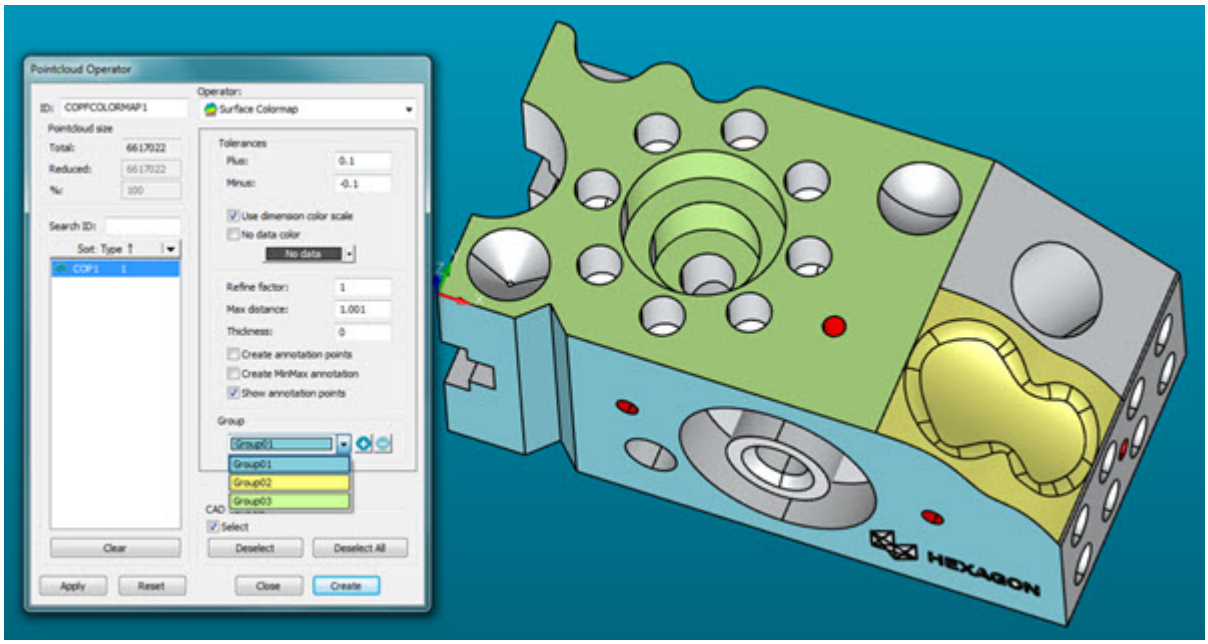
5. To group the selected (highlighted) surfaces, click the **Add a new data group** (**+**) button located to the right of the **Group** list.

   This group remains the active group until you create a new group. PC-DMIS applies any changes that you make to the tolerances or COLORMAP parameters to the active group. Also, if you select additional surfaces, PC-DMIS adds them to the active group.

   To identify which surface belongs to which group, PC-DMIS highlights the selected CAD surfaces with the group color. To identify which group a grouped surface belongs to, press and hold the Shift key and then left-click the surface. The **Group** list updates to show the group to which it's assigned.

   If you click a CAD surface that is not in the active group, PC-DMIS removes it from the group it is currently assigned to, and then adds it to the active group.

6. To create another group, click the **Add a new data group** (**+**) button again, click the surfaces on the CAD, and update the tolerances and any COLORMAP parameters as necessary. Repeat to create more groups.



Example of grouped CAD surfaces

7. To make changes to a group, select it from the **Group** list and make the necessary changes.

8. To delete a group, select it from the **Group** list and click the **Remove the current data group** (**-**) button.

> When a COLORMAP contains two or more groups with different tolerances, the color scale is automatically set to show the deviations with percentages.

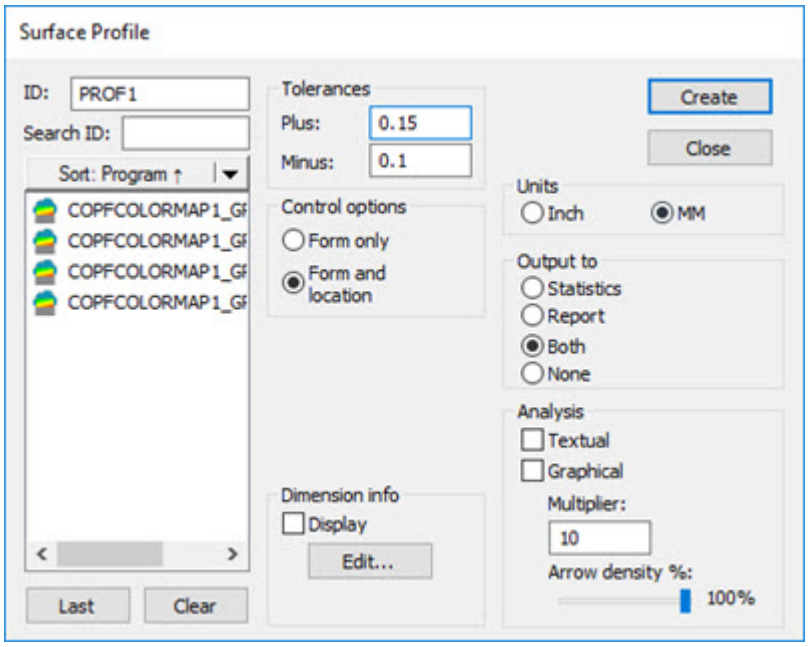
Dimensioning Surface Profile using Pointcloud Colormap with Groups

You can use Pointcloud COLORMAP Groups to dimension Surface Profiles if you have the **Use Legacy Dimensions** option (**Insert | Dimension**) active. PC-DMIS does not currently support any other dimensioning method.

1. Create the Pointcloud COLORMAP Groups as described in Method 2.

2. For legacy dimensions, do this:

   Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**). The software displays the **Surface Profile** dialog box for legacy dimensions:

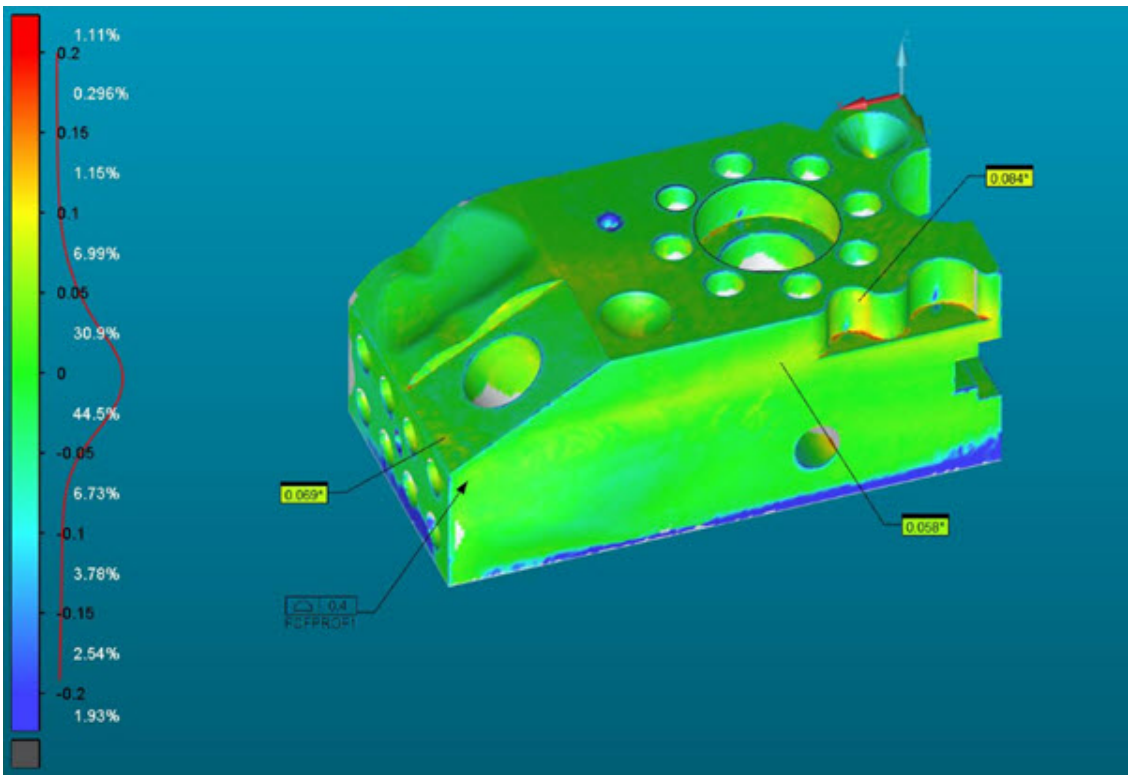


Surface Profile legacy dialog box for pointcloud colormap with groups

   For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

Dimensioning Surface Profile Using the Pointcloud Surface COLORMAP

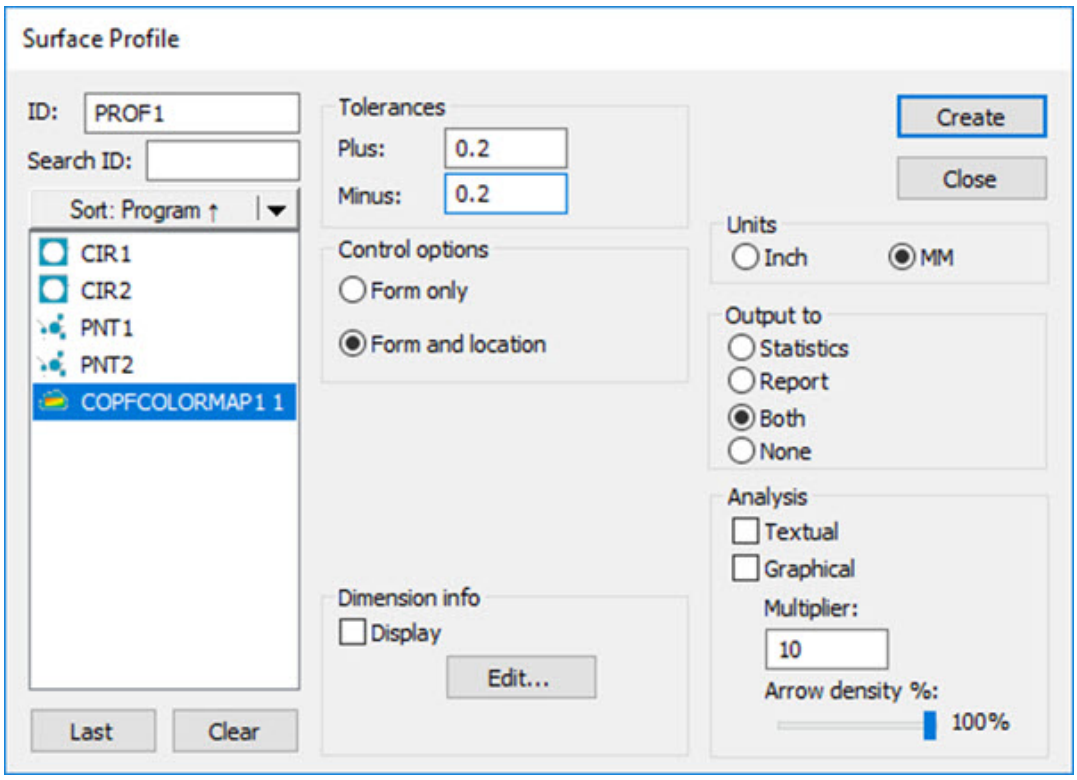
You can use a Pointcloud Surface COLORMAP to create a Dimension Surface Profile.



Example of a Dimension Surface Profile created using a Pointcloud Surface Colormap

To create a Dimension Surface Profile from a Pointcloud Surface COLORMAP:

1. Create a Pointcloud Surface COLORMAP. For details, see "POINT COLORMAP".
2. Make sure you have the **Use Legacy Dimensions** option selected (**Insert | Dimension | Use Legacy Dimensions**).
3. Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**) or select it from the menu (**Insert | Dimension | Profile | Surface**). The **Surface Profile** dialog box opens.

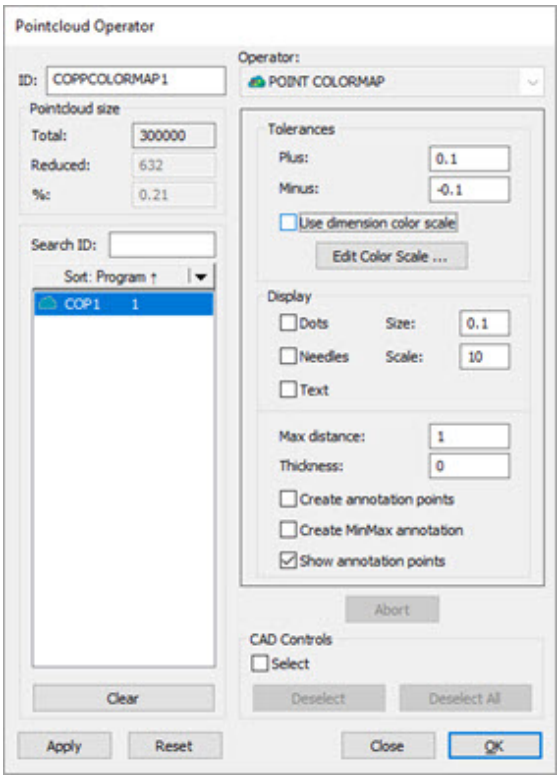


Surface Profile legacy dialog box for Pointcloud Surface Colormap

For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

4. From the **Features** list box, select the desired Pointcloud Surface COLORMAP.
5. Set the other options as needed and then click **Create**.

POINT COLORMAP



Pointcloud Operator dialog box - Point Colormap Operator

The Point Colormap operation evaluates the deviations of the data points contained in a COP command compared to a CAD object. You can use Method 1 to color the points in the COP, or you can choose to represent the deviations by colored dots, colored needles that show the actual deviations, or as numerical values of the deviations with Method 2. You need to specify the plus and minus tolerances, and the scale to use.

Show / Hide Colormaps

You can show or hide colormaps in the Graphic Display window in different ways. When hidden, PC-DMIS does not show the colormaps in the Graphic Display window as you move through the Edit window.

The **Activate Colormap** button has two states: Enabled and Disabled. From the **Graphics Items** toolbar or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**), click the **Activate Colormaps** button ( ) so it's in the enabled state ( ). PC-DMIS now actively shows colormaps in the Graphic Display window.

To hide the colormaps in the Graphic Display window, click the **Activate Colormaps** button again so it's in the disabled state ( ). You can also select **None** from the **Colormaps** list to disable colormaps.

To show the colormaps:

- Click the **Activate Colormaps** button so it's in the enabled state. When you enable this button, PC-DMIS displays the colormaps in the Graphic Display window based on the cursor position in the Edit window.
- Select a colormap from the **Colormaps** list.
- When you apply or execute a colormap, PC-DMIS automatically sets the **Activate Colormaps** button to the enabled state.

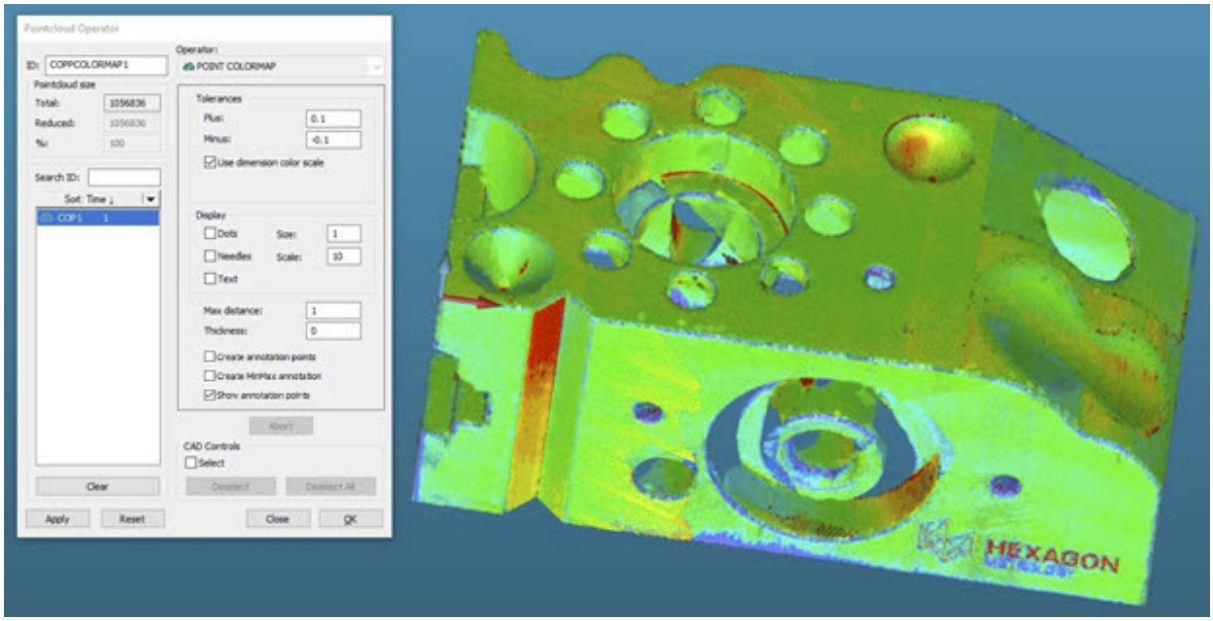
When the cursor is on a Mesh, Point, Surface or Thickness colormap in the Edit window, the active colormap appears in the Graphic Display window. PC-DMIS also displays the **Colormap ID** in the **Colormap** combo box.

If your cursor is above all the colormaps in the Edit window, PC-DMIS does not show any colormap in the Graphic Display window, and sets the **Colormap** combo box to **None**.

You can create a Pointcloud Colormap in two ways:

Method 1: In the **Display** area of the **Pointcloud Operator** dialog box, clear all three of the check boxes (**Dots**, **Needles** and **Text**)

With all three of the **Display** check boxes unchecked, PC-DMIS projects the points onto the tessellated CAD model. The software computes the deviations, and then colors the Pointcloud accordingly.



Example of a point colormap using Method 1 (CAD model hidden)

This method allows you to create annotation points as well. For details on the annotation-related check boxes in the **Pointcloud Operator** dialog box, see the

appropriate description in the "Surface Colormap" topic, starting with the "Create annotation points" check box description.

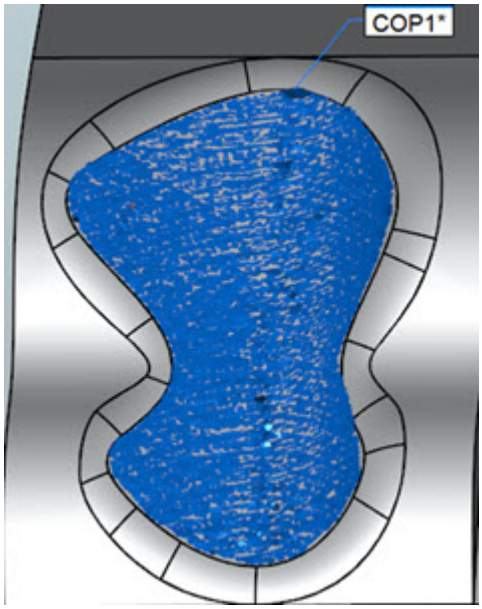
Method 2: In the **Display** area of the **Pointcloud Operator** dialog box, select any of the three check boxes.

When you select any or all of the **Display** check boxes, PC-DMIS projects the points onto the actual CAD model. The software computes the deviations and then colors the Pointcloud accordingly. This process is more time-consuming and more accurate. This is because the software projects the points onto the actual CAD model instead of on the tessellated CAD model. However, because this operation is more time-consuming, it is best to first filter the pointcloud, or confine it to selected CAD faces.

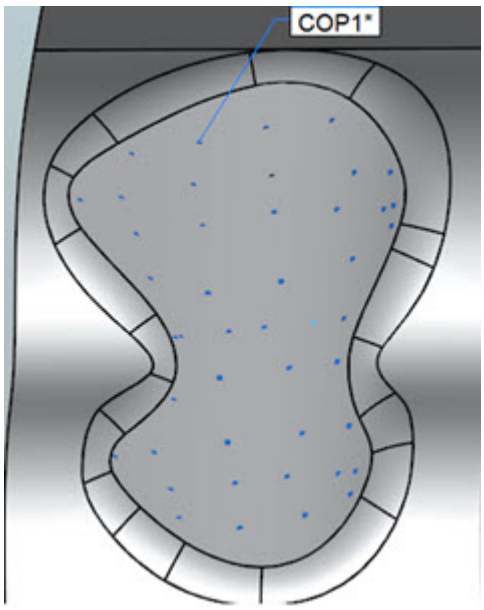
To apply the POINT COLORMAP operation to a pointcloud, on the **Pointcloud** toolbar, click **Pointcloud Point Colormap** (), or from the menu you can select **Insert | Pointcloud | Point Colormap**.

This is the recommended process when you create a point colormap using dots, needles and/or text (Method 2):

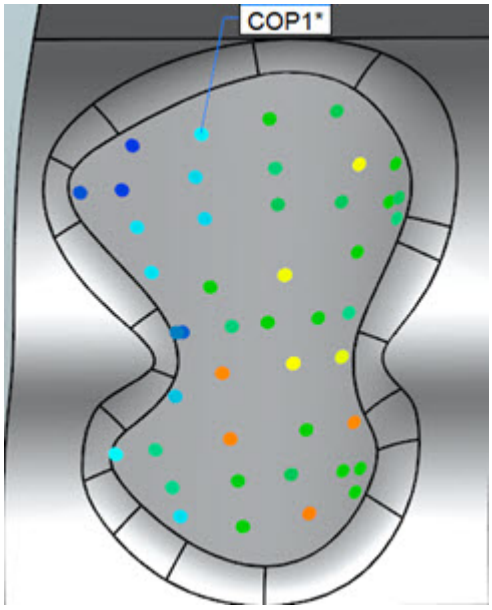
1. Clean the data or select just those surfaces where you want the point colormap.



2. From the **Filter** COP Operator, use the **DISTANCE** type setting to filter the data.



3. Create the point colormap.



Example of the recommended steps to apply a point colormap

The Point Colormap operator has these properties:

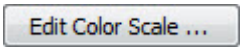
Tolerances - Use this property to set the upper (plus) and lower (minus) tolerance values:

Plus - The upper tolerance value

Minus - The lower tolerance value

Use dimension color scale check box - When you select this check box, the software uses the **Dimension Colors Bar** to define the color bar for the Point Colormap color properties. For details on the **Dimension Colors Bar**, see "Using the

Dimension Colors Window (Dimension Colors Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



Edit Color Bar - If you do not select the **Use dimension color scale** check box, the software enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale, and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

Dots - This option allows the use of colored dots.

Size - This option defines the size of the dots.

Needles - This option allows the use of scaled deviations (using the **Scale** value below) as a colored line segment normal to the CAD.

Scale - This option defines the scale value PC-DMIS uses for the needle representation.

Text - This option defines the numerical value of the deviation.

Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

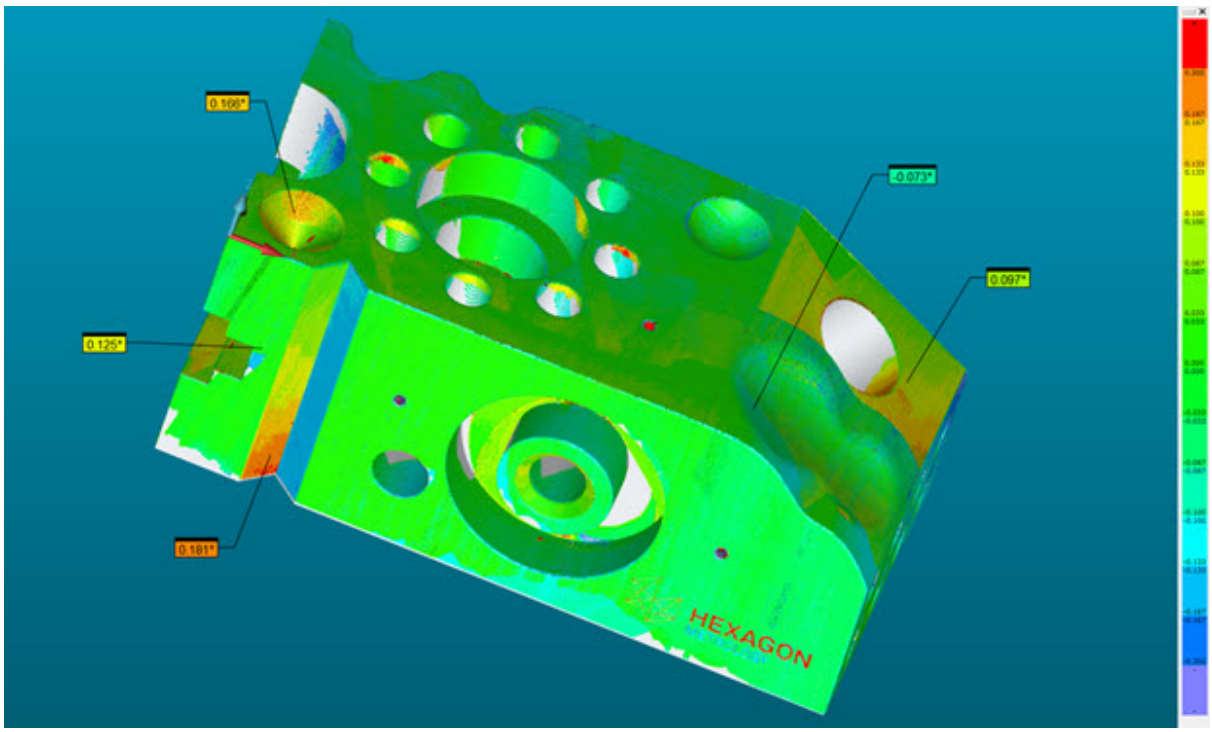
Thickness - This allows you to add a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

The following three annotation point check box options are only available if you DO NOT select the **Dots**, **Needles**, and **Text** check boxes. Also, the CAD model must be visible in order to create annotation points.

Create annotation points check box - For details on this check box, see the "Create annotation points" description in the "Surface Colormap" help topic.

Create MinMax annotations check box - For details on this check box, see the "Create MinMax annotations" description in the "Surface Colormap" help topic.

Show annotation points check box - When you select this check box, the software displays all annotation points.



Example of a point colormap with annotations

Click **Create** to insert a `COP/OPER,POINT COLORMAP` command into the Edit window as shown below.

```
COPPCOLMAP1=COP/OPER,POINT COLORMAP,PLUS
TOLERANCE=0.0394,MINUS TOLERANCE=-0.0394,THICKNESS=0,

SHOW DOTS=YES,DOT SIZE=0.0787,SHOW NEEDLES=YES,NEEDLE
SCALE=10,SHOW LABELS=YES,

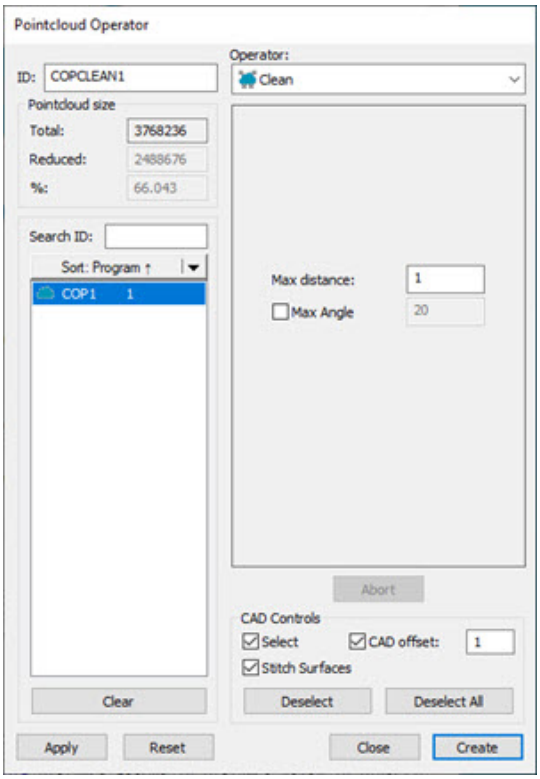
SIZE=50023

REF,COP2,,
```

Colormaps in the Report

For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

CLEAN



Pointcloud Operator dialog box - Clean Operator

The Clean operation uses the distance of the points to the CAD model of the part to eliminate outliers. If the distance of a point is greater than the value of **Max distance**, PC-DMIS considers the point an outlier that does not belong to the part. To use this operation, you must establish at least a rough alignment (see "Creating a Pointcloud/CAD Alignment").

To apply the Clean operation to a Pointcloud, click **Clean Pointcloud** ( ) on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Clean**. This immediately cleans the pointcloud.

Select **Insert | Pointcloud | Operator** to open the **Pointcloud Operator** dialog box. Select **Clean** from the **Operator** list.

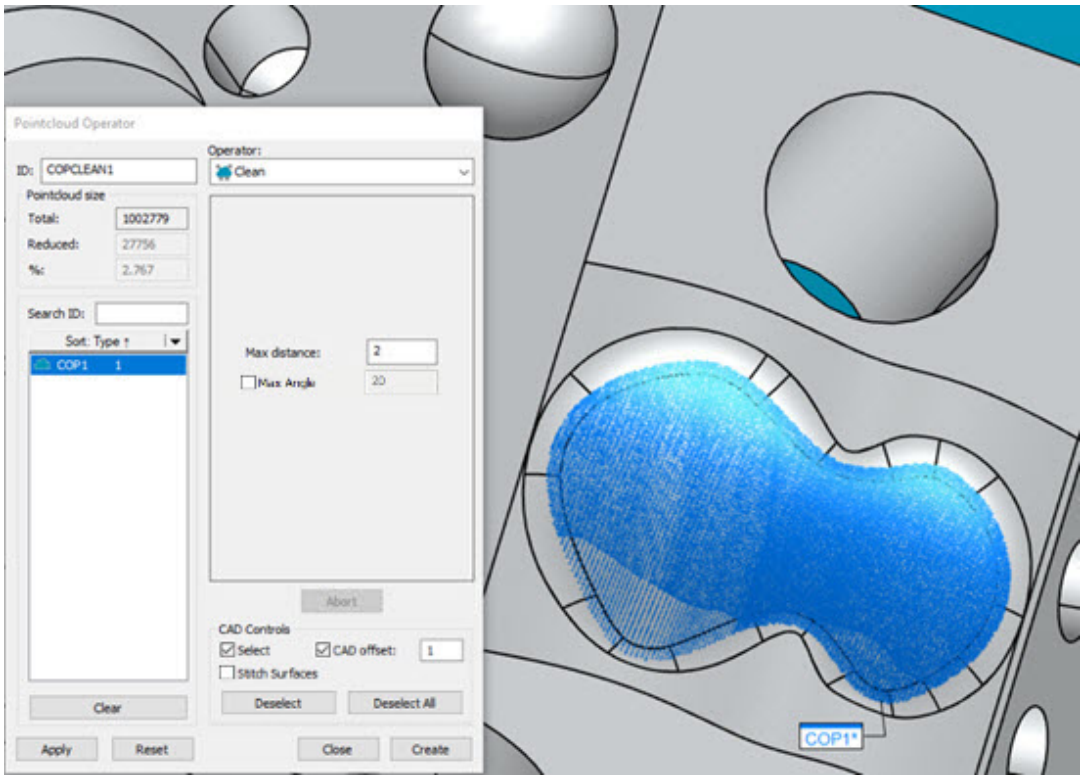
The dialog box for the **Clean** operator contains these options:

Max distance - This is the maximum distance a point can be to the CAD model and not be considered an outlier.

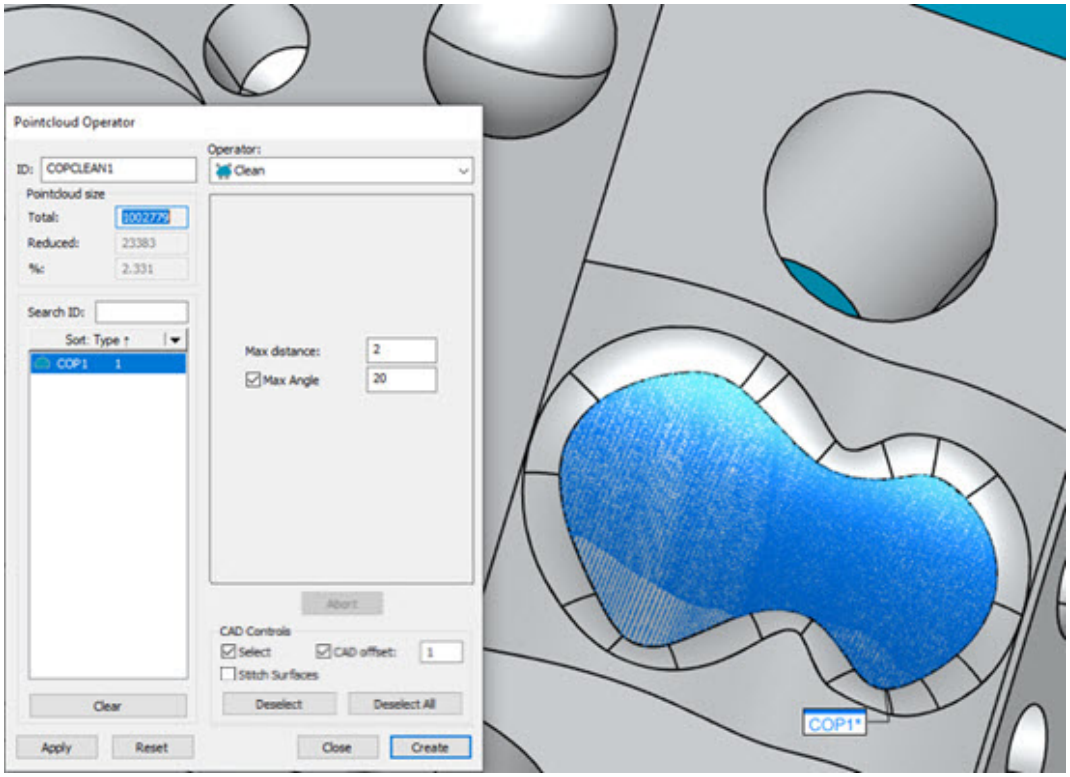
Max Angle check box and value - Select the check box to enable this function. When you enable this function, you can enter an angle value. The software returns the relevant points whose estimated normals are within the specified angle of the CAD normals. This useful tool can result in a "cleaner" Clean when you select data related to CAD faces.

By default, PC-DMIS disables the **Max Angle** setting so it does not impact existing measurement routines.

The following examples show the results of the **Max Angle** check box cleared and then with it marked.



Example 1 - Clean operator with the Max Angle option disabled



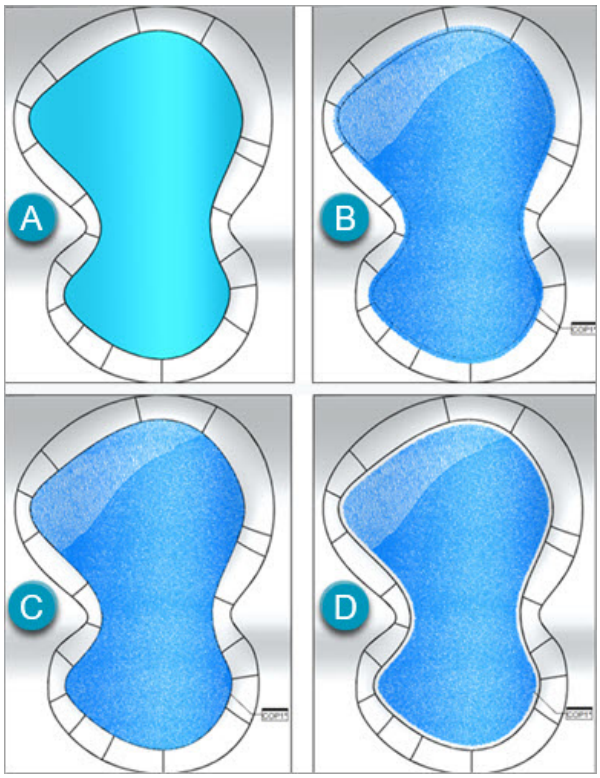
Example 2 - The same Clean operator with the Max Angle option enabled

The Edit window code for the second example would look like this:

```
 COPCLEAN1 =COP/OPER,CLEAN,MAX DISTANCE-2,APPLY MAX
ANGLE=YES,MAX ANGLE=20,SIZE=23383,REF=COP1,,
```

CAD Controls - If you select the **Select** check box, you can click specific CAD surfaces in the Graphic Display window to apply the Clean operation. The software highlights the selected surfaces in red. The operation affects every point in the pointcloud on the selected surfaces. PC-DMIS discards any point located at a distance greater than the specified **Max distance** from all selected surfaces. For example, suppose you select a single surface and enter a value of 10. This means that the software cleans any point in the COP located 10 or more units away from the selected surface. Any point in the COP within 10 units of the selected surface remains.

With the **Select** check box selected, the **CAD Offset** check box becomes available. Select the **CAD Offset** check box to enable the **CAD Offset** input box. Enter a value that PC-DMIS uses to "shrink away" from the CAD edges. This allows you to isolate points relative to specific CAD faces and ignore points along the edge within this fixed offset distance.



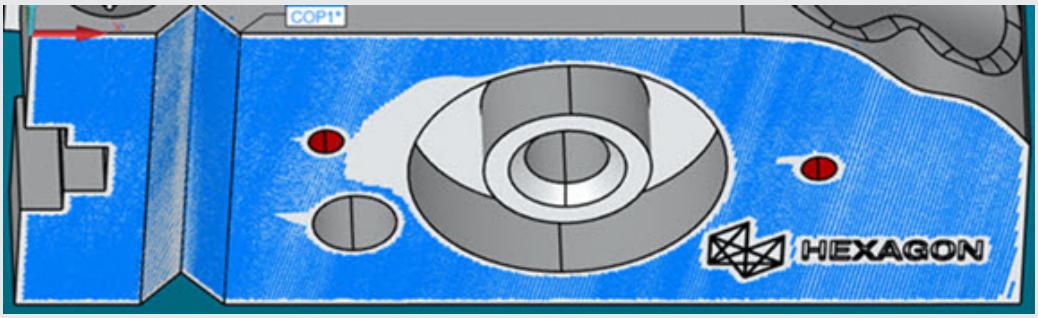
Example using the Clean operator with the Max Distance and CAD Offset options

***A** - CAD Surface selected in the Graphic Display window*
***B** - Clean operation applied with a Max Distance of 1 mm*
***C** - Clean operation applied with a Max Distance of 1 mm and Max Angle of 20 degrees*
***D** - Clean operation applied with a Max Distance of 1 mm, Max Angle of 20 degrees, and CAD Offset of 1 mm*

If the **CAD Offset** value is less than the **Max distance** value, PC-DMIS displays a message that the **CAD Offset** value must be greater than or equal to the **Max distance** value.

When you click **OK**, PC-DMIS resets the **CAD Offset** value to the current **Max distance** value.

You can also select multiple CAD surfaces when using the Clean operation and the **CAD Offset** option. If the CAD surfaces are tangent to each other, the software typically applies the offset to the outside boundaries. However, if the surfaces are not tangent, or if there are discontinuities in the CAD model, the selected surfaces may be offset individually.



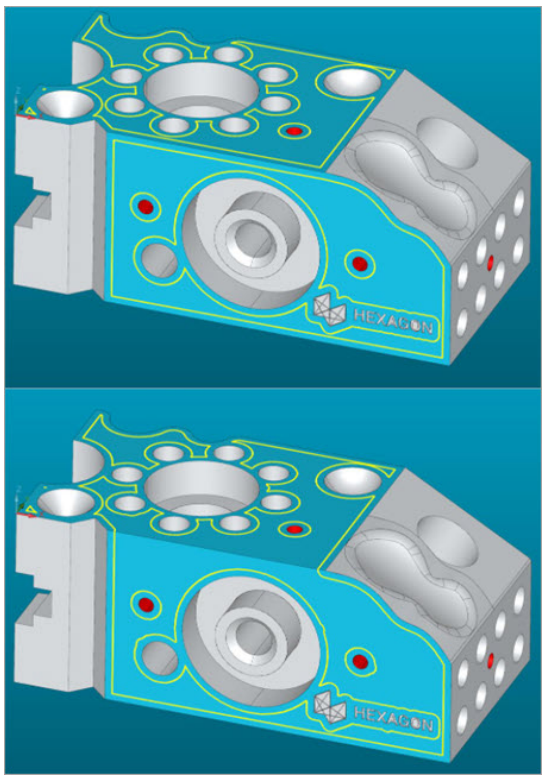
Example of the Clean operator applied to multiple tangent CAD surfaces with a CAD Offset of 1 mm.

With the COPCLEAN function, if you select a large number of CAD faces, mark the **Select** check box, and type a **CAD Offset** value in the **CAD Controls** area of the **Pointcloud Operator** dialog box, PC-DMIS displays a message.

This message indicates that if you click **OK** to perform the CAD Offset operation with the current settings, PC-DMIS will take a long time to process due to the large number of CAD faces you picked.

You can select the **Don't ask me again** check box so that the message doesn't reappear.

With the **Select** and **CAD offset** check boxes selected, PC-DMIS enables the **Stitch Surfaces** check box. If you mark this check box and then select multiple contiguous surfaces on the CAD model, the software stitches these surfaces together and processes them as one surface.

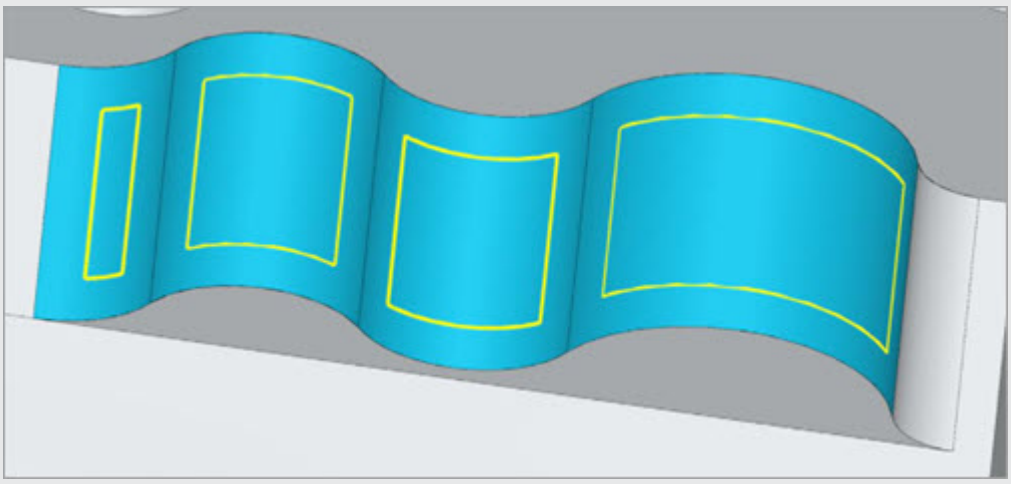


Examples before (top) and after (bottom) applying the Stitching Surfaces option

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.

In this example you can see that without stitching, there are gaps (twice the CAD offset) between each curve:



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.

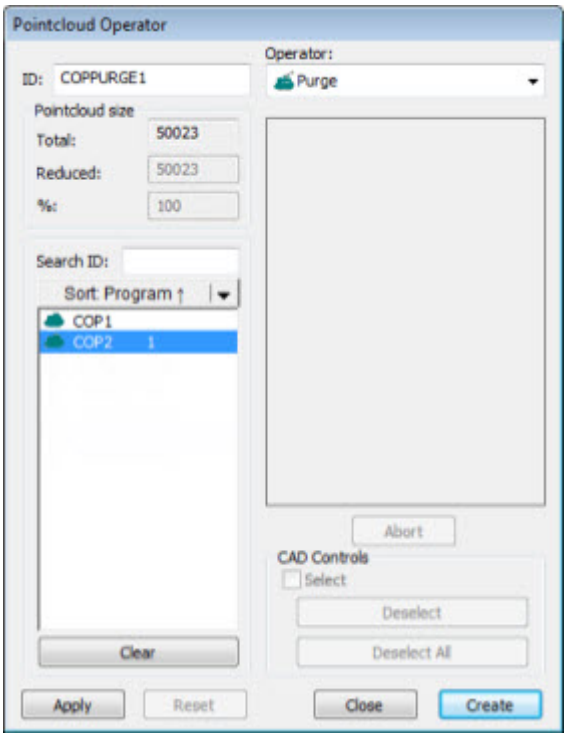
Click **OK** to continue the operation or click **Cancel** and the software will not execute the operation.

When you finish the updates to the dialog box, click **Create** to insert a `COP/OPER,CLEAN` command into the Edit window as shown here:

```
 COPCLEAN1 =COP/OPER,CLEAN,MAX DISTANCE=1,APPLY MAX
ANGLE=NO,STITCH SURFACES=YES,

    SIZE=41709,REF=COP1,,
```

PURGE



Pointcloud Operator dialog box - Purge Operator

From the COP command referenced by this operator, the Purge operation removes all data points that do not belong to this operator. The Purge operation is irreversible and affects all other operator commands that refer to the same COP container, so use the operation with caution.

To apply the Purge operation to a pointcloud, on the **Pointcloud** toolbar, click **Purge Pointcloud** (), or select **Operation | Pointcloud | Purge**.

Clicking **Create** inserts a `COP/OPER,PURGE` command into the Edit window like the following examples:

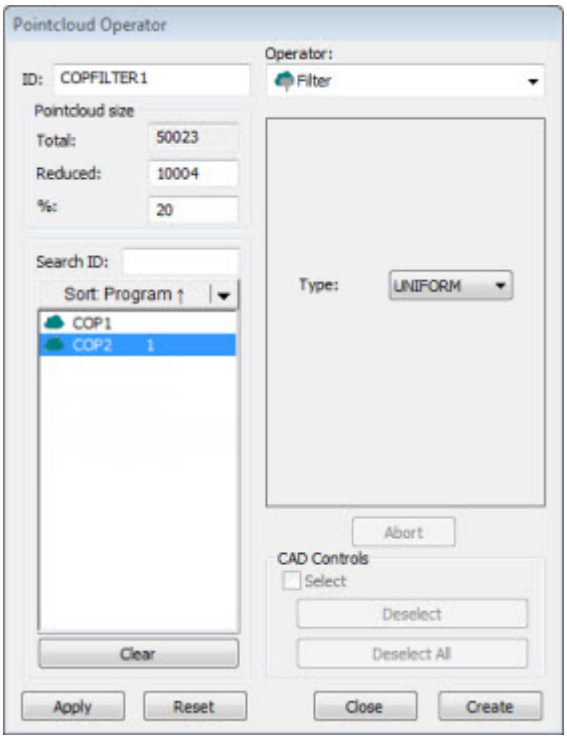
```
COPPURGE1=COP/OPER,PURGE,SIZE=0

    REF,COPSECTION1,,
```



Once you apply this command to a COP, it is permanent and there is no way to restore the COP data. You cannot click **Undo** to restore the COP data.

FILTER



Pointcloud Operator dialog box - Filter Operator

The Filter operation filters data to a smaller subset of points.

To apply the Filter operation to a pointcloud, on the **Pointcloud** toolbar, click **Filter Pointcloud** ( ), or select **Operation | Pointcloud | Filter**.

The Filter operator has these options:

Type - Indicates the type of filter operator to apply. The available options are:

UNIFORM - This option generates a subset of points distributed evenly in the X, Y, and Z directions. It produces the same effect as a regular grid in 2D, but in this case the effect is a 3D grid.

CURVATURE - This option generates a subset of points with the highest estimated curvatures, mainly around edges, vertices, and highly curved areas of the surface.

RANDOM - This option generates a subset of points randomly distributed in the pointcloud.

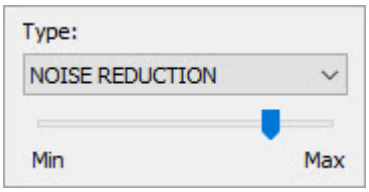
DISTANCE - This option generates a subset of points where points are at least the specified **Distance** value apart from each other.

Distance - When you select **DISTANCE**, the value you enter in this box specifies the distance for the distance filter.

INCIDENCE ANGLE - This option generates a subset of points which excludes (filters out) points that have a normal vector orientation that fall outside the specified angle, relative to the laser sensor orientation. This filter allows you to remove laser points caused by secondary reflections or "noise". You can see the effect of this filter after you click the **Apply** button in the dialog box.

- A valid value is any real number from 10 to 90, inclusive.
- To use this filter, the pointcloud data must have vector information.

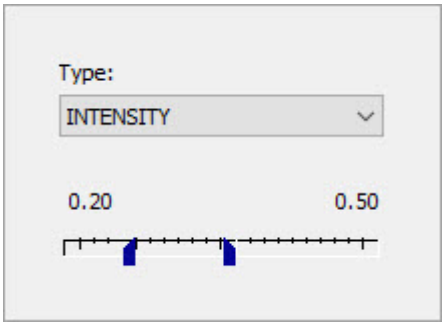
NOISE REDUCTION - The **Noise Reduction** filter works on the global density of the selected pointcloud. PC-DMIS removes the points that are too far away from the global density of the pointcloud.



The slider represents the minimum and maximum values for the **Noise Reduction** filter. The **Min** value represents 0 (zero). Zero means the software applies no noise filter to the data. The **Max** value represents the highest value you can set the noise filter to. The highest value is 99. The default value is 80, and that is where the slider is set to in the above image.

As you move the slider, PC-DMIS updates the display in the Graphic Display window to show the *excluded* points in red and the *included* points in green.

INTENSITY - If the selected pointcloud contains intensity values, you can use the **INTENSITY** filter to define an intensity range in which PC-DMIS keeps only the points that fall within that defined range.



Currently, only the AT600 Laser Tracker, the AS1 Laser Sensor (when running through RDS version 6.6 or newer) and the HPL-10.10 Laser Sensor can provide intensity values.

The sliders represent the minimum and maximum values for the intensity range. With the sliders set in the above image, all points with intensity values from 0.20 to 0.50 would be kept.

As you adjust the sliders, PC-DMIS dynamically updates the Graphic Display window to show the effects of this filter.

To filter COP data:

1. From the **Type** list, select a filter type.
2. From the list of commands, select the Pointcloud command to which you want to apply the filter.
3. In the **Reduced** or **%** boxes, specify the number of points or the percentage of points to keep after applying the filter. This does not apply to the **Distance** filter.
4. Click the **Apply** button.

   PC-DMIS filters the data, and the Graphic Display window shows the result. The size of the filtered data may differ slightly from the value that you specified. It is even more noticeable when you execute the measurement routine, and the software collects the data from the scan commands. It is generally impossible to get the same number of points from a laser sensor if you repeatedly scan the same entity.
5. When the results are acceptable, click the **Create** button. PC-DMIS adds a `COP/OPER,FILTER` command to the measurement routine that contains all the information related to the applied filter.

Suppose your COP1 had 10,000 points initially, and you have this `COP/OPER,FILTER` command in the Edit window:

```
COPFILTER3=COP/OPER,FILTER,UNIFORM,SIZE=3000

    REF,COP1,,
```

The filter replaces the 10,000 points held in COP1 with the filtered 3,000 points. COP1 then holds the filtered 3,000 points for its pointcloud.

- PC-DMIS flags the 7000 points it didn't use, so that you can undo the Filter operation with the Reset operation.
- You can permanently delete the 7000 points that PC-DMIS did not use with the PURGE operation.

For more information, see "RESET" and "PURGE".

Pointcloud EXPORT

You can export pointcloud data in one of two ways, either approach exports the data in a specified format to an external file.

Method 1 -

This method uses a menu item or toolbar button to export the full pointcloud from a COP command right away.

Method 2 -

This method uses an Export operator in a COP operator command to handle the export. This provides you with additional options to export a subset of the data. PC-DMIS exports the data only when it executes the command.



When you export XYZ file types, you can define the separator character to use. For details, see "`ExportXYZSeparator`" in the "PointcloudOperator" section of the PC-DMIS Settings Editor documentation.

Method 1 - Pointcloud Toolbar or Menu Option

This method uses a menu item or toolbar button to export the full pointcloud from a COP command right away.

1. Position your cursor on or immediately below the pointcloud command in the Edit window that you want to export.
2. Then from the **Pointcloud** toolbar, click one of these options to open the **Export Pointcloud Data** dialog box:
   - **Export Pointcloud in XYZ format** (

)
   - **Export Pointcloud in IGS format** (

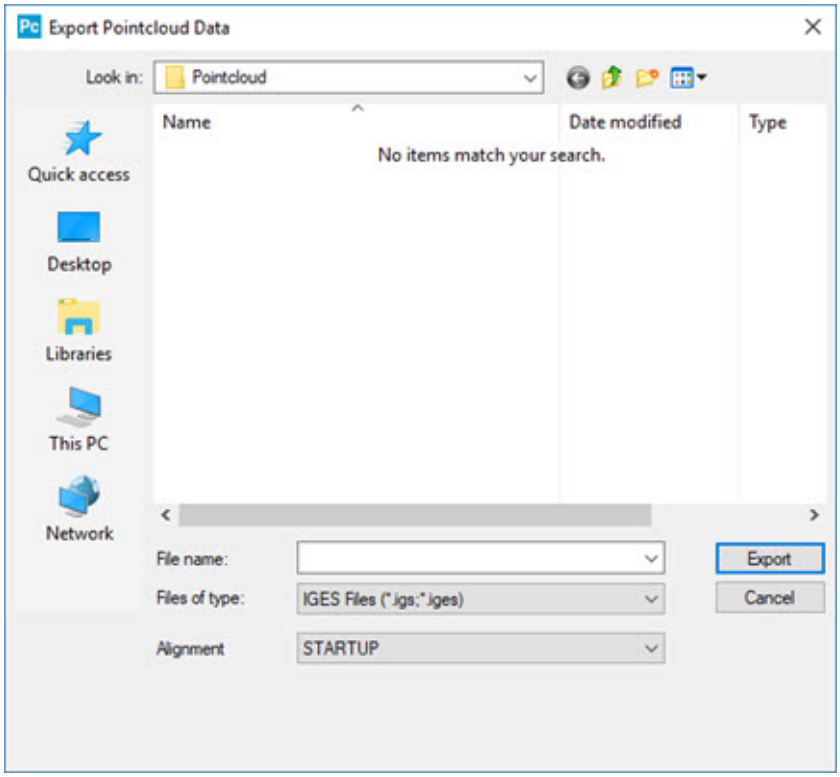
)
   - **Export Pointcloud in PSL format** (

)
3. In the **Export Pointcloud Data** dialog box, navigate your file system and open the folder where you want to export the file.
4. In the **File name** box, define the export file name.
5. From the **Alignment** list, select the type of alignment to use.

6. Click **Export** to generate the export file in the defined folder.

Alternatively, instead of using the **Pointcloud** toolbar, you can select the **File | Export | Pointcloud** menu option and choose **XYZ**, **IGS**, or **PSL**.

Export Pointcloud Data **Dialog Box**



Export Pointcloud Data dialog box

File name - This option indicates the name of the export file.

Files of type - This option shows the data format for the export operation you chose. To use a different file type, close this dialog box and select a different export type from the menu or toolbar.

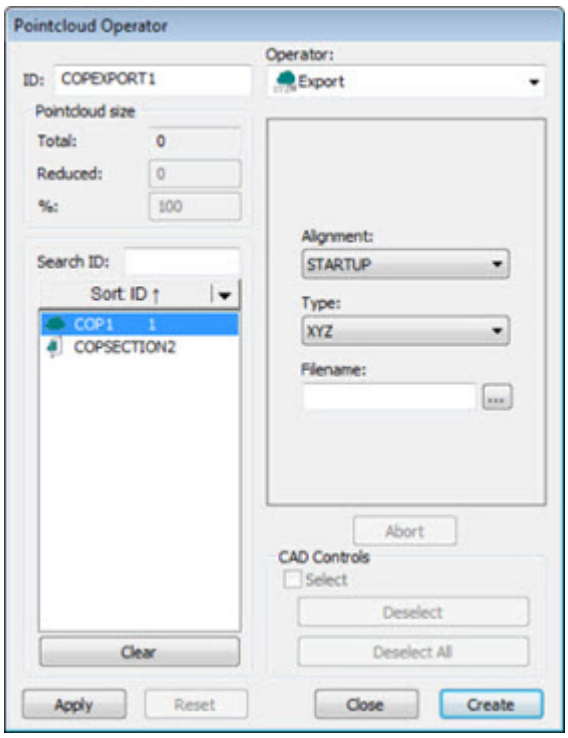
Alignment - This option indicates the type of alignment to include when you export the data.

Export - This button exports the pointcloud data for the COP command when you click it.

Method 2 - Export Operator

This method uses an Export operator in a COP operator command to handle the export. This provides you with additional options to export a subset of the data. PC-DMIS exports the data only when it executes the command.

1. Select the **Insert | Pointcloud | Operator** menu option to insert a `COP/OPER,EXPORT` command into the Edit window. When you select this menu option, the **Pointcloud Operator** dialog box appears.



Pointcloud Operator dialog box - Pointcloud Export Operator

2. From the **Operator** list on the top right, choose **Export**.
3. From the **Type** list, select the data format you want to use for the export file.
4. In the **Filename** box, type the full output path an file name, or use the **...** button to the right to define the path and file name.
5. Click **Create** to insert a `COP/OPER,EXPORT` command that exports the data upon execution similar to this:

```
COPEXPORT1=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/
Dataout.IGS,SIZE=1623201

    REF,COP1,,
```

Pointcloud Operator - Available Options

PC-DMIS offers a number of options for cases where you may not always wish to export the entire COP.

- If you applied a filter to the COP command, then you should reference the `COPFILTER` command for export rather than the original COP command.

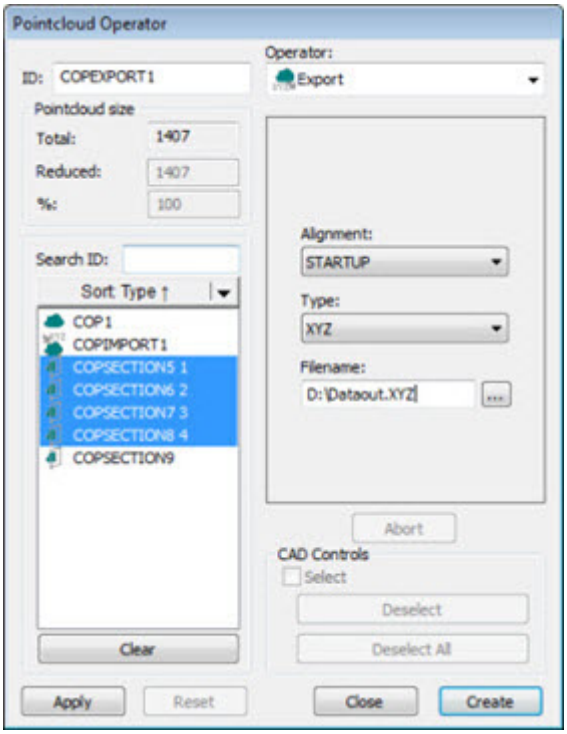
  For example, `REF, COPFILTER1,` rather than `REF, COP1,` . This ensures that the exported file reflects the filter set.

```
    COPEXPORT2=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/
    Dataout.IGS,SIZE=0

        REF,COPFILTER1,,
```

- If your `COP/OPER, EXPORT` points to a cross section using IGES type, PC-DMIS exports the selected cross sections as a B-spline curve using IGES Type 126.
- If your `COP/OPER, EXPORT` points to a colormap using the XYZ type, PC-DMIS exports the nominal XYZ, measured XYZ, and T deviation data for each point.

It is also possible to select more than one command in the list of commands to export them in a single operation:



Pointcloud Operator dialog box with multiple commands selected

In this case, PC-DMIS inserts the command into the Edit window as shown below.

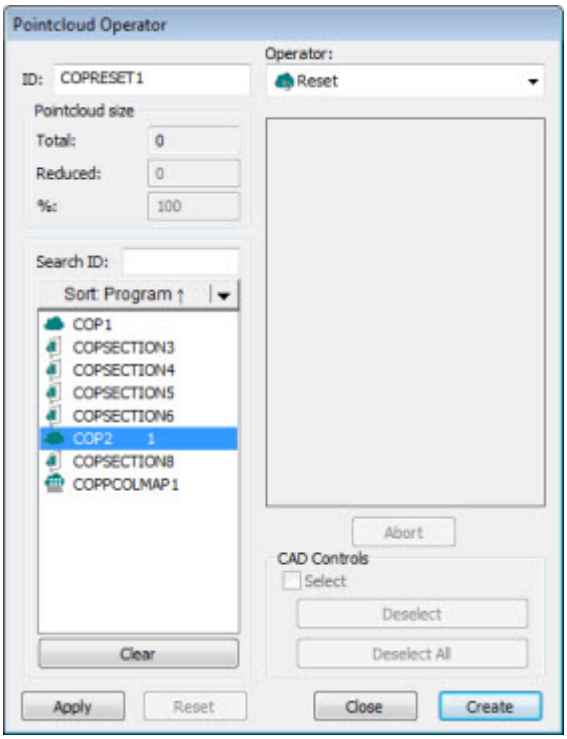
```
 COPEXPORT1=COP/OPER,EXPORT,FORMAT=XYZ,FILENAME=D:/
Dataout.XYZ,SIZE=1246

     REF,COPSECTION1,COPSECTION2,COPSECTION3,COPSECTION4,,
```

If you use multiple commands, they must be of the same type. For example, you cannot select a mix of cross section and filter commands or a mix of surface colormap and point colormap commands.

Also, when you select multiple point colormap commands for export, you cannot select multiple point colormap commands where only some of them have a display method. The selected point colormap commands either must all have no display method or must all have a display method ("Dots", "Needles", or "Text").

RESET



Pointcloud Operator dialog box - Reset Operator

The Reset operation has a behavior similar to Undo. It resets the data referred to in a previous operator command so that the new operator command represents all the data of the referred COP command and not only a subset.

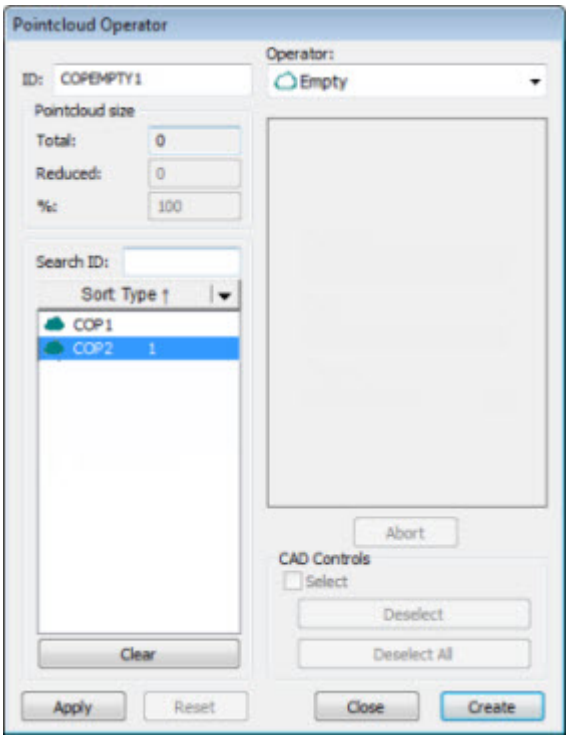
To apply the Reset operation, on the **Pointcloud** toolbar, click **Reset Pointcloud** ( ), or select **Operation | Pointcloud | Reset**.

Click **Create** to insert a `COP/OPER,RESET` command into the Edit window as shown below.

```
COPRESET7 =COP/OPER,RESET,SIZE=0

    REF,COPFILTER 2,,
```

EMPTY



Pointcloud Operator dialog box - Empty Operator

This operation deletes all of the data contained in a selected COP or operator command. When you execute this command, PC-DMIS removes the data of the associated COP.

To apply the EMPTY Pointcloud operation to a pointcloud:

1. If you have more than one pointcloud defined, position your cursor at the location of the pointcloud you want to empty. If you only have one pointcloud defined, place the cursor on or above it.

2. Click **Empty Pointcloud** ( ) on the **Pointcloud** toolbar or select **Operation | Pointcloud | Empty**.

3. Click **Create** to insert a `COP/OPER,EMPTY` command into the Edit window as shown in the code below:

```
COPEMPTY2 =COP/OPER,EMPTY,SIZE=0

    REF,COP2,,
```

> Once you apply this command to a COP, it is permanent and there is no way to restore the COP data. You cannot click **Undo** to restore the COP data.

Pointcloud IMPORT

You can import pointcloud data in one of two ways. Both approaches import data from an external file of a specified format into a COP command.

Method 1 -

> This method uses a menu item or toolbar button to import pointcloud data from an external file into a COP command that you select in the Edit window or into a new COP command if there isn't an existing one. The import occurs as soon as you select the file and click **Import**.

Method 2 -

> This method uses an Import operator in a COP operator command to control how the data is imported. PC-DMIS imports the data into a COP command that you select in the dialog box. PC-DMIS does this only when it executes the command.

Method 1 - Pointcloud Toolbar or Menu Option

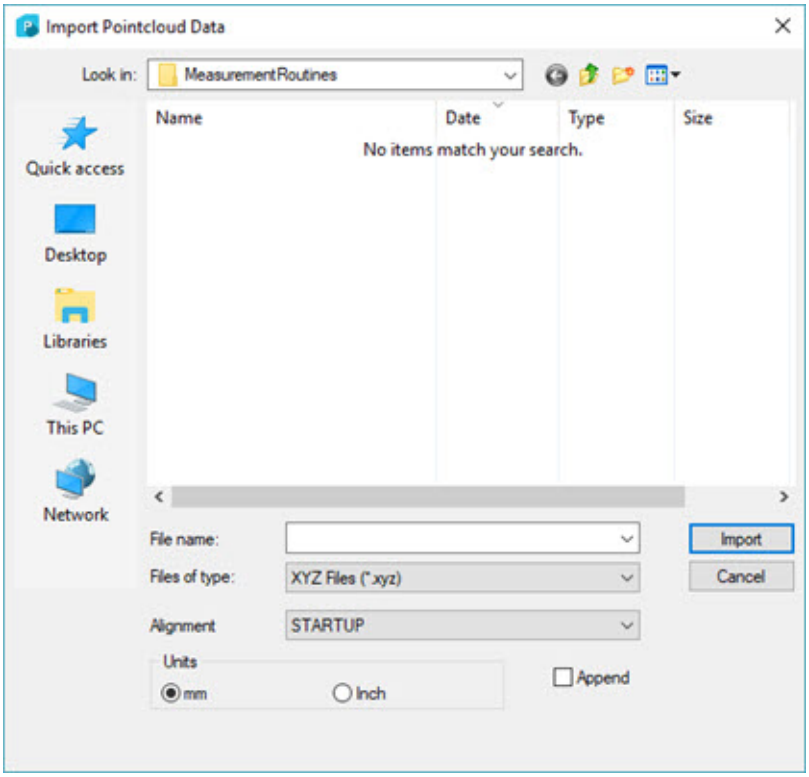
This method uses a menu item or toolbar button to import pointcloud data from an external file into a COP command that you select in the Edit window or into a new COP command if there isn't an existing one. The import occurs as soon as you select the file and click **Import**.

1. From the Edit window, select the COP command where you want to import the data.
2. Then, from the **Pointcloud** toolbar, click one of these options to open the **Import Pointcloud Data** dialog box:
   - **Import Pointcloud in XYZ format** ()
   - **Import Pointcloud in PSL format** ()
   - **Import Pointcloud in STL format** ()
   - **Import Pointcloud in NSD format** ()
3. In the **Import Pointcloud Data** dialog box, navigate to and select the pointcloud data file.
4. If you want to add the imported data to existing COP data, select the **Append** check box.
5. In the **Units** area, select the units of the COP data you are importing.

6. Click **Import** to load the data into the selected COP.

Alternatively, instead of using the **Pointcloud** toolbar, you can select the **File | Import | Pointcloud** menu option and choose **XYZ**, **PSL**, **STL**, or **NSD**.

Import Pointcloud Data **Dialog Box**



Import Pointcloud Data dialog box

Filename - This option indicates the path and file name of the file to import.

Files of type - This option indicates the format of the file you are importing—**XYZ**, **PSL** (Polyworks), **STL**, or **NSD** (3DReshaper). To use a different file type, close this dialog box and select a different export type from the menu or toolbar.

Alignment - Defines the alignment to apply to the imported pointcloud data.

Append - Select this check box to append the imported data to the existing COP. If you clear this option, PC-DMIS replaces the existing COP data.

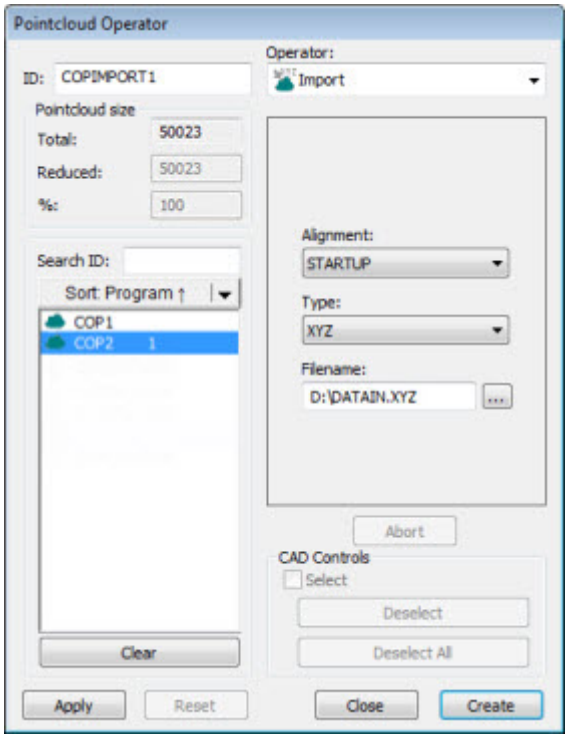
Units - Defines the units used by the import file.

Import - This button imports the pointcloud data into the selected COP when you click it.

Method 2 - Import Operator

This method uses an Import operator in a COP operator command to control how the data is imported. PC-DMIS imports the data into a COP command that you select in the dialog box. PC-DMIS does this only when it executes the command.

1. Select the **Insert | Pointcloud | Operator** menu option to insert a `COP/OPER,IMPORT` command into the Edit window. When you do this, the **Pointcloud Operator** dialog box appears.



Pointcloud Operator dialog box - Pointcloud Import Operator

2. From the **Operator** list, choose **Import**.

2. From the list of COP commands, choose the COP to import into.

3. From the **Type** list, select the file format (**XYZ**, **PSL**, **STL**, or **NSD**).

4. In the **Filename** box, type the full file path and name, or use the **...** button to browse for the file.

5. From the **Alignment** list, select the alignment to apply to the imported pointcloud data.

6. From the **Units** list, select the units used by the import file.

7. Select the **Append** check box if you want to add the imported data to an existing COP.

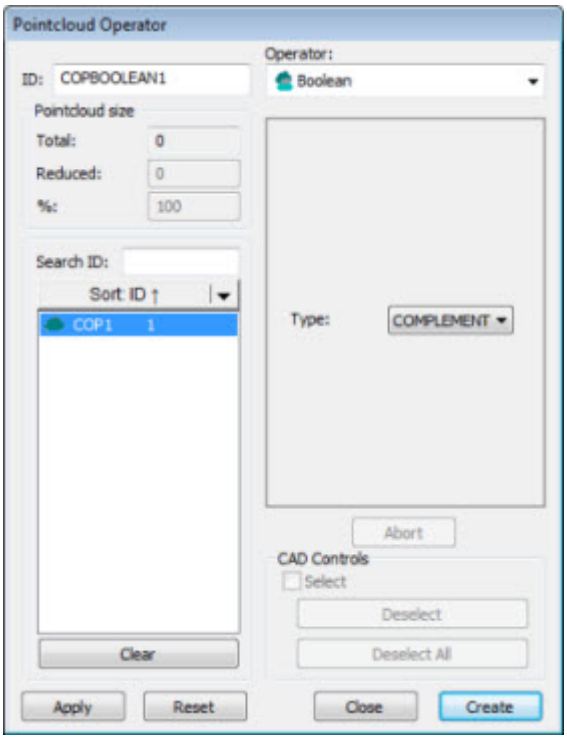
8. Click **Create** to insert a `COP/OPER,IMPORT` command that imports the data upon execution, similar to this:

```
COPIMPORT1=COP/OPER,IMPORT,FORMAT=XYZ,

FILENAME=D:/DATAIN.XYZ,SIZE=0

    REF,COP1,
```

BOOLEAN



Pointcloud Operator dialog box - Boolean Operator

The software applies this operation on one or two operators or COP commands that you select.

The Boolean operator works with data from one COP only, or from commands that reference the same COP. Boolean operations between data in different COPs are not allowed.

For example, suppose you have two pointclouds named `COP1` and `COP2` defined in your measurement routine with different sets of pointcloud data. If you create two COPSELECTION objects from `COP1` labeled `COPSELECTION1` and `COPSELECTION2`, you can use the Boolean operator on `COP1` and one or both of the COPSELECTION objects since they are all created from the same dataset. However, you cannot use the `COP2` object since it is based on a different set of pointcloud data.

PC-DMIS allows you to manually change Boolean command COP references in the Edit window as long as they share the same dataset. If they do not share the same dataset, PC-DMIS changes the reference back to the original reference.

To apply the Boolean operation to a pointcloud, click **Pointcloud Boolean Operation** () on the **Pointcloud** toolbar.

The Boolean operator uses the following option:

Type - This is a list of Boolean type options. Select a type from the list that you want to apply:

COMPLEMENT - This type generates the points that are not visible in a single-selected command.

UNITE - When applied to the two selected commands, this type generates a set of data points that contains all of the points in those commands.

INTERSECT - This type generates the set of data points that have the same locations in two selected commands.

DIFFERENCE - This type removes from the first selected command all of the points that are common with the second selected command.

Click **Create** after you edit the command to insert a `COP/OPER,BOOLEAN` command into the Edit window as you can see here:

```
COPBOOLEAN1=COP/OPER,BOOLEAN,UNITE,SIZE=0
    REF,COPOPER2,COPOPER3,,
```

Gages

PC-DMIS gages are quick-check tools designed to measure lengths along an axis or a direction (Caliper), or a radius on a pointcloud cross section (2D Radius Gage).

PC-DMIS also provides a Temperature Gage and a Thickness Gage. This section describes only the gages that you can use with pointclouds and meshes.

For details on the Temperature Gage, see "Temperature Gage" in the PC-DMIS Core documentation.

For details on the Thickness Gage, see "Thickness Gage" in the PC-DMIS Core documentation.

Caliper Overview



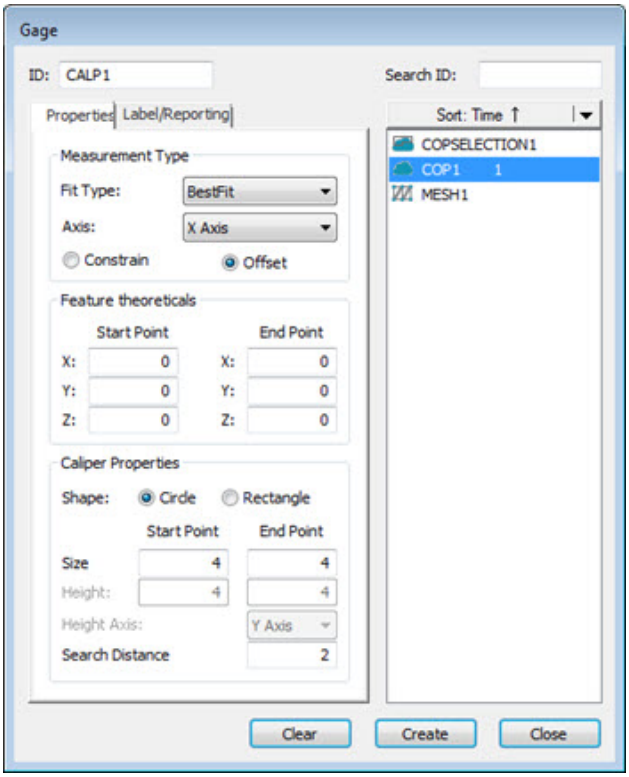
This option is only available if your PC-DMIS license includes the Small COP or Big COP option.

The Caliper is a quick-check tool that works similarly 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.

Select the **Caliper** option from the **Insert | Gage** menu.

You can also access the **Gage** dialog box in these ways:

- Click the **Caliper** button ( ) from the **QuickCloud** toolbar.
- From the **QuickMeasure** toolbar, click the **Gage** drop-down arrow, and then click the **Caliper** button.

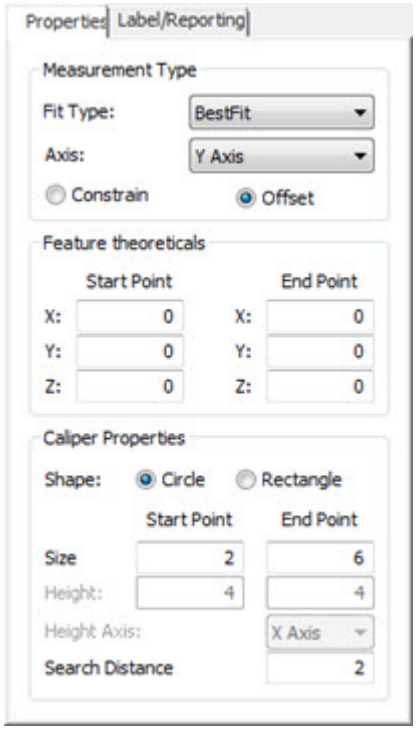


Gage dialog box

A caliper has two tips, which are used to measure the distance between two opposing sides. The caliper's tip has a user-defined size. Click in the Graphic Display window to select the start and end points. Using the data within the tip size, the caliper end points stop at the high points on the selected data (or optionally, on the calculated best fit points). The software performs a search distance along the caliper axis to determine the relevant points.

The **Gage** dialog box has these tabs:

Gage dialog box - Properties tab



Gage dialog box - Properties tab

The **Properties** tab of the **Gage** dialog box has these sections:

Measurement Type

Fit Type: Click the drop-down arrow to display these options:

MaxFit: This setting is the default. Using the tip size and search distance, the caliper end points stop at the high points on the selected surfaces. A search distance along the caliper axis is used to determine the relevant points.

BestFit: A best fit least square fit is applied to all data points that fall within the caliper tip size and search distance. The resulting best fit points are used to determine the caliper length. This alternative method may be used if the scan data contains "noise", but can result in the caliper shown inside the pointcloud or mesh.

Axis: The Caliper can be constructed along the X, Y, or Z axis. Select **Parallel** to construct it Normal to the first surface picked. Select **None** to apply no constraint (3D distance between two points).

Constrain: Select this option to make the two end points exactly opposite one another along the selected axis.

Offset: Select this option to allow the two end points to be offset from each other in position. The measured length remains along the selected axis.

Feature theoreticals

Start Point: This option is the XYZ coordinate location of where the Caliper begins.

End Point: This option is the XYZ coordinate location of where the Caliper stops.

Caliper Properties

Shape: Select the appropriate tip shape, **Circle** (default) or **Rectangle**. If you select **Rectangle**, the **Height** and **Height Axis** options are enabled.



The **Rectangle** option is only enabled when you select the **X Axis**, **Y Axis**, or **Z Axis** option from the **Measurement Type** section. If you select **Parallel** or **None**, the **Rectangle** option is disabled.

Size / Width: The Caliper can have different size start and end tips. Enter the **Size Start Point** and **End Point** values for the circle tip, or the **Width Start Point** and **End Point** values for a rectangle tip. When the distance is then computed, the tip stops at the high point the way a caliper would.

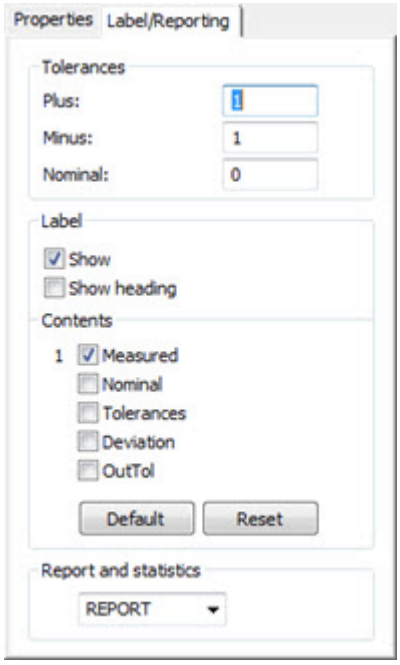
Height: These values define the height of a rectangular tip's **Start Point** and **End Point**. The height size runs along the selected axis. This option is only enabled for rectangular calipers.

Height Axis: Select the option from the list to set the axis used to control the rotation of the rectangle. This option is only enabled for rectangular calipers.

Search Distance: This value defines the length, from nominal, on either side of the picked point. The search distance along with the caliper tip shape creates a cylindrical zone. All data within this zone is evaluated to determine the caliper high point.

For details, see the "Creating a Caliper" topic.

Gage dialog box - Label/Reporting tab



Gage dialog box - Label/Reporting tab

The **Label/Reporting** tab of the **Gage** dialog box has these sections:

Tolerances section

> The default Caliper tolerances are defined by the Dimension Color scale. For details, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

The **Tolerances** section allows you to type plus and minus tolerances for the Caliper length.

To enter the plus, minus, and nominal tolerances:

1. Type the plus tolerance value in the **Plus** box.
2. Type the minus tolerance value in the **Minus** box.

If a CAD model is used, the nominal (theoretical) caliper length is determined from the CAD. If no CAD model is used, the nominal value is updated with the initial measured value. The Nominal value can be edited.

Label section

Show check box: When this check box is selected, the Caliper label and graphic are displayed in the Graphic Display window.

Show heading check box: Toggles the display of the row and column headings in the Caliper label. When this check box is selected, the label's row and column headings are displayed.

Contents area

> The order in which you select the following check boxes defines the order in which they appear in the label. The order number appears to the left of each selected item. When you clear a check box, the software reorders the order numbers of the remaining selected check boxes accordingly.

Measured check box: When this check box is selected, the measured data is displayed in the label.

Nominal check box: When this check box is selected, the nominal data is displayed in the label.

Tolerance check box: When this check box is selected, the tolerance data appears in the label.

Deviation check box: When this check box is selected, the deviation data between the measured and nominal values appears in the label.

OutTol check box: When this check box is selected, the out-of-tolerance data appears in the label.

Default button: Click to set the current selection of check boxes as the default.

Reset button: Click to clear all check boxes in the **Contents** area. The software then resets the section to the auto-setting configuration showing the measured value.

Report and Statistics section

From this section, you can use the options to control the output results:

STATS - 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 output anywhere

When PC-DMIS executes the command, the results are sent to the specified output.

If you choose Stats or Both, a preceding STATS/ON command must exist inside the Edit window to send the results to the stats file.

The items that appear in the text format output are defined by the dimension format command in your measurement routine. For details, see the topic "Dimension Format" in the PC-DMIS Core documentation.

Clear button: Click to reset the **Gage** dialog box to the auto settings configuration.

Create button: Click to create a new Caliper defined with the settings you made in the **Gage** dialog box. The software creates the Caliper.

Close button: Click to close the **Gage** dialog box without creating a Caliper.

Caliper Line Thickness

You can set the thickness of the Caliper line through the **OpenGL** tab of the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see the topic "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.

Creating a Caliper

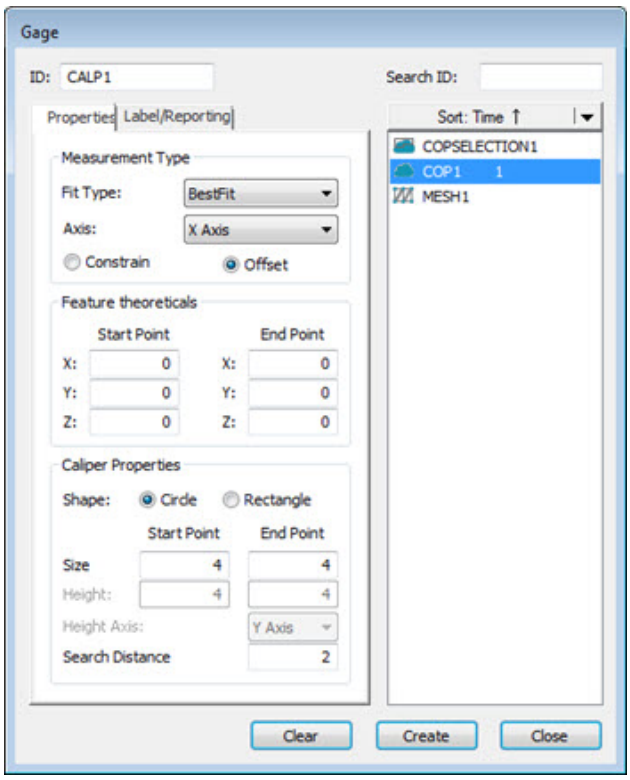
Before you create a Caliper, you must have either a COP, Mesh, or a COPOPER (such as the COPSELECT, COPCLEAN, or COPFILTER) object already defined in your measurement routine. A CAD model is not required.

To create a Caliper feature:

1. Select the **Caliper** option from the **Insert | Gage** menu. The **Gage** dialog box opens.

You can also access the **Gage** dialog box in these ways:

- Click the **Caliper** button () from the **QuickCloud** toolbar.
- From the **QuickMeasure** toolbar, click the **Gage** drop-down arrow, and then click the **Caliper** button.



Gage dialog box

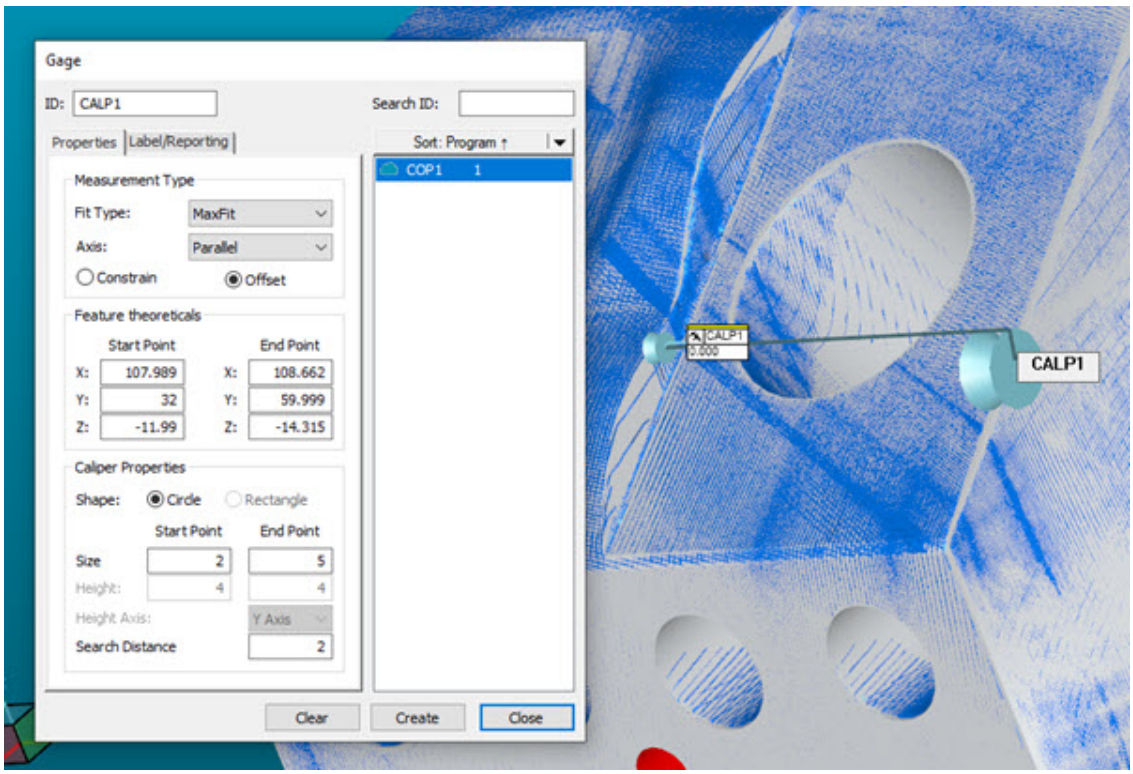
2. Select the COP, COPOPER, or Mesh data object to use.
3. In the **Measurement Type** area, from the **Fit Type** list, select a type.
4. Select an axis in the **Axis** list, and then select the **Constrain** or **Offset** option:

   Select the **Constrain** option to make the two end points exactly opposite one another along the selected axis.

   Select the **Offset** option to allow the two end points to be offset from each other in position. The measured length remains along the selected axis.
5. In the **Caliper Properties** area, select the **Circle** or **Rectangle** shape option.
6. Edit the current value, or select the appropriate values for the following options:

   Circle-shaped caliper tip options

   - **Size:** The default value is 4 mm for the **Start Point** and **End Point** options. You can set the start and end points of the caliper to different sizes depending on the CAD surfaces.



Example of a caliper created with different start and end point sizes

> For non-planar surfaces, you should set the size to a larger value, such as 8-10 mm, to capture the high point. For planar surfaces, you can set it to a smaller value, such as 2 mm.

- **Search Distance:** The default value is 2 mm. This value defines the length, from nominal, on either side of the picked point. The search distance along with the caliper tip shape creates a cylindrical zone. All data within this zone is evaluated to determine the caliper high point.

Rectangle shaped caliper tip options

- **Width:** The default value is 4 mm for the **Start Point** and **End Point** options. The value sets the width of the caliper tip's start and end points.
- **Height:** The default value is 4 mm for both the **Start Point** and **End Point**. The value sets the height of the caliper tip's start and end points.

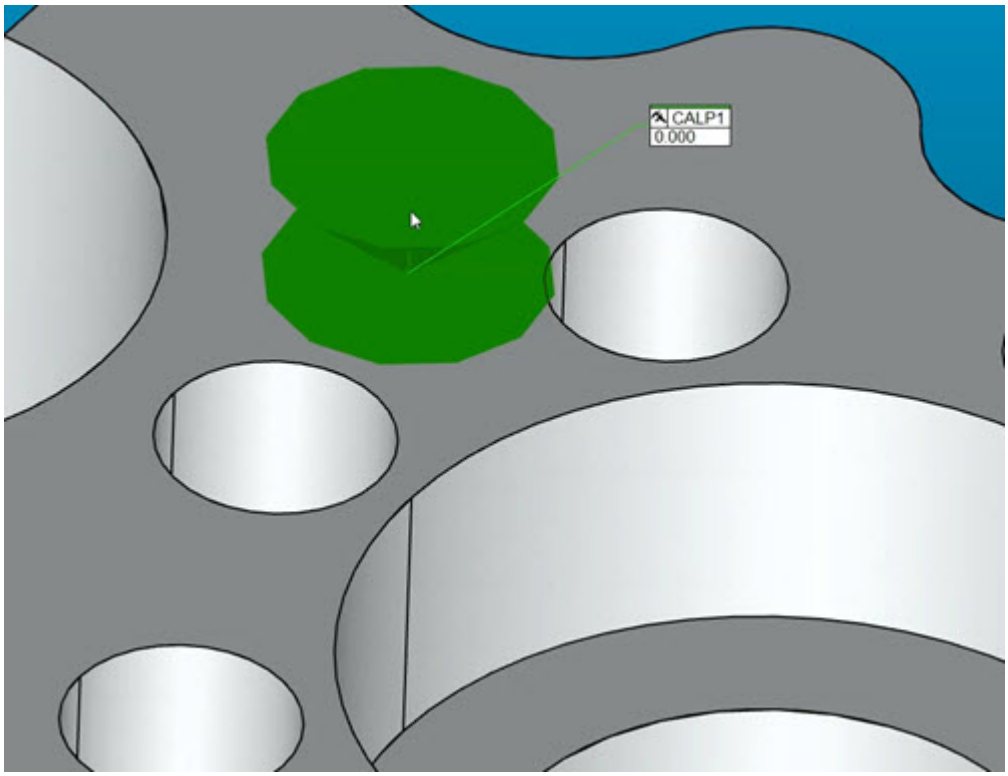
> For non-planar surfaces, you should set the width and height to a larger value, such as 8-10 mm, to capture the high point. For planar surfaces, you can set the width and height to a smaller value, such as 2 mm.

- **Height Axis**: The default value depends on the **Axis** option you select in the **Measurement Type** area. From the list, select the option to define the axis that controls the rotation of the rectangle.

- **Search Distance:** See the description in the **Circle-shaped caliper tip options** section.

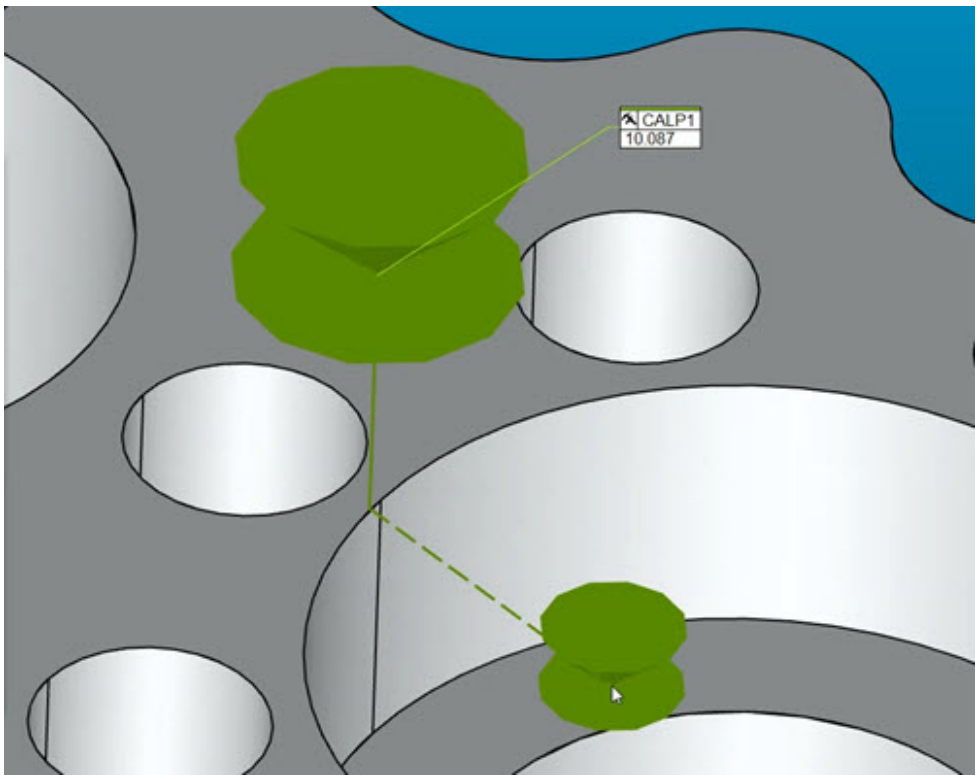
> Changes to any of the **Gage** dialog box properties become the default values the next time the dialog box opens.

7. From the Graphic Display window, click to define the start point. To remove the first selected point, press the Delete key.



8. Move your cursor to the second location, and then click to define the end point. As you move the cursor, the length value updates in the Graphic Display window. If the selected object (COP or Mesh) contains data, the length that is shown is the measured value. If the selected object is empty, and a CAD model is used, the length value that is shown is the nominal value.

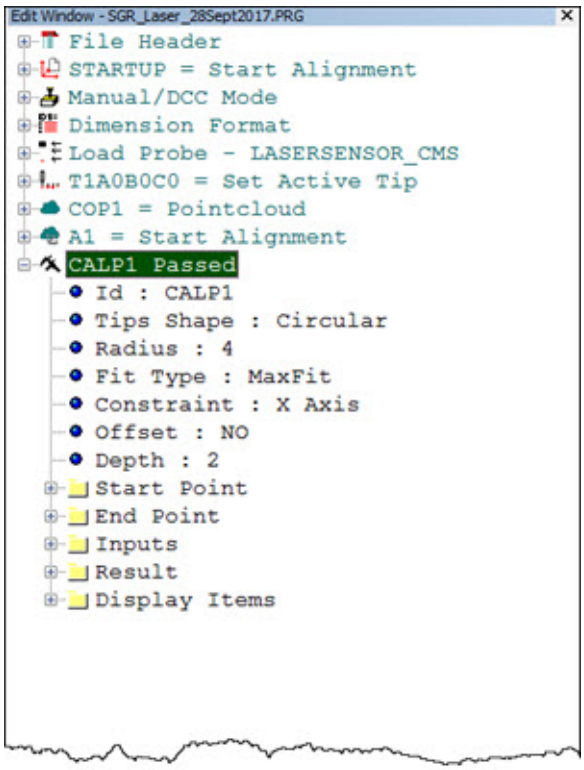
> You can also enter the XYZ values for each in the **Start Point** and **End Point** XYZ boxes.



Caliper Line Thickness

You can set the thickness of the Caliper line through the **OpenGL** tab of the **CAD and Graphic Setup** dialog box (**Edit | Graphic Display Window | OpenGL**). For details, see the topic "Changing OpenGL Options" in the "Setting Your Preferences" chapter of the PC-DMIS Core documentation.

9. Click **Create** to define the caliper and add it to the commands in the Edit window.



Caliper Start Point, Mid Point, and End Point

The software extracts the nominal and measured start and end points of the Caliper gage when:

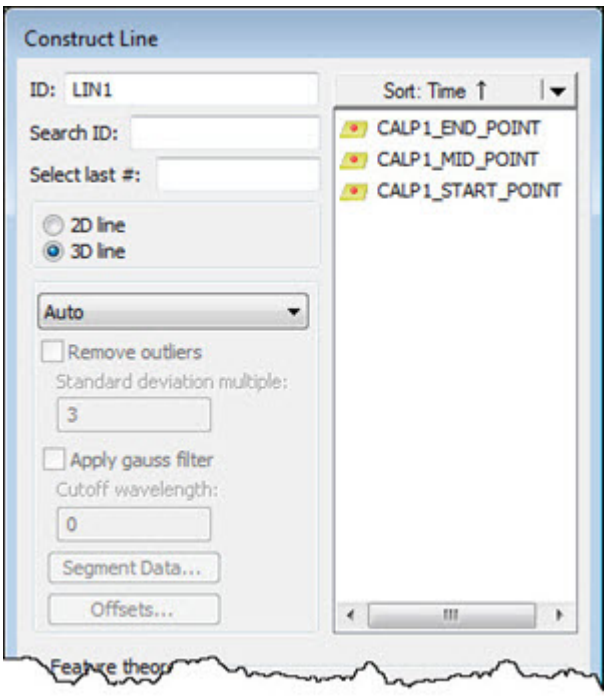
- You create the Caliper
- You execute the Caliper in the measurement routine

The software uses the start and end points to calculate the mid point. PC-DMIS then projects the mid point to the selected axis.

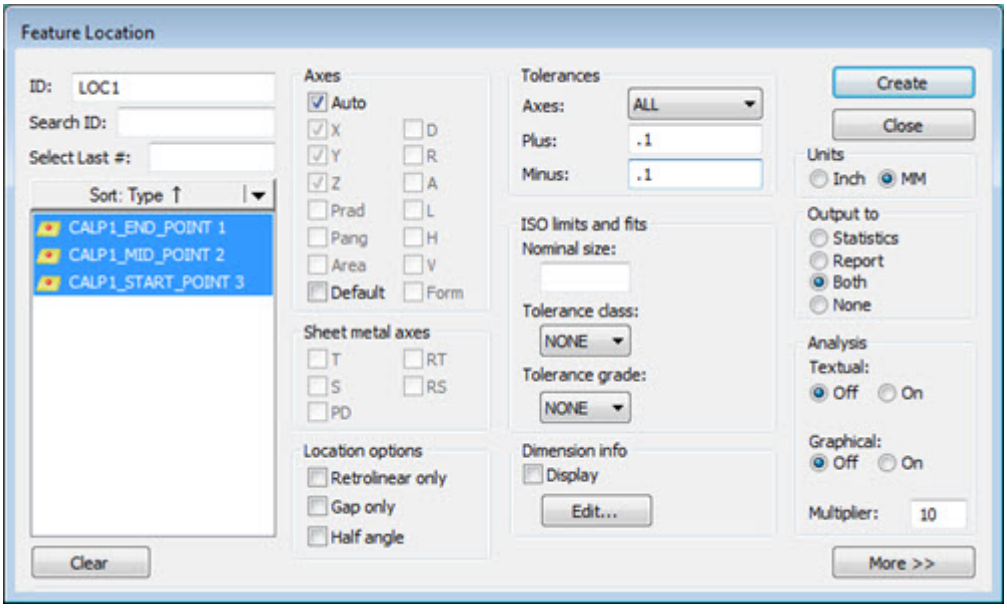
These points are not individual features in the Edit window. They are internal components of the Caliper gage.

The start point, mid point, and end point automatically appear as constructed offset points in the **Dimension**, **Construction**, and **Alignment** dialog boxes. You can dimension the points and use them in a best fit alignment, such as when you align a cast part that has excess material.

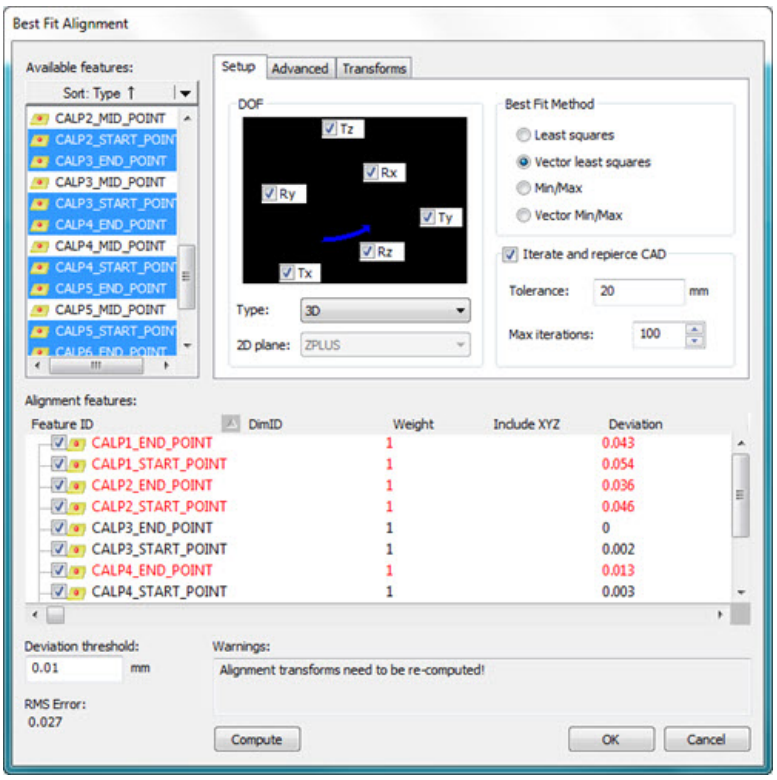
The following examples show various uses of the Caliper start, mid, and end points when you create features and alignments:



Example of start, mid, and end point options when creating a constructed feature

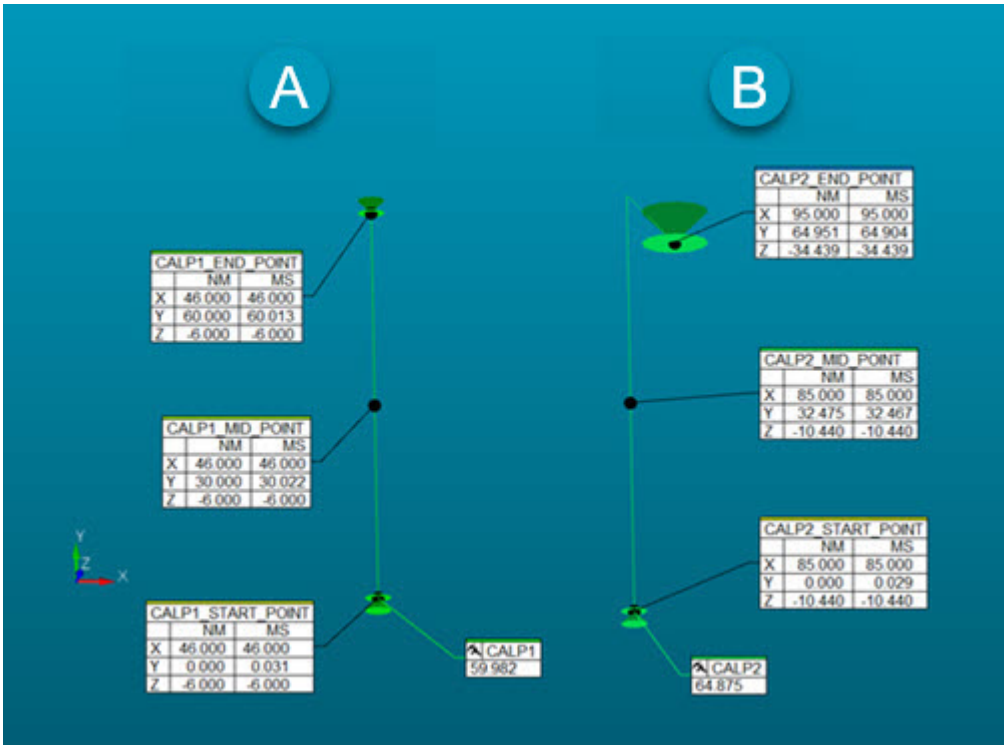


Example of start, mid, and end point options when creating a feature location dimension



Example of start, mid, and end point options when creating an alignment

This example shows the use of the Constrain and Offset methods when you define a Caliper feature:



Examples of the Caliper points using the Constrained (left) and Offset methods (right)
(A) - Caliper1 end points constrained to Y axis
(B) - Caliper2 end points offset to Y axis

2D Radius Gage Overview

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.

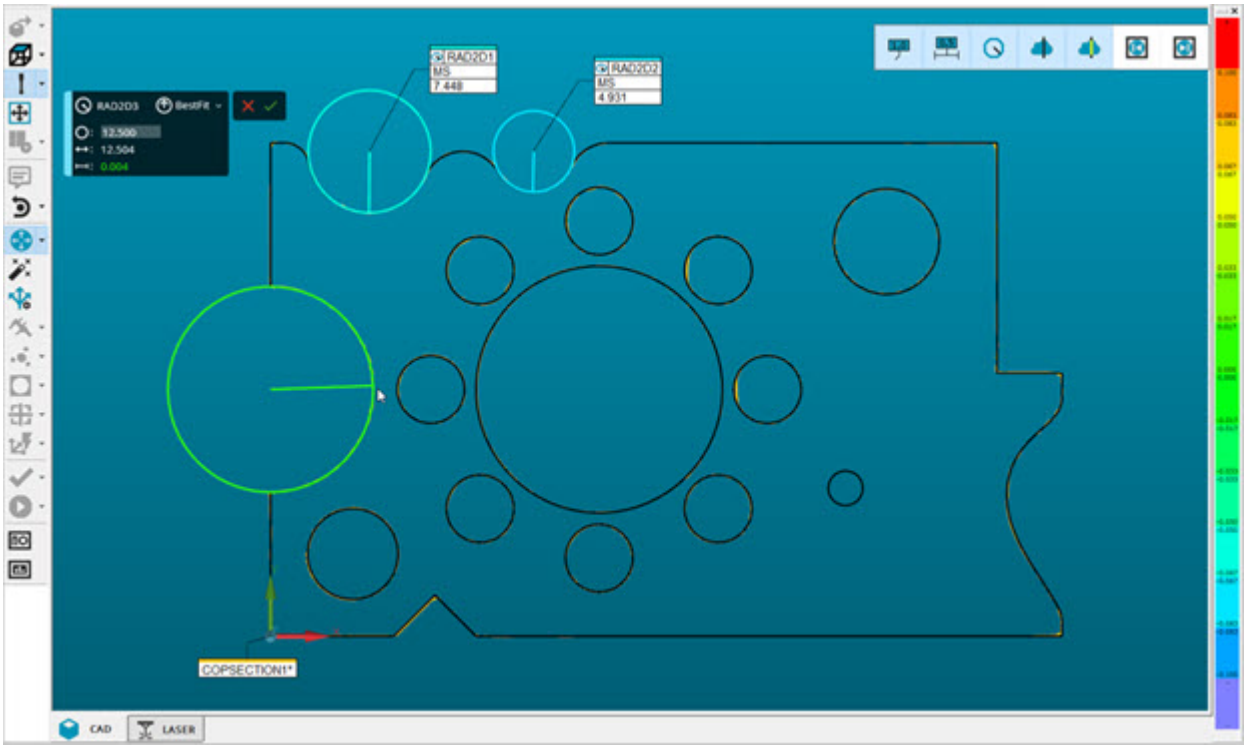
You can create a 2D Radius Gage graphically on a cross section in the 2D Slide Show view.

To create a 2D Radius Gage graphically, do the following:

1. After you create the cross sections, from the **Mesh**, **Pointcloud**, or **QuickCloud** toolbar (**View | Toolbars**), click the **Cross Section Slide Show** button () to display the cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.
2. Hold down the Shift key, and move the mouse cursor onto the radius to view the nominal, measured, and deviation values in the display widget.
3. Left-click to select the radius. You can create or cancel the radius gage from the widget dialog box.

The software uses a Least Square Best Fit algorithm to calculate the 2D Radius by default. The active tolerances are set on the Dimension Colors Bar. The radius gage graphic uses the color from the Dimension Colors Bar which corresponds to its deviation. For details on how to edit the dimension color scale, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

You can change the tolerances for the gage from the Edit window, or press the F9 key to view the **2D Radius Gage** dialog box.



Examples of the 2D Radius Gage

By default, PC-DMIS automatically includes the 2D Radius Gage in the report.

	MM	RAD2D2 - COPSECTION1				
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
R	7.503	0.100	0.100	7.457	-0.046	0.000

Example of a 2D Radius Gage report

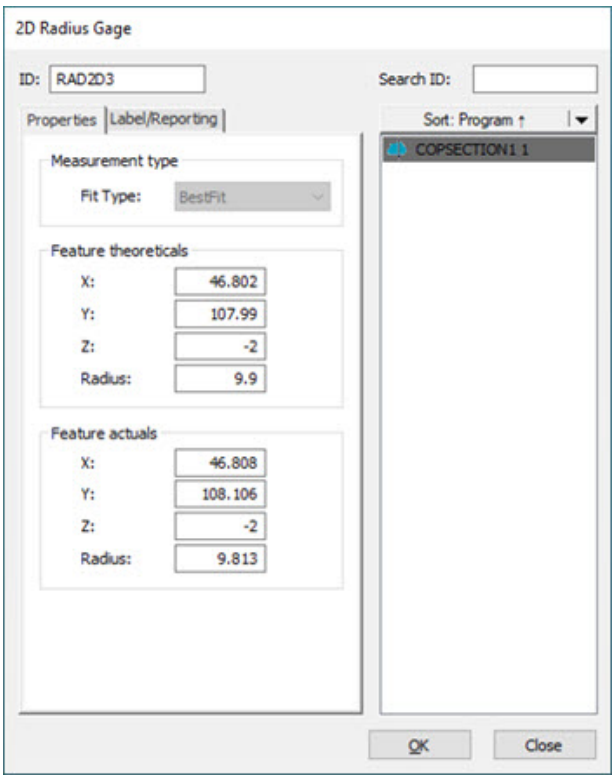
You can turn off showing the 2D Radius Gage in the report from the **Label/Reporting** tab of the **2D Radius Gage** dialog box. For details, see "2D Radius Gage Dialog Box".

Once you create a 2D Radius Gage, you can use it in Location and Distance Dimensions, and Constructions. For Location Dimension, Form is not supported.

2D Radius Gage Dialog Box

The **2D Radius Gage** dialog box has these tabs:

Properties tab



2D Radius Gage dialog box - Properties tab

The 2D Radius Gage is automatically linked to the cross section on which it was created. Since you created the 2D Radius Gage on the cross section, you cannot change the associated cross section.

The **Properties** tab of the **2D Radius Gage** dialog box has these areas:

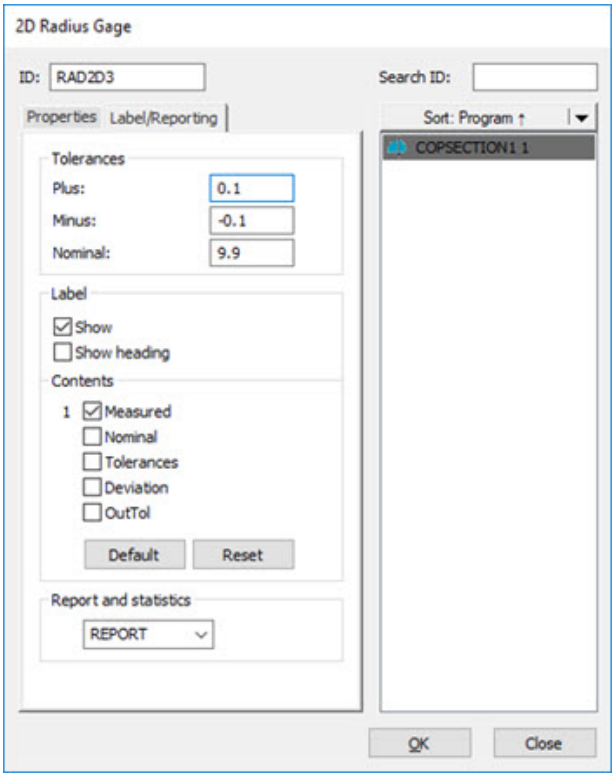
Measurement type

- **Fit Type** - Click the drop-down arrow to display these options:
- **Best Fit** - The software applies a Best Fit Least Square fit to all data points that fall within the radius search zone.

Feature theoreticals - The software displays the XYZ centerpoint location and size of the nominal radius. You can edit the nominal values.

Feature actuals - The software displays the XYZ centerpoint and size of the measured radius. You cannot edit the actuals.

Label/Reporting tab



2D Radius Gage dialog box - Label/Reporting tab

The **Label/Reporting** tab of the **2D Radius Gage** dialog box has these areas:

Tolerances

The **Dimension Color Scale** defines the default 2D Radius Gage tolerances. For details, see "Editing Dimension Colors" in the PC-DMIS Core documentation.

The **Tolerances** section allows you to type the plus and minus tolerances for the radius.

To enter the plus, minus, and nominal tolerances, do the following:

1. Type the plus tolerance value in the **Plus** box.
2. Type the minus tolerance value in the **Minus** box.

If you are using a CAD model, the cross section nominal (black) polyline defines the nominal (theoretical) radius. If you are not using a CAD model, the software updates the nominal value with the initial measured value. You can edit the nominal value.

Label

Show check box - When you select this check box, the software displays the **2D Radius Gage** label and graphic in the Graphic Display window.

Show heading check box - This check box toggles the display of the row and column headings in the **2D Radius Gage** label. When you select this check box, the software displays the label's row and column headings.

Contents

The order in which you select the following check boxes defines the order in which they appear in the label. The order number appears to the left of each selected item. When you clear a check box, the software reorders the order numbers of the remaining selected check boxes accordingly.

Measured check box - When you select this check box, the software displays the measured data in the label.

Nominal check box - When you select this check box, the software displays the nominal data in the label.

Tolerances check box - When you select this check box, the software displays the tolerance data in the label.

Deviation check box - When you select this check box, the software displays the deviation data between the measured and nominal values in the label.

OutTol check box - When you select this check box, the software displays the out-of-tolerance data in the label.

Default button - Click to set the current selection of check boxes as the default.

Reset button - Click to clear all check box selections in the **Contents** area. The software then resets the section to the auto-setting configuration showing the measured value.

Report and statistics

From this section, you can use the options to control the output results:

STATS - When you select this option, the software sends the output to statistical files.

REPORT - When you select this option, the software sends the output to the inspection report.

BOTH - When you select this option, the software sends the output to the inspection report and the statistical files.

NONE - When you select this option, the software doesn't send output anywhere.

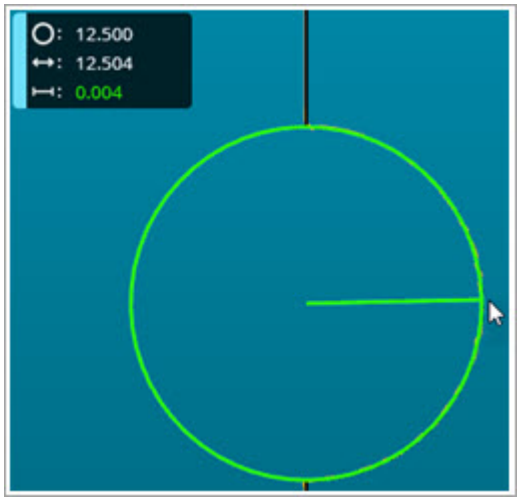
When PC-DMIS executes the command, the software sends the results to the specified output.

If you choose **STATS** or **BOTH**, a preceding STATS/ON command must exist inside the Edit window in order to send the results to the statistical file.

Creating a 2D Radius Gage

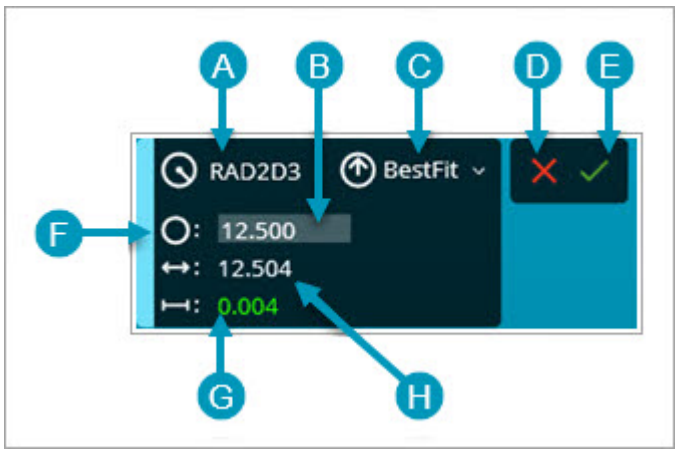
To create a 2D Radius Gage with a Cross Section:

1. Create the cross section.
   - For details on creating a pointcloud cross section, see "CROSS SECTION".
   - For details on creating a mesh cross section, see "Mesh CROSS SECTION Operator".
2. Select the **Cross Section Slide Show** button ( ) from the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**) to view the cross section in a 2D view.
3. Hold the Shift key down and point to the desired radius. A display widget appears. The display widget shows the nominal, measured, and deviation values for the radius.



2D Radius Gage display widget showing the nominal, measured and deviation values for the radius

4. Click to select the radius. A widget dialog box appears.



***A** - 2D Radius Gage ID*
***B** - Radius nominal value*
***C** - Algorithm used to calculate the radius*
***D** - Cancel button*
***E** - Create button*
***F** - Use the handle to move the widget dialog box*
***G** - Radius deviation value*
***H** - Radius measured value*

2D Radius Gage widget dialog box

From the widget dialog box, you can do the following:

- Change the 2D Radius Gage ID (**A**) and the nominal value (**B**).
- From the list (**C**), select the algorithm the software uses to calculate the radius.
- Click the **Create** button (**E**) to create the radius gage or the **Cancel** button (**D**) to close the widget dialog box without creating the radius gage.
- Reposition the widget. To do this, point to the handle on the left side of the widget (**F**). Click and drag the widget in the Graphic Display window to reposition it.

5. When you create the 2D Radius Gage, PC-DMIS creates its associated command in the Edit window. You can create additional radius gages as needed.

Once you create a 2D Radius Gage, you can use it in Location and Distance Dimensions, and Constructions. For Location Dimension, Form is not supported.

To change the radius settings:

- Edit them directly in the Edit window.
- Click the radius gage command in the Edit window, and then press F9 to open the **2D Radius Gage** dialog box to make your changes.

How the 2D Radius Gage is Calculated

- When the cross section has both nominal (black polyline) and measured (yellow polyline) data:

  Calculating the Nominal 2D Radius

  Starting from the initial picked measured point, the nominal radius is found on the nearest black polyline. The software calculates the nominal (theoretical) radius for a least square best fit circle, using all the nominal points that are within 0.005 mm standard deviation.

  Calculating the measured 2D Radius

  The software calculates a least square best fit circle, using the actual points on the yellow polyline that are associated with the nominal points.

- When the cross section has only nominal data (black polyline):

  Starting from the initial picked nominal point, the software finds the radius on the nearest black polyline. The software calculates the nominal (theoretical) radius for a least square best fit circle, using all the nominal points that are within 0.005 mm standard deviation.

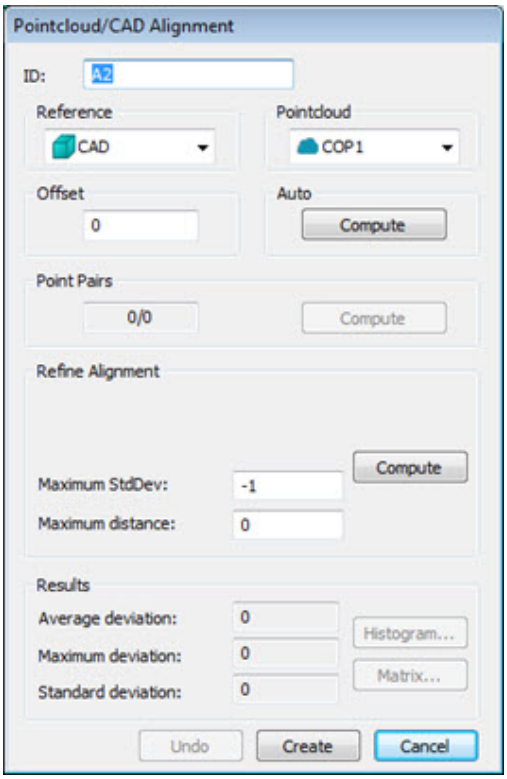
- When the cross section has only measured data (yellow polyline):

  Starting from the initial picked measured point, the software calculates the radius for a least square best fit circle. The software uses all the measured points within 0.050 mm standard deviation and a search distance of 0.25 mm to find any additional segments that belong to the radius.

Pointcloud Alignments

In order to use the data you've collected in your pointclouds properly, you need to create an alignment between the pointclouds and the CAD data of your part model or between pointclouds. This is done using the **Pointcloud/CAD Alignment** dialog box.

Pointcloud/CAD Alignment Dialog Box Description



Default view of Pointcloud/CAD Alignment dialog box

The **Pointcloud/CAD Alignment** dialog box contains these options:

ID - This box displays the identification label for the alignment.

Reference - Use this list to select the point of reference for your alignment, usually either from the CAD itself or a defined COP.

Pointcloud - This list lets you choose the pointcloud to use in the alignment.

Offset - This box defines an offset value for a surface CAD model and is typically used with sheet metal parts. Applying an offset value essentially gives the surface CAD model a thickness. This allows you to align the pointcloud data to a different face that isn't represented in the surface CAD model. For example, if you have a surface CAD model for the top of a part, but you want to align to a corresponding bottom surface, you could apply an offset value of the part's thickness to align the scanned data to the bottom side.

- To apply a thickness in the same direction as the surface normal vector, use a positive value.
- To apply a thickness opposite the surface normal, use a negative value.

This option is available for pointcloud to CAD alignments. For details on pointcloud to CAD alignments, see "Creating a Pointcloud to CAD Alignment".

Auto - This area lets you automatically align the CAD with the pointcloud with the **Compute** button. This option is available for Pointcloud to CAD alignments.

Point Pairs - This area lets you create a rough alignment based on selected points from the CAD that correspond to selected points from the pointcloud. Once you have the needed pairs selected, you can use the **Compute** button to perform the rough alignment.

Refine Alignment - This area allows for a more refined alignment. Only the **Maximum Distance** option is available for Pointcloud to Pointcloud alignments.

Depending on the alignment being made, the **Refine Alignment** area of the dialog box may consist of the following items:

Compute - This button begins the refined alignment process. A progress bar on the status bar shows the progress as the process moves through the alignment iterations.

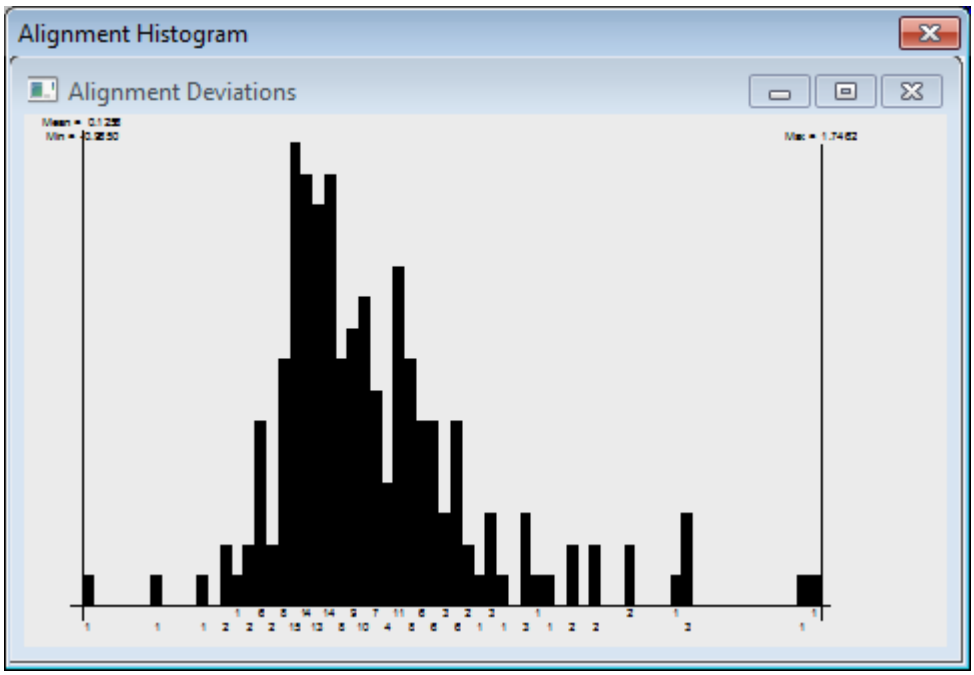
Maximum StdDev - This box defines the maximum standard deviation PC-DMIS uses during the execution of an auto alignment. If PC-DMIS exceeds the entered value during the command execution, the software prompts you to pick point pairs on the CAD/Pointcloud. A value of -1 disables the Maximum StdDev functionality.

Maximum Distance - This box defines the maximum distance PC-DMIS looks from the CAD for valid COP points. If you do not enter a value, PC-DMIS uses the default value of 0 (zero) and the maximum distance becomes half the distance of the CAD bounding box.

Results - This area contains the following items:

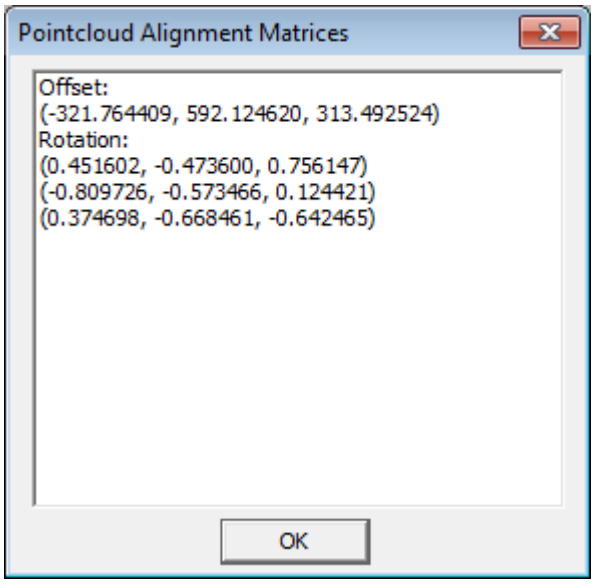
Information boxes showing the **Average Deviation**, **Maximum Deviation**, and **Standard Deviations** of the pointcloud in relation to the CAD model.

Histogram - This button takes a random sample of points from the pointcloud, projects them onto the CAD, and then shows the deviations for that sample in the **Pointcloud Alignment Histogram** dialog box.



Sample pointcloud Alignment Histogram dialog box

Matrix - This button displays the **Pointcloud Alignment Matrices** dialog box for the pointcloud alignment. It shows the numerical values of the alignment, the offset and the rotation matrix.



Pointcloud Alignment Matrices dialog box for the alignment

Creating a Pointcloud to CAD Alignment

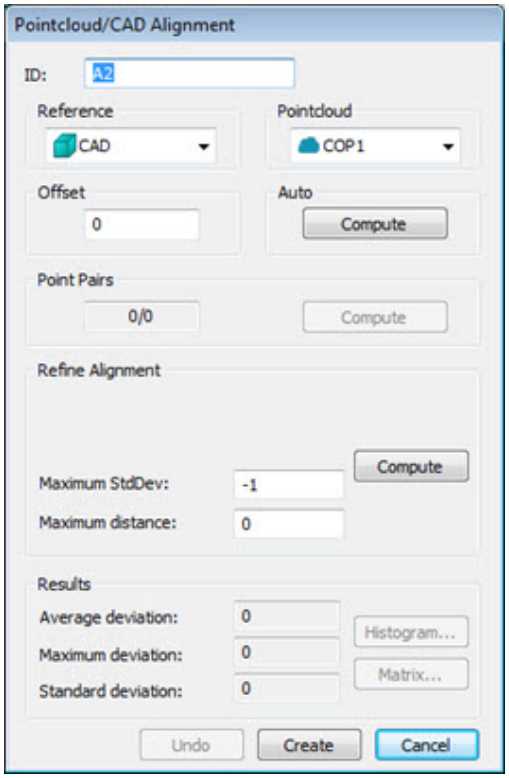
To create a Pointcloud to CAD alignment, do the following:

1. Ensure that you have an imported CAD model in the Graphic Display window and a `COP` command in the measurement routine. PC-DMIS requires these elements to align pointclouds to the CAD.

2. Select the **Insert | Pointcloud | Alignment** menu option.

If you are in Command Mode, you can type the `COPCADBF` command in the Edit window between the `ALIGNMENT/START` and `ALIGNMENT/END` commands to open this dialog box.

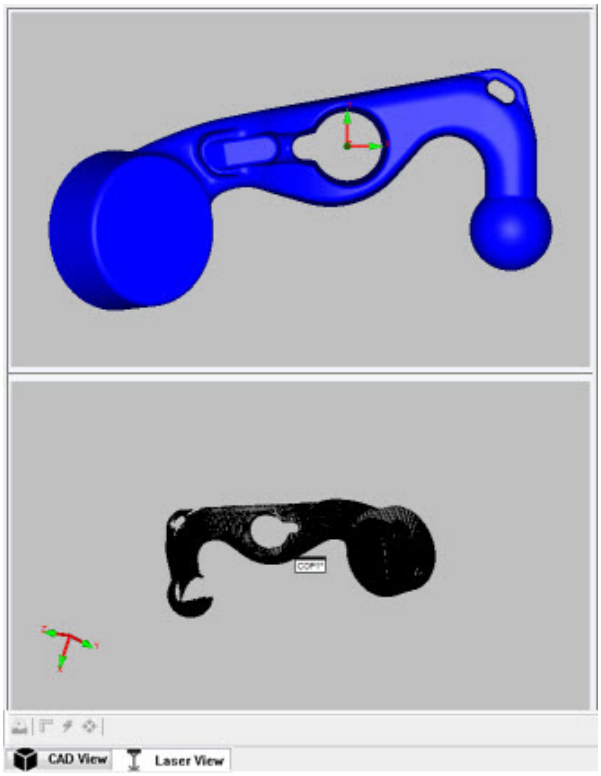
The **Pointcloud/CAD Alignment** dialog box opens:



Pointcloud/CAD Alignment dialog box

For a complete description of the **Alignment** dialog box, see the "Pointcloud/CAD Alignment Dialog Box Description" topic in the PC-DMIS Laser documentation.

3. A temporary split-screen view of the CAD model and pointclouds appears in the Graphic Display window. You can use this split-screen view to visually see the alignment taking place. From the **Reference** drop-down list, select your point of reference; usually, either the CAD model itself or a defined COP is available.



Split-screen view showing the CAD model on the top view and the pointcloud on the bottom view

4. If you have more than one pointcloud in your measurement routine, from the **Pointcloud** list, choose the pointclouds.

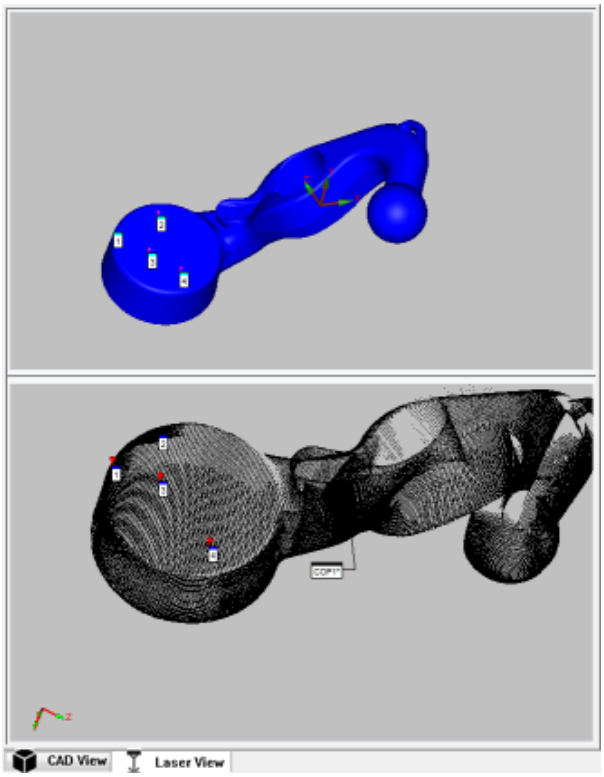
5. Perform the alignment:

    a. In the **Auto** section, click the **Compute** button. You should use this only when you have a full scan of the external faces of the part. This automatically performs an alignment of the Pointcloud to the CAD and also a refinement on the alignment as it's generating.

    b. First, use the **Pointcloud/CAD Pairs** area to perform a rough alignment that brings the pointcloud close enough to the CAD (if not already close) to be able to refine the alignment further if needed. You should use this type of alignment if the pointcloud is not complete or it contains scanned data that belongs to a fixture, the table, and so forth.

        i. Click a desired number of points on the pointcloud.

        ii. Click corresponding locations on the CAD model.

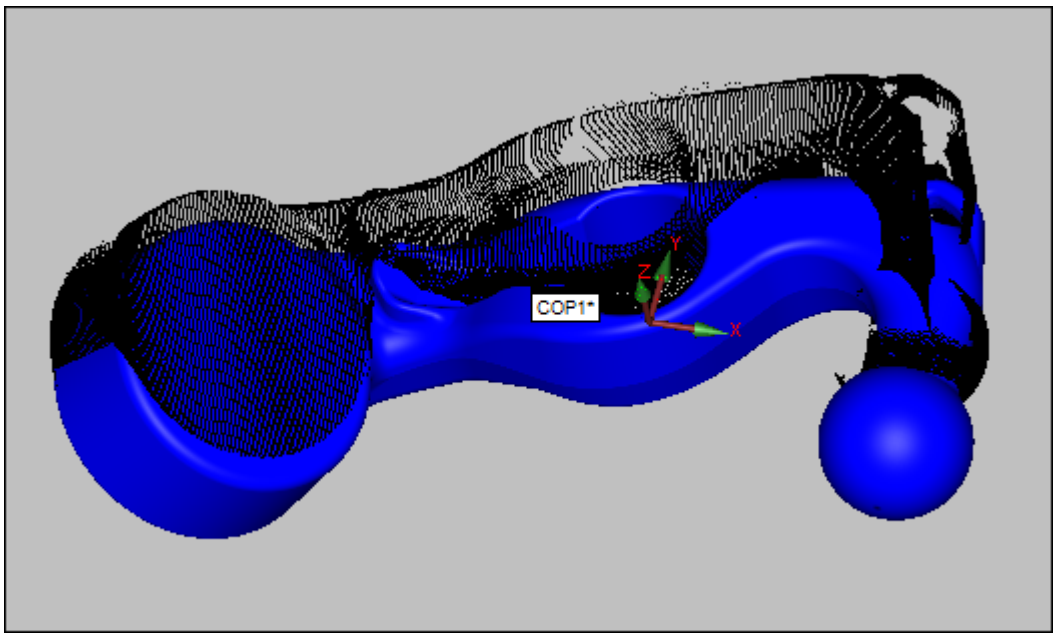


Split-screen view showing selected CAD points (top) and corresponding pointcloud points (bottom)

iii. The more points you take around the different areas of the model and pointcloud, the better the rough alignment.

iv. Click **Compute** to create the rough alignment.

c. Next, use the **Refine Alignment** area whenever you want to refine your alignment. This brings the pointcloud closer to your CAD model. To obtain a good, refined alignment, the pointcloud points should be close enough to the CAD points through an initial rough alignment.



A sample rough pointcloud to CAD alignment that needs a refinement

i. In the **Maximum StdDev** box, define the maximum standard deviation for the auto alignment execution between the points in the pointcloud and the CAD model. When the auto alignment command is executed, if the standard deviation of the Pointcloud/CAD deviations is greater than the maximum value defined, you can select point pairs to get a better alignment. The default value of -1 is equivalent to an infinite allowed standard deviation.

ii. Define the maximum distance of the points from the CAD for use in the best fit routines. The default value is 0. In this case, PC-DMIS uses an internal max distance based on the size of the pointcloud.

iii. Click **Compute** to refine the alignment.

6. If a portion of the pointcloud doesn't align nicely with the CAD, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters; or, you can try a different alignment.

7. Use the **Results** area to determine how well the pointcloud aligned with the CAD. Make any changes to the **Offset** or **Refine Alignment** values to improve the alignment if necessary. If any changes are made, be sure to click the **Compute** button to regenerate the alignment with the new values.

8. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the `COPCADBF` command in the Edit window. For information on the Edit window command, see the "COPCADBF Command Mode Text" topic.



If needed, you can adjust the `CadGridSizeForPointcloudCadAutoAlignment` entry to define the distance between the grid of points used to align the pointcloud to the CAD model.

COPCADBF Command Mode Text

The COPCADBF command allows you to perform a best fit alignment of the pointcloud data with the CAD data.

Below is an example code snippet for a COPCADBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
   COPCADBF/REFINE=n1,n2,n3,n4,n5,SHOWALLPARAMS=TOG1,
   ROUGH ALIGNPAIR/
      THEO/<x,y,z>,<i,j,k>,
      MEAS/<x1,y1,z1>
   REF,TOG2,,
ALIGNMENT/END
```

n1 represents the total number of sample points to use in the refinement.

n2 represents the maximum number of iterations.

n3 represents the offset value for applying a thickness.

n4 represents the maximum standard deviation value.

n5 represents the maximum distance value.

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/
   THEO/x,y,z,i,j,k,
   MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined and selected using the Graphic Display window. The values next to `THEO/` represent the point on the CAD. The values next to `MEAS/` represent the corresponding point on the COP. These pairs are used to determine a rough transformation between the CAD and the COP which allows the COP to come close enough to the CAD to allow further refinements of the alignment.

TOG2 lets you choose the pointcloud to use for the alignment.

Creating a Pointcloud/Pointcloud Alignment

The Pointcloud to Pointcloud alignment functionality allows you to best fit align two pointclouds which have been collected in two different reference frames that have some overlap. A typical example is two scans in two pointcloud commands, representing areas of a part that cannot be scanned in the same part orientation.

The alignment is done in two steps:

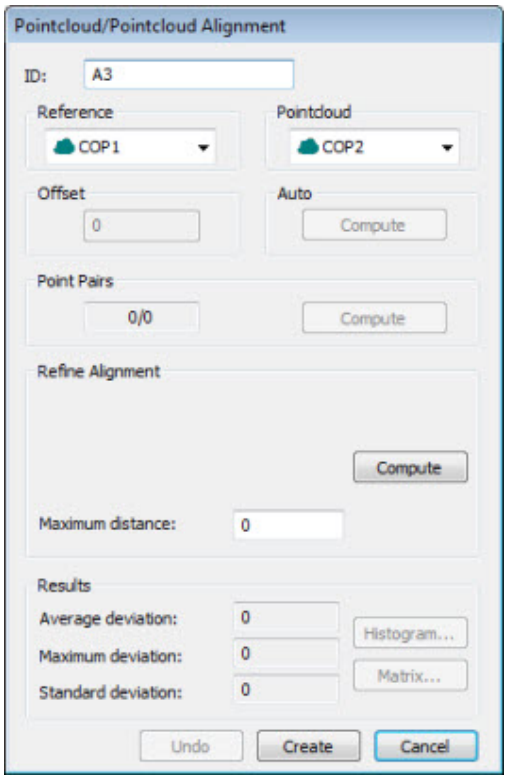
- A rough alignment, where you select pairs of points in the overlapping area of the two pointclouds.
- A refined bestfit, which tries to bring the second pointcloud as close as possible to the reference pointcloud.

To create a Pointcloud to Pointcloud alignment, do the following:

1. Ensure that you have two or more COP commands in the measurement routine that you can use for the alignment. PC-DMIS requires these elements so it can align the pointclouds.
2. Select the **Insert | Pointcloud | Alignment** menu option.

   If you are in Command Mode, you can type the `COPCOPBF` command in the Edit window between the `ALIGNMENT/START` and `ALIGNMENT/END` commands to open this dialog box.

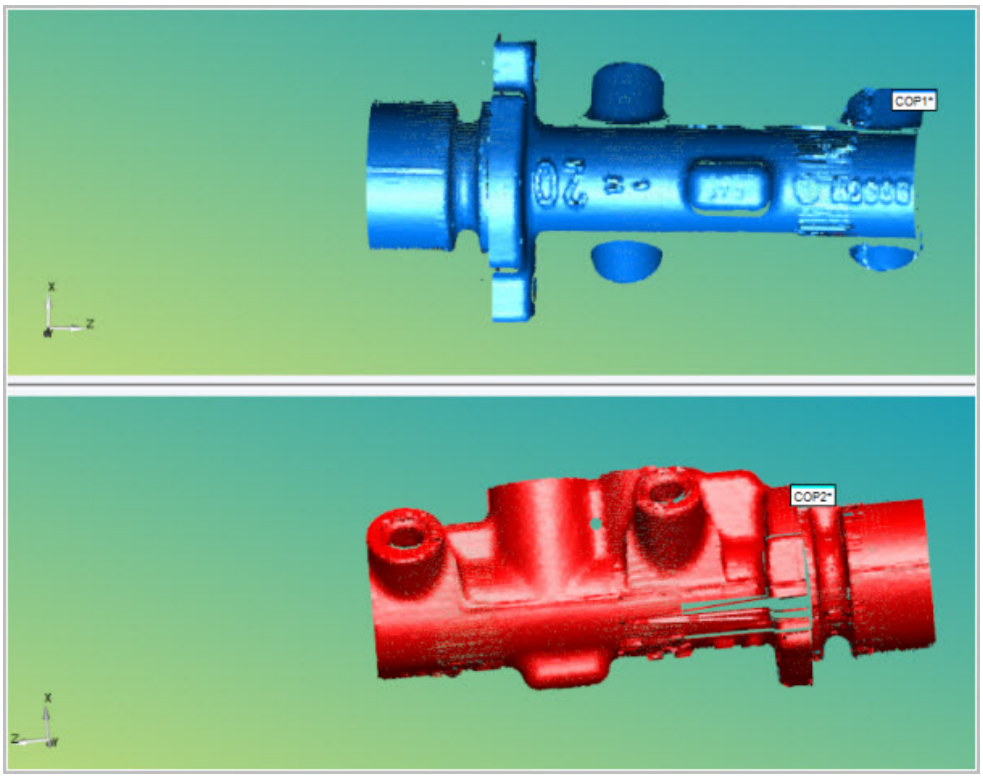
   The **Pointcloud/Pointcloud Alignment** dialog box opens:



Pointcloud/Pointcloud Alignment dialog box

For a complete description of the dialog box, see the topic "Pointcloud/CAD Alignment Dialog Box Description".

3. A temporary split-screen view of the two pointclouds appears in the Graphic Display window. You can use this view to visually see the alignment taking place. Select the first COP that you want to use as a point of reference from the **Reference** list.



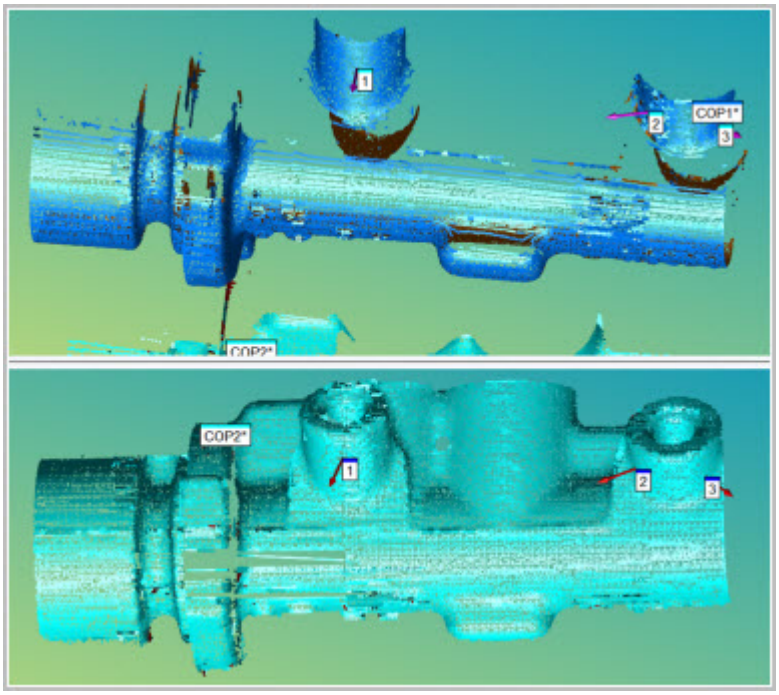
Split-screen view showing a pointcloud to pointcloud alignment

4. Use your mouse to manipulate and orient each view as needed to create the Point Pairs.

5. Perform the alignment:

   - First, use the **Point Pairs** area to perform a rough alignment that brings the pointclouds close enough to each other. This is a mandatory step.

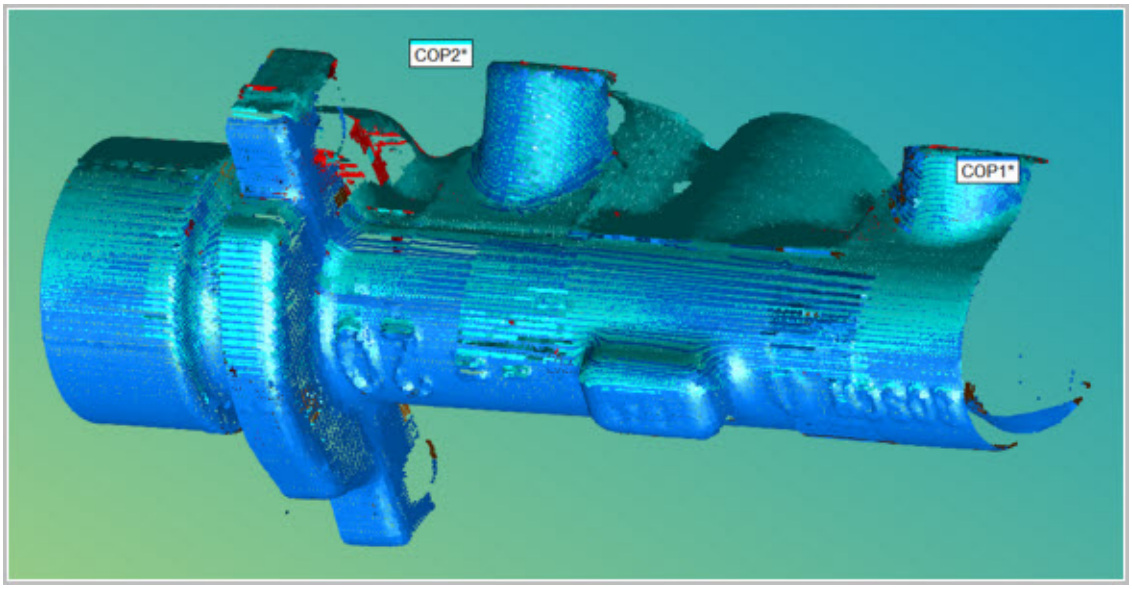
     - Click a desired number of points (at least three pairs) on each of the pointclouds within the overlap area. ONLY click points on the overlap area of the two pointclouds.



Split view showing selected COP1 and COP2 pointclouds

- The more points you take around the overlap area of the pointclouds results in an improved alignment. Click **Compute** to create the rough alignment.

- Next, use the **Refine Alignment** area whenever you want to refine your alignment. This brings the two pointclouds closer to each other. In order to get a good, refined alignment, the two pointcloud points should be close enough to each other through the initial rough alignment.



A sample rough pointcloud to pointcloud alignment that needs a refinement

- Define the maximum distance between the points in the two pointclouds using the **Maximum Distance** box. The default value is 0 (zero). If you

use the default value, PC-DMIS uses an internal default value related to the dimensions of the pointclouds.

- Click **Compute** to refine the alignment.

6. If a portion of one pointcloud doesn't align nicely with the other, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters, or you can try a different alignment.

7. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the `COPCOPBF` command in the Edit window. For details on the COPCOPBF command, see the "COPCOPBF Command Mode Text" topic in the PC-DMIS Laser documentation.

COPCOPBF Command Mode Text

The COPCOPBF command allows you to perform a best fit alignment of the reference pointcloud with a second pointcloud.

Below is an example code snippet for a COPCOPBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
   COPCOPBF/REFINE,SHOWALLPARAMS=TOG1,
   ROUGH ALIGNPAIR/
      THEO/<x,y,z>,<i,j,k>,
      MEAS/<x1,y1,z1>
   REF,TOG2,TOG3,,
ALIGNMENT/END
```

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
   ROUGH ALIGNPAIR/
      THEO/x,y,z,i,j,k,
      MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined and selected using the Graphic Display window. The values next to `THEO/` represent the point for the reference COP. The values next to `MEAS/` represent the corresponding point on the second COP. These pairs are used to determine a rough transformation

between the reference COP and the second COP which allows the two pointclouds to come close to allow further refinements of the alignment.

TOG2 determines the reference COP used for aligning to the second COP.

TOG3 determines the second COP used for the alignment back to the reference COP.

Note on Adding or Updating Alignments in the Edit Window

If you add an alignment before a Colormap or Clean COP command in the Edit window, or if you change an alignment above a Colormap in the Edit window, PC-DMIS displays a message to ask whether you want to update the alignment's dependent commands.

Select the best option for your measurement routine:

- **Yes** - This option converts the dependent commands to the new alignment coordinate system.
- **No** - This option doesn't change the dependent commands.

TCP/IP Pointcloud Server

PC-DMIS has several options that use TCP/IP communication to watch or connect to a third-party client.

Generic OFFLINE TCP/IP Import Pointcloud Function

This OFFLINE function allows you to import a pointcloud from a client application into PC-DMIS (the Server application). When PC-DMIS receives the new pointcloud data, it automatically executes the inspection routine offline. See "Generic Import File Formats".

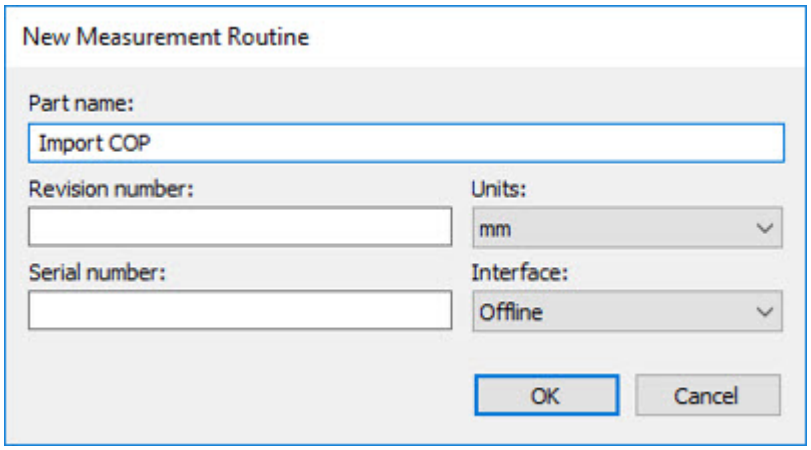
From the **Pointcloud** toolbar, click the **TCP/IP Pointcloud Server receive data** button  to put PC-DMIS in a "watch" state. When in this state, PC-DMIS is ready and waiting to receive a pointcloud file. The client application must initiate sending the pointcloud data. This button only appears when you run PC-DMIS in Offline mode. Click the button a second time to turn this function off.

When PC-DMIS detects a new pointcloud file:

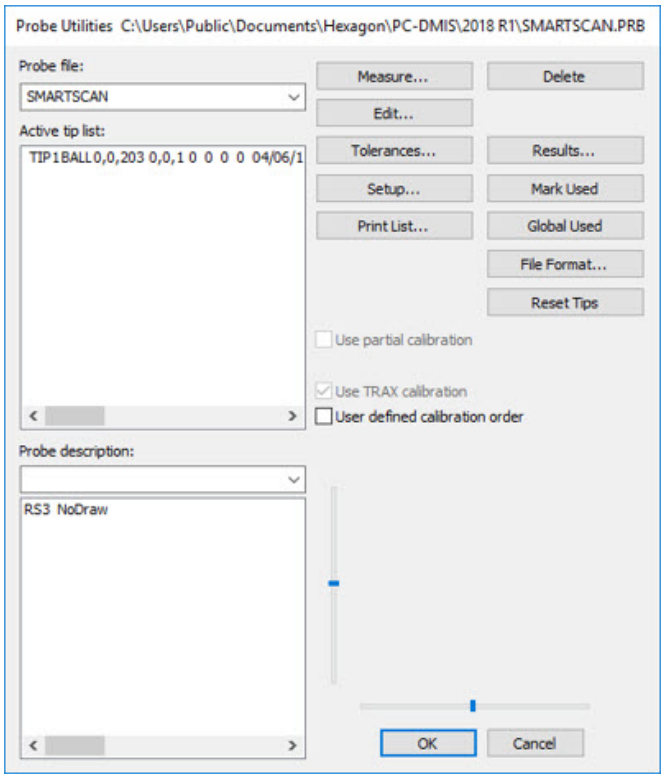
- If the measurement routine already contains a COP (cloud of points), PC-DMIS replaces the COP with the received data and then executes the measurement routine.
- If the measurement routine does not contain a COP, PC-DMIS creates a COP feature, imports the data, and then executes the measurement routine.

To create the initial measurement routine for OFFLINE execution:

1. Create the PC-DMIS measurement routine with the offline interface.



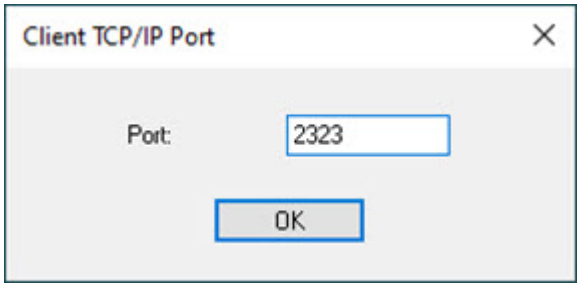
2. The software displays the **Probe Utilities** dialog box. Select SMARTSCAN as the active offline laser probe.



3. From the **Pointcloud** toolbar, select the **TCP/IP Operations** button and then the **TCP/IP Pointcloud Server receive data** button  .

> The **TCP/IP Pointcloud Server receive data** button is only available when PC-DMIS is run in Offline mode.

4. From the **Client TCP/IP Port** dialog box, enter the Port ID, and click **OK**. You can find the Port ID in the client application. The default Port ID is 2323.



5. PC-DMIS begins to import the pointcloud data as soon as the client application initiates the Send function. The software displays the progress of the incoming data in the PC-DMIS status area, which is located in the lower-left corner.

6. Create any needed pointcloud commands (for example, Pointcloud Alignment, Pointcloud Surface Colormap, etc.), Auto features, and Dimensions.

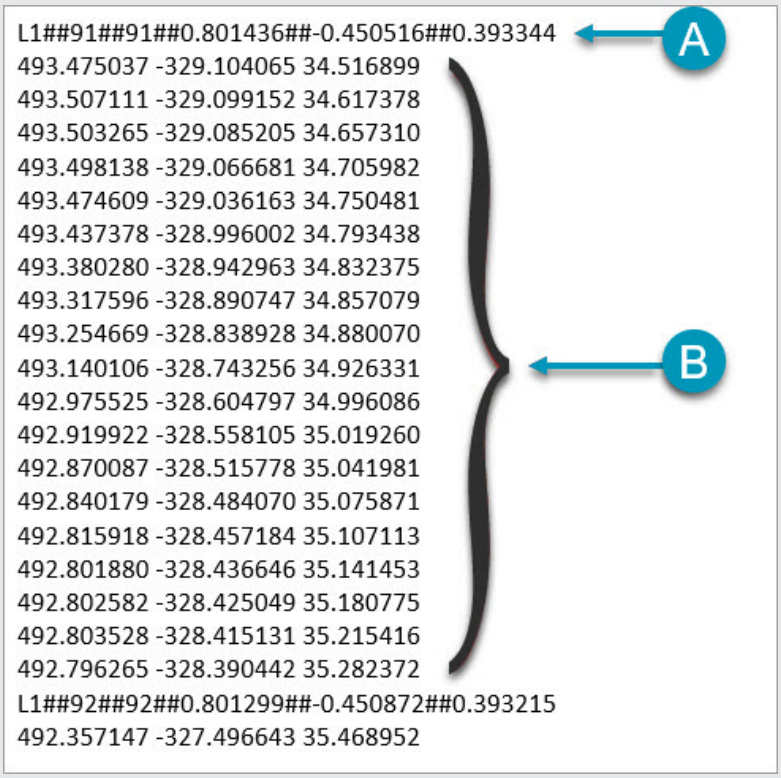
7. Save the measurement routine.

Generic Import File Formats

PC-DMIS allows the import of these pointcloud formats:

- Data per set of points

Data where each laser stripe, line or patch defines the vector (IJK) of the stripe, followed by XYZ value of the points on the stripe. Point XYZ values can be space-delimited or comma-delimited.

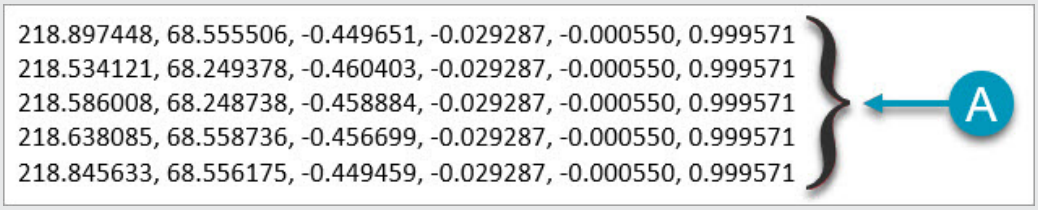


A - Line (laser stripe or patch) number unique identification number (optional) IJK of line (from sensor orientation)

B - XYZ value of points on the line

- Data point

The data file defines the XYZ or XYZIJK values for each point. For these data types, XYZIJK is preferred as PC-DMIS uses the vector of the points in pointcloud operations such as Surface Colormaps and Feature Extraction. The following example shows points with XYZIJK values.



A - XYZIJK value of each point

Generic ONLINE TCP/IP Export Pointcloud Functions

PC-DMIS can send your pointcloud data to a custom-built, third-party software application. It uses a TCP/IP communication protocol to do this. To establish the connection, your custom application must be able to load a dynamic link library (.dll) file named PcDmisPointCloudClientDll.dll. You can request this file from Hexagon Technical Support.

Once your application loads the .dll file, click on one of these TCP/IP Pointcloud server icons available from the PC-DMIS **Pointcloud** toolbar to establish the connection:



TCP/IP Pointcloud Server Connection with Local Copy - Establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data *remains inside* the measurement routine.



TCP/IP Pointcloud Server Connection without Local Copy - Establishes the connection with the client and sends the pointcloud data directly to the client. When the scan finishes, the pointcloud data *is deleted from* the measurement routine.

Extracting Auto Features from Pointclouds

Laser Auto features can be extracted from scanned pointcloud data. Once you configure the Auto features, you can then scan the part. PC-DMIS extracts the Auto feature information from the scan. You can include and extract multiple Auto features from a single pointcloud.

Review the following topics to execute Auto feature extraction from manual scans:

- Defining a laser Auto Feature by clicking on a Pointcloud
- Example of How to Use the Depth Option to Create an Auto Circle Feature
- Executing Scan-Extracted Auto Features
- Aligning Measured Auto Features to CAD

For an example of how PC-DMIS uses the **Depth** option to create an Auto Circle, see "Example of How to Use the Depth Option to Create an Auto Circle feature".

Defining a Laser Auto Feature by Clicking on a Pointcloud

Often, you click on the CAD to define an Auto feature. In the case where no CAD exists, you can perform a scan of the part, and then:

- Click on the individual Pointcloud points to define your Auto feature

- Or you can box-select the feature from the pointcloud

To define an Auto feature from Pointcloud points:

1. Scan the surface of the part in which the needed Auto feature exists.
2. Click the Auto feature from the **Auto Feature** toolbar or the **Insert | Feature | Auto** submenu. This opens the **Auto Feature** dialog box.
3. Either select points from the pointcloud that best define the feature's nominal position or drag a box directly on the pointcloud to have PC-DMIS extract the feature from the points within the dragged box. PC-DMIS defines the Auto feature based on your selection.

Defining Features by Selecting Points

The following table shows the number of points that are needed to define an Auto feature's location.

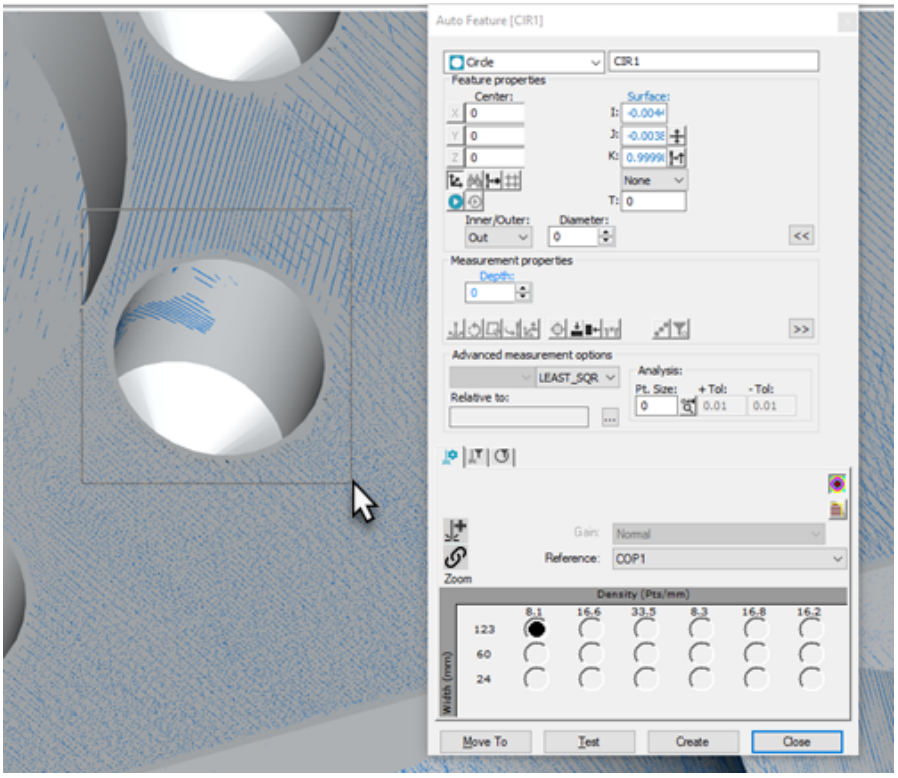
Feature	Points to Select
Surface Point	Select one point at the needed location within the measured surface area.
Edge Point	Select one point at the needed location along the measured edge.
Plane	Select at least three points that best define the needed plane's nominal position.
Circle	Select at least three points around the perimeter of the measured circle.
Round Slot	Select three points along one of the slot's arcs, then select another three points along the other arc.
Square Slot	Type the slot's nominal **Width** in the **Auto Feature** dialog box. Select two points along a long side of the slot. Select one point on a short side of the slot. Select one point on the other long side of the slot. Finally, select one point on the other short side of the slot.
Flush and Gap	Select a point on each side of the gap.
Cylinder	Select three points for each of two circles that define the extents of the cylinder's form and length.
Sphere	Select at least five points around the surface of the measured sphere.

Defining Features by Box-Selecting

In Learn mode, you can drag a box around the desired feature on the pointcloud to extract supported Auto features using the selected data points.

This functionality has these limitations:

- PC-DMIS only calculates the surface vector. You may need to define the angle vector manually, such as for a polygon feature.
- If your box selection includes points at multiple depths in the Z axis, it may result in a poor feature extraction. You can avoid this by either clipping the acquisition or by using `COP/OPER,SELECT` to exclude those points prior to the box selection.



Example Circle Auto feature creation by box-selecting

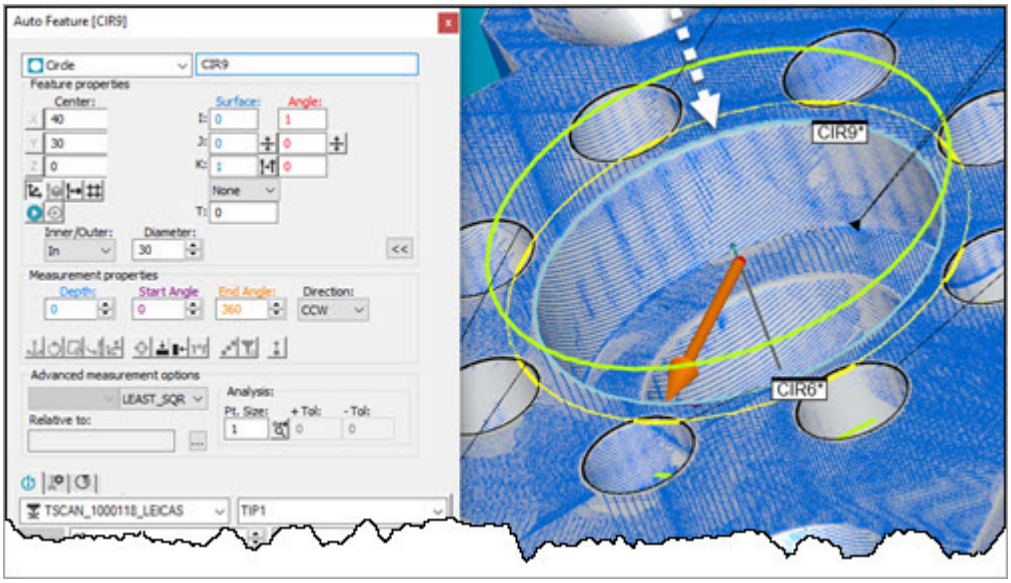
Box-selecting works with these supported features:

- Surface Point
- Plane
- Circle
- Round Slot
- Square Slot
- Sphere
- Polygon

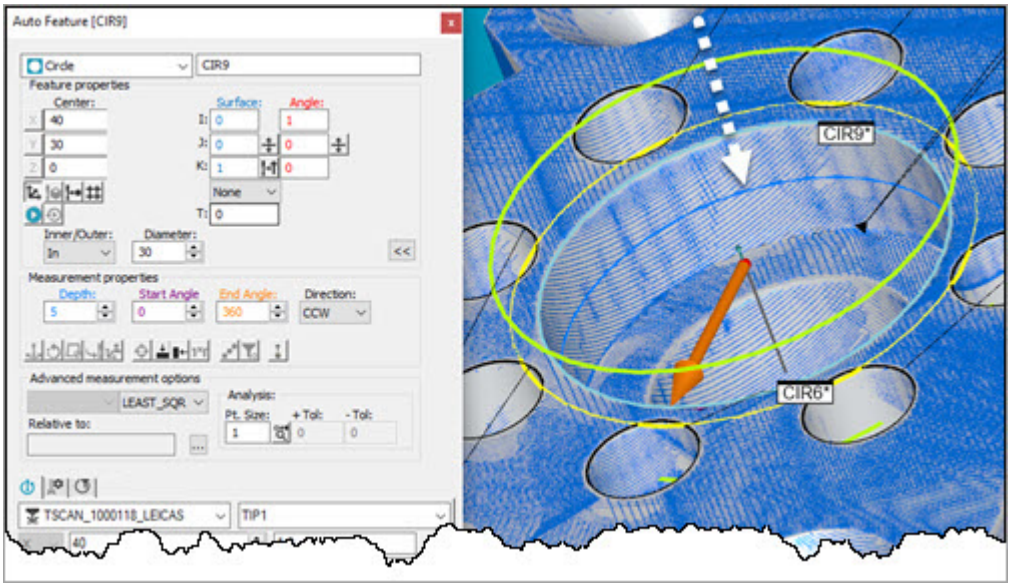
For all other Auto features, you must use the point selection method.

Example of How to Use the Depth Option to Create an Auto Circle Feature

For pointcloud data, if you set the **Depth** value to 0 (zero), PC-DMIS uses the points on the plane of the circle along the edge. The dotted white arrow points to where this is in the image below.



If you enter a **Depth** value of 5, for example, PC-DMIS takes the points 5 units below the plane of the circle and then projects them back up to the plane of the circle to solve it. The dotted white arrow points to where this is in the image below.



This may appear confusing if you turn the **Show Measured Points** button on. This is because PC-DMIS shows the hits at the plane of the circle that the points were projected to instead of at the **Depth** location. How PC-DMIS shows this is by design. The points you see are actually the same points taken at the **Depth** location, but they are projected to the

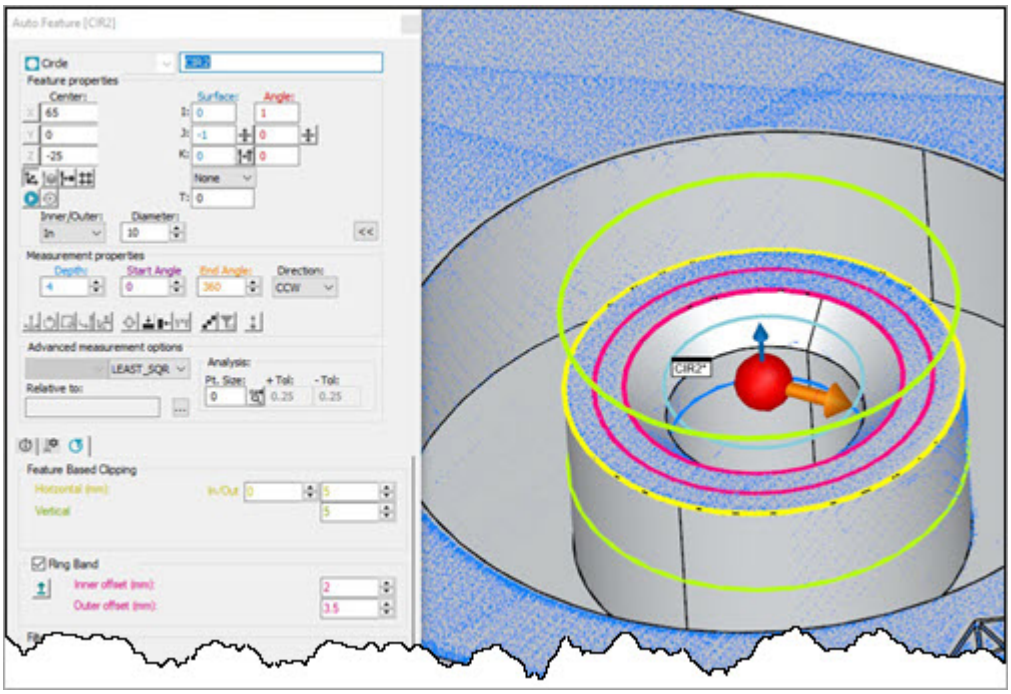
plane of the circle. You can test this if you try to create a circle at a depth where no points exist. The test will fail.

Application

As long as the edge of your circle has good definition, you do not need to set a **Depth** value. It can remain at zero. This is especially true because you need to scan points in a repeatable manner and doing so at a depth is more difficult.

Except in the case of chamfers or extruded holes, you can keep the **Depth** value set to 0 (zero).

In this example, **Depth** is set to 4 to take the measurement of the cylinder down and away from the chamfer. The Ringband is used to move the circle away from the chamfer.



Executing Scan-Extracted Auto Features

When executing manual scans from which Auto features are extracted, you should do the following:

1. Scan the Auto features in your measurement routine in any order. You can do this with one or more passes. After the first pass, if the scan's pointcloud points have changed for a feature, then the feature's measured values are recalculated.

2. When all the Auto features associated with the scan have solved successfully, the command in the Edit window is highlighted in yellow.

3. When Auto features have solved and have been reported correctly, the command in the Edit window is highlighted in green.

4. If additional scan data is taken for a feature that has already been solved, the feature's measured values are updated again with the new solution.

5. Once all of the included Auto Features have been solved, you may choose to continue scanning to further refine the measured results, or you can click the **Scan Done** button () from the **Execution** dialog box. You can also finish the scan by pressing the Done button on your measurement arm.



The **Scan Done** button is not available until all the included Auto features are successfully measured.

See the "Using Pointclouds" topic.

Aligning Measured Auto Features to CAD

The procedure is only available when you measure Auto Features with a manual laser sensor (on a portable arm) and with imported CAD data. This allows you to select the *actual* measured features from the pointcloud that correspond to the selected *nominal* features from the CAD.

To align measured Auto features to CAD nominals:

1. Import CAD data.

2. Open the **Auto Feature** dialog box for a feature you want to include in the manual alignment.

3. Select the nominal location for the feature. Click on the CAD surface next to the feature to do this.

4. Change any Auto feature parameters as needed and click **Create** to add the auto feature to the measurement routine.

5. Repeat steps 2 through 4 for each Auto feature you want to include in the alignment.



PC-DMIS automatically adds a new extraction COP when you begin to create a new laser Auto feature. You may include the features of the manual alignment in the same pointcloud. The Laser Probe Toolbox: Laser Scan Properties tab determines the COP from which the software extracts the laser Auto features.

6. Execute the measurement routine. PC-DMIS prompts you to scan the laser Auto features as part of a Portable Laser Alignment.

7. Scan the part to include the Auto features for the manual alignment. You may need to perform more than one scan to adequately define each feature.
8. Press the **Done** button on your measurement arm when you have completed scans.
9. PC-DMIS now prompts you to define the first manual alignment feature. Follow the instructions provided in the dialog box and status bar and then click **OK**. At the end of the selection, the software displays the preliminary form of the Auto feature.
10. Repeat Step 9 for each of the manual alignment features.

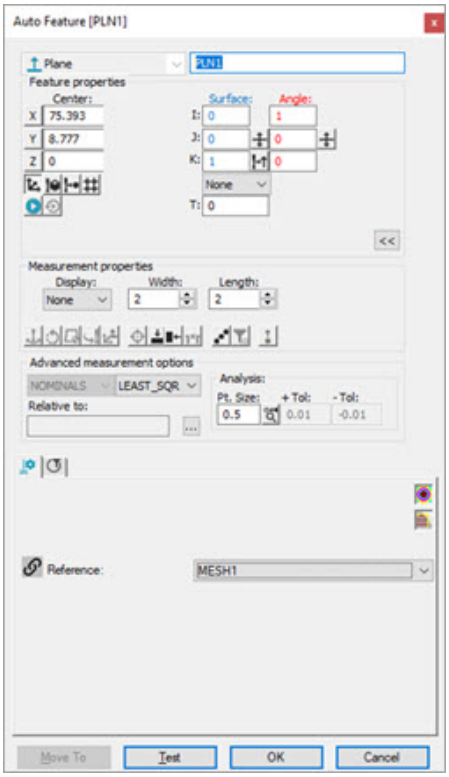
> PC-DMIS solves the laser auto feature with the theoretical values from the CAD and actual values from the measured pointcloud.

11. Select the **Insert | Alignment | New** menu item (Ctrl + Alt + A) to open the **Alignment Utilities** dialog box.
12. From the list box, select the alignment features, and click **Auto Align**. PC-DMIS aligns the defined features from the pointcloud with the corresponding CAD nominals. This establishes the manual laser alignment.

Extracting Auto Features from a Mesh

You can extract a laser Auto feature from a Mesh data object with the laser **Auto Feature** dialog box.

> For details about extracting laser Auto Surface Points from a Mesh data object, see "Extracting a laser Auto Surface Point from a Mesh".



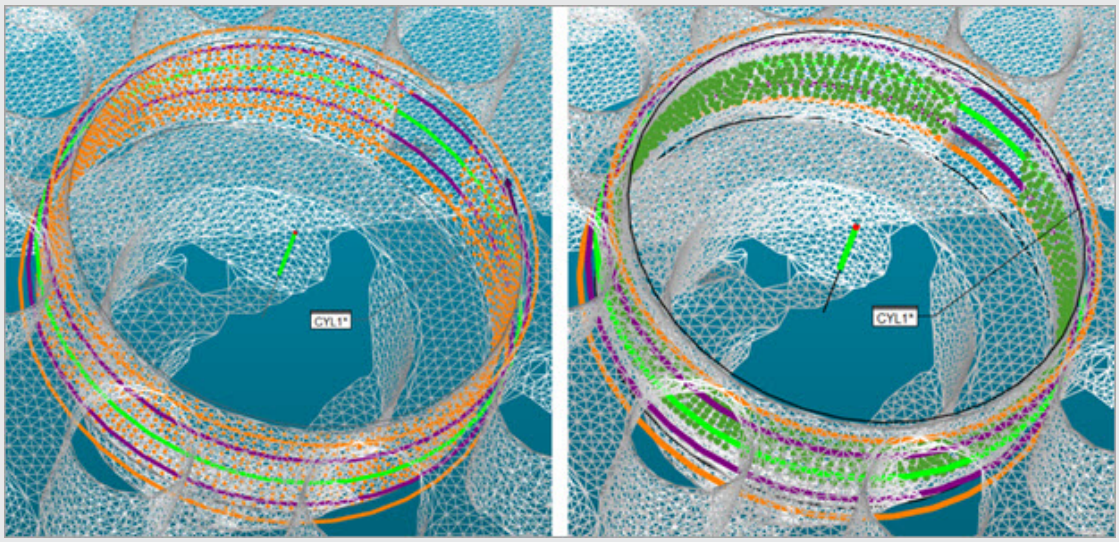
If there's only one Mesh in your measurement routine, PC-DMIS defaults to that Mesh data object as your reference data object. If there's a COP (or more than one COP) and one or more Mesh data objects, you need to select the correct reference data object from the **Reference** list of the **Feature Extraction** tab in the Probe Toolbox.

When PC-DMIS extracts the laser Auto feature from a Mesh data object, it first considers all the triangle vertices inside the extraction zone, defined by the horizontal and vertical clipping. To view the points that fall within the extraction zone, click the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab.

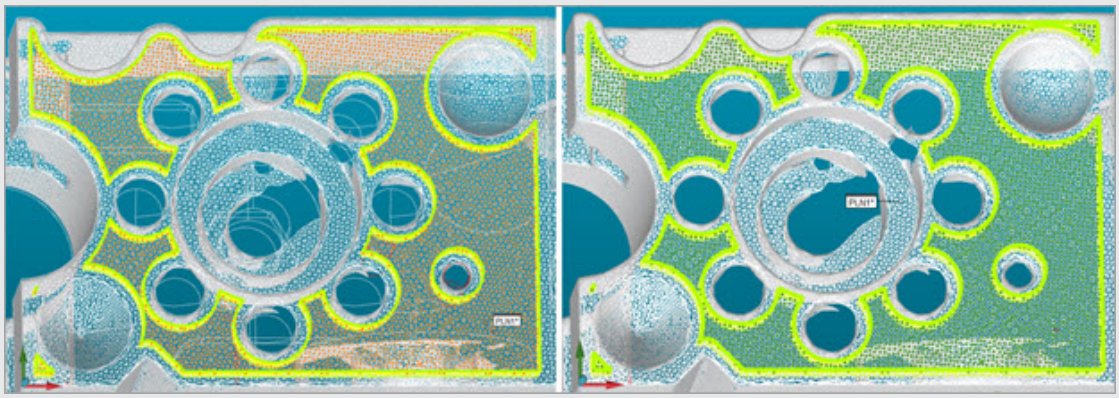
Click the **Test** button to measure the feature and view its measured points.

Examples of features extracted from a Mesh data object

You can see in the images below examples of extracting features from a Mesh object.



Example of a Cylinder Auto feature extracted from a mesh data object



Example of a Plane Auto feature extracted from a mesh data object

The orange points represent segregated points found within the extraction zone, while the green points represent measured points after PC-DMIS performs the test operation when you click the **Test** button.

Extracting a Laser Auto Surface Point from a Mesh

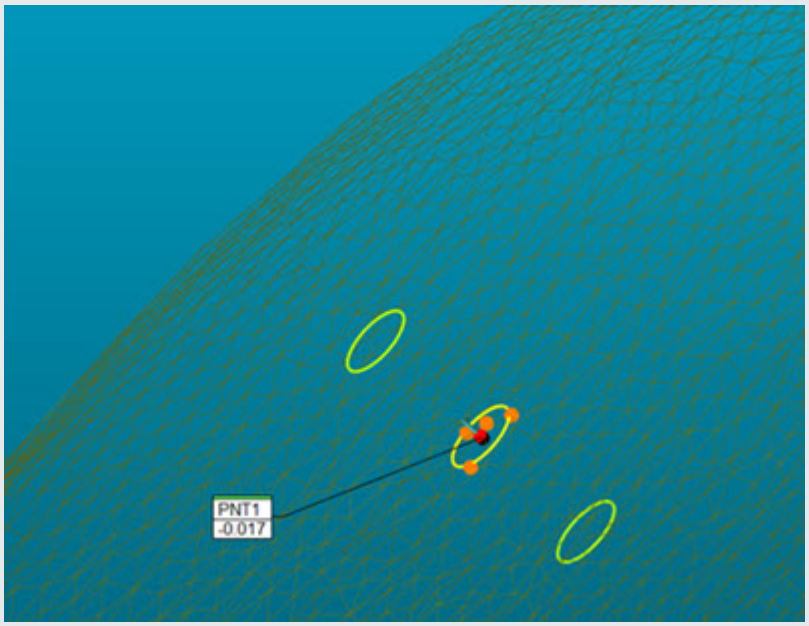
You can extract a laser Auto Surface Point from a mesh data object with the **Laser Auto Surface Point** dialog box.

When you extract the laser Auto Surface Point from a Mesh data object, PC-DMIS first considers all the triangle vertices inside the extraction zone, defined by the horizontal and

vertical clipping. To view the points that fall within the extraction zone, click the **Show/Hide segregated points** button () on the **Laser Scan Properties** tab.

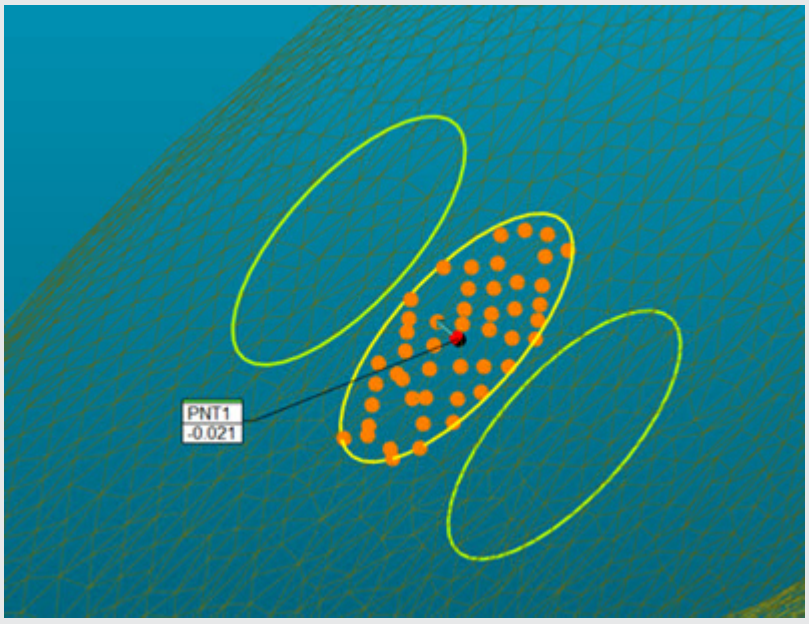
To get a more accurate result on a curved surface when you extract an Auto Surface Point from a Mesh, use a smaller horizontal clipping zone to limit the points (vertices) that PC-DMIS uses to calculate the measured value.

For example, when you use a small clipping zone, PC-DMIS uses points close to the nominal location to calculate the deviation. This results in a more accurate measurement on a curved surface:



Surface point with small (0.25 mm) horizontal clipping

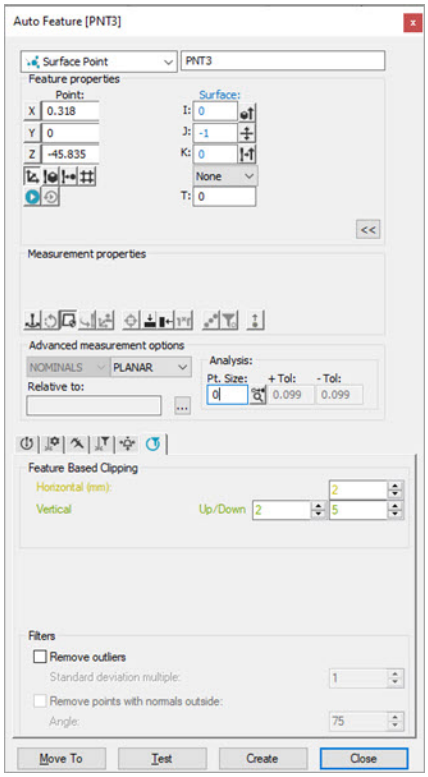
However, if you use a larger horizontal clipping zone, PC-DMIS uses more points to calculate the deviation. Avoid this when you measure points on a curved surface.



Surface point with larger (1 mm) horizontal clipping zone

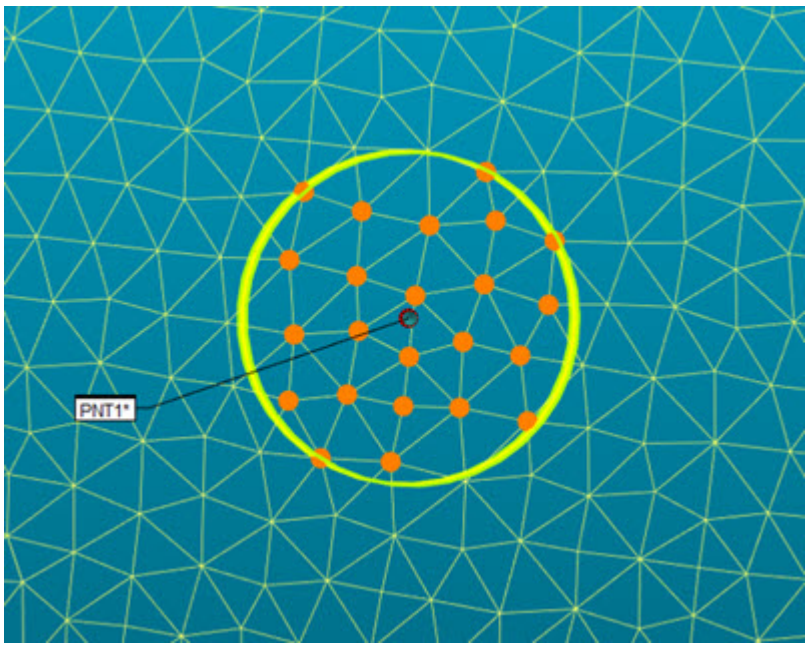
To extract a Surface Point from an existing Mesh:

1. Click the **Surface** menu option (**Insert | Feature | Auto | Point**) to open the **Auto Feature** dialog box. If you don't see the advanced options in the dialog box, click the **Show advanced measurement options** button.



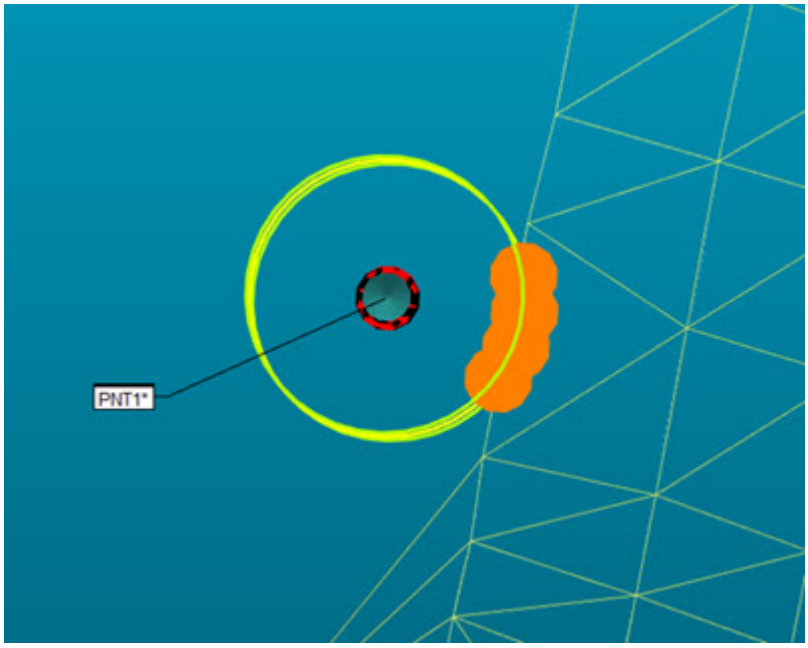
Auto Feature dialog box for Surface Point with the advanced measurement options

2. Select the **Mesh** reference for the surface point from the **Reference** list.
3. From the Graphic Display window, click the CAD to select the point nominal location and vector.
4. Click the **Show/Hide segregated points** button to view the points which fall within the extraction zone.



Example of the extracted points that fall within the extraction zone

If the number of vertices within the zone is less than three, then the clipping zone will intersect the Mesh and use the intersection points for the Auto Surface Point feature measurement.



Example of the extracted points that fall within the extraction zone with less than three vertices

5. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Region Properties** tabs to enter the information.

6. If desired, click the **Test** button to test the feature.

> **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.

7. Click **Create** and then **Close**.

Creating Auto Features with a Laser Sensor

With PC-DMIS Laser, you can use your laser sensor to create these Auto features:

- Laser Surface Point
- Laser Edge Point
- Laser High Point
- Laser Plane
- Laser Circle
- Laser Slot
- Laser Flush and Gap
- Laser Polygon
- Laser Cylinder
- Laser Cone
- Laser Sphere

> This topic only discusses Auto features as they pertain to laser sensor operations. For detailed information on Auto features, see the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

Implementation of QuickFeatures in PC-DMIS Laser

In order to cleanly implement the QuickFeature functionality, rules must be applied when switching between certain feature types that have the Inner/Outer options (Laser Circle, Laser Round Slot, Laser Square Slot, Laser Cylinder, Laser Cone, and Laser Sphere, for example).

> This functionality is not available for Flush and Gap features since the hovering mouse functionality is not available for this feature type.

Since the **Inner** option enables LEAST_SQR and MAX_INSC and the **Outer** option enables LEAST_SQR and MIN_CIRCSC, the following rules apply:

- Whenever the **Inner/Outer** option that is selected in the dialog box as the default matches the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature.
- When the **Inner/Outer** option that is selected in the dialog box as the default doesn't match the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature only if LEAST_SQR was set as a default. In all the other cases, the created feature will have the Inner/Outer information coming from the CAD and the best fitting algorithm option set to LEAST_SQR.

For example, if you set Outer Circle as the default and MIN_CIRCSC as the best fitting algorithm, and then you quick select an inner circle, you'll get an Inner Circle with LEAST_SQR option as result.

For more information on ways to create QuickFeatures, see the "Quick Ways to Create Auto Features" topic in the "Creating Auto Features" chapter of the PC-DMIS Core documentation.

Extracting QuickFeatures or Auto Features when Created and Linked to a COP

When you use a Portable or CMM interface with an existing Pointcloud (COP) that is aligned to a CAD model, PC-DMIS extracts and measures the laser Auto feature when you create the feature.

This applies to the following:

- When you create a laser Auto feature with the QuickFeature method (Shift + Click).
- When you create a laser Auto feature from the **Auto Feature** dialog box (**Insert | Feature | Auto**).



This function is not available for directly-measured laser Auto features.

Extracting with CAD data

When you are measuring a part for which CAD data has been imported, you can extract Auto features by doing the following:

1. Scan the surface of the part in which the needed Auto feature exists.
2. Click on the CAD surface near the edge of the needed feature. PC-DMIS solves the Auto feature with nominal data from the CAD and measured data from the Pointcloud data that is closest to the feature nominals.

3. Continue selecting the needed Auto features that reside within the Pointcloud.

Common Laser Auto Feature Dialog Box Options

In PC-DMIS Laser, the **Auto Feature** dialog box works alongside the Probe Toolbox to create a complete laser Auto feature command. To edit an Auto feature, you can use the Edit window and modify the command there, or you can change parameters inside the **Auto Feature** dialog box and the Probe Toolbox. For information on the toolbox, see "Using the Probe Toolbox in PC-DMIS Laser".

The following **Auto Feature** dialog box options are common to all of the supported laser Auto feature types and are discussed briefly for each dialog box area.

- Feature Properties Area
- Measurement Properties Area
- Advanced Measurement Options Area
- Command Buttons
- Directly Measured Laser Auto Features

For additional information, see "The Auto Feature Dialog Box" in the "Creating Auto Features" chapter in the PC-DMIS Core documentation.

Options used for specific Auto features are discussed in their respective sections.

Feature Properties Area

XYZ Center or Point - These boxes show the feature's XYZ center or point location in part coordinates.

IJK Surface, Edge, Slot, or Gap Dir (Vector) - These boxes allow you to set the surface normal vector, edge vector, slot vector, or gap direction of the feature.

IJK Angle Vector - These boxes allow you to define the feature's secondary vector. This helps control the orientation of the feature.

Polar/Cartesian - This button toggles the display between Polar and Cartesian modes.

Find Nearest CAD - When you select an axis (X, Y, or Z) from one of the **Center** boxes and click this button, PC-DMIS finds the closest CAD element in the Graphic Display window to that axis.

Point Read From Machine - When you click this button, PC-DMIS uses the XYZ location of the machine for the feature's XYZ coordinates.

Find Vector(s) - This button pierces all surfaces along the XYZ point and IJK vector looking for the closest point. The software displays the surface normal vector as the IJK NOM VEC but the XYZ values do not change.

> This option is only available for Surface and Edge Point features.

Flip Vector - This button flips the surface normal vector. For example, 0,0,1 would flip to 0,0,-1.

Thickness - This field (**T**) applies a thickness to the feature. You can specify whether to use actual or theoretical values and then enter the value for the thickness.

Swap Vectors - Click this button to switch the current edge vector and surface vector with each other.

> This option is only available for Edge Point features.

Measure Now - This button determines whether or not PC-DMIS measures the feature when you click **Create**.

Re-Measure - This button determines whether or not PC-DMIS automatically re-measures the feature a second time once you measure the feature. PC-DMIS uses the measured values from the first measurement as the target locations for the second measurement.

> This is only available for Circle, Cylinder, Square Slot, Round Slot, and Notch features, and you must be in DCC Mode.

Measurement Properties Area

For information about the specific parameters configured in this section, see the following topics:

- Edge Point Specific Parameters
- Plane Specific Parameters
- Circle Specific Parameters
- Slot Specific Parameters
- Flush and Gap Specific Parameters
- Cylinder Specific Parameters

- Sphere Specific Parameters

Auto Wrist - This button causes the probe orientation to move to a vector that closely corresponds to the surface vector of the Auto feature.

View Normal - Click this button to orient the CAD so that you look down on the feature.

View Perpendicular - Click this button to orient the CAD so that you look at the side of the feature.

Advanced Measurement Options Area

Best Fit Math Type

Specifies the method that PC-DMIS uses to calculate the feature.

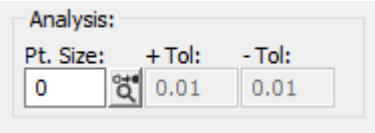
The following options are available for laser Auto Circle and Auto Cylinder: **LEAST_SQR**, **MAX_INSC**, and **MIN_CIRCSC**.

For more information about the calculation types, see "Best Fit Type for Circle" in the "Constructing New Features from Existing Features" chapter of the PC-DMIS Core documentation.

Relative to

This option allows you to keep the relative position and orientation between a given feature (or features) and the Auto feature. Click the ... button to open the **Relative Feature** dialog box to select the feature or features to which the Auto feature is relative. Multiple features can be defined for each axis (XYZ) relative to your Auto feature.

Analysis Area



The **Analysis** area allows you to determine how each measured hit/point is displayed.

Pt. Size - This value determines how big the measured points are drawn in the **CAD** tab. This value specifies the diameter in the current units (mm or inch).

Graphic Analysis button - When this button is switched on, PC-DMIS does a tolerance check on each point (how far from the calculated actual feature it is) and draws it in the appropriate color based on the current defined color range for the dimension.

+ Tol - This option provides the positive tolerance from the nominal. It is specified in the current measurement routine's units. Points that are greater than this value from the nominal are colored based on the standard PC-DMIS positive-tolerance color.

- Tol - This option provides the negative tolerance from the nominal. It is specified in the current measurement routine's units. Points that are less than this value from the nominal are colored based on the standard PC-DMIS negative-tolerance color.

For information on editing dimension colors for the positive and negative tolerances, see the "Editing Dimension Colors" topic in the "Editing the CAD Display" chapter in the PC-DMIS Core documentation.

Command Buttons

>> - This button expands the **Auto Feature** dialog box to show additional, more advanced, Auto feature options.

<< - This button hides the more complex features of the **Auto Feature** dialog box.

Move To - This button moves the field of view of the Graphic Display window and centers it on the feature's XYZ location. If the feature is comprised of more than one point (such as a line), then clicking this button switches between the points that make up the feature. For a laser Slot Auto feature, the field of view moves to the center of the slot feature.

Test - This button tests the Auto feature before PC-DMIS creates it. For laser features, the machine scans over the feature and calculates the measured value for the feature.

Create - This button creates the Auto feature. The **Auto Feature** dialog box remains open.

Close - This button closes the **Auto Feature** dialog box without creating a feature.

Directly Measured Laser Auto Features

The **Reference** parameter, found on the **Laser Scan Properties** tab of the laser **Auto Feature** dialog box, defines the pointcloud or mesh from which PC-DMIS extracts the Auto feature. If you select the **Disabled** option from the list, you can scan the feature directly. The software stores the scanned stripes in an internal COP. This is referred to as a "Directly Measured Laser Auto Feature".

When you run PC-DMIS in either Online or Offline mode, the internal scanned stripes are visible in the Graphic Display window only while the laser **Auto Feature** dialog box is open, and you enable the **Show/Hide Stripes** button  . When you close the dialog box, the scan stripes are no longer visible. After you create the Auto feature and press F9 to edit the Directly Measured laser Auto feature, the stripes become visible again.



You can only use the **Disabled** parameter in DCC Mode.

Online

When you run PC-DMIS in Online mode with a CMM, you can directly measure a laser Auto feature. To do this, you must set the **Reference** parameter to **Disabled**.

WARNING – When you select the **Disabled** parameter with the machine online, and the **Measure Now** button is selected, the machine moves to the feature and begins the scan using the selected settings as soon as you click the **Create** or **OK** button.

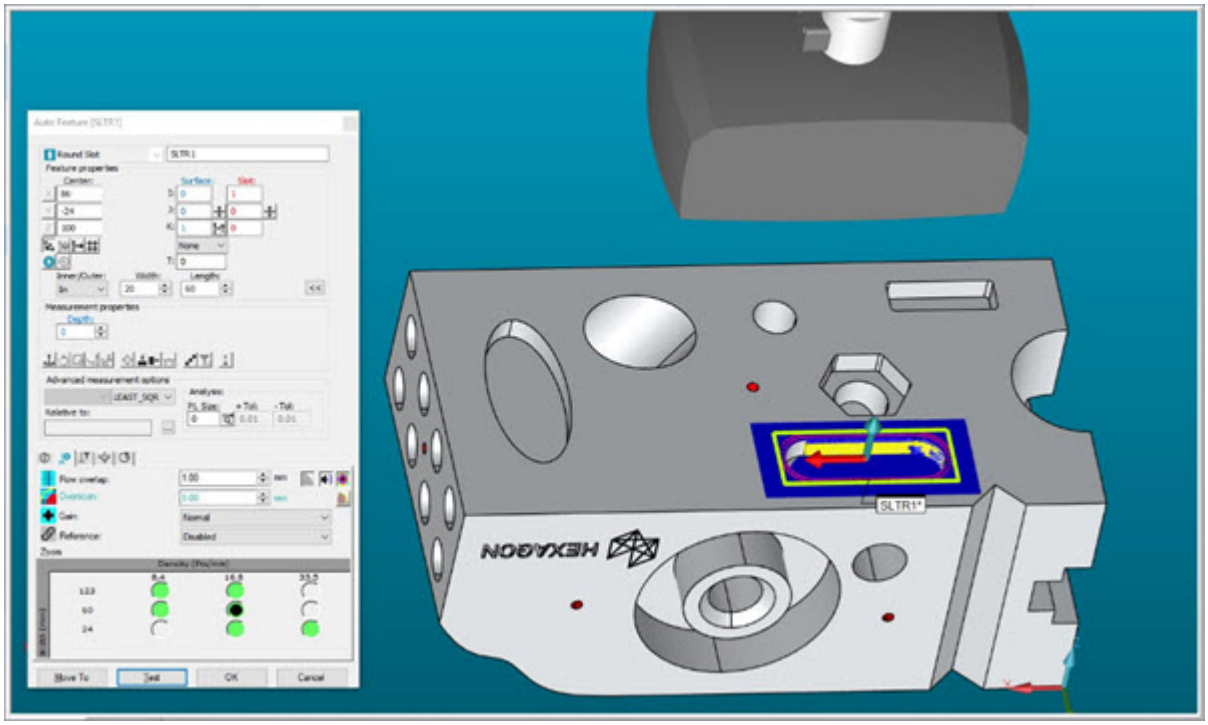
When online with the CMM and you click the **Test** button, the machine moves to the feature and begins the scan.

Offline

When you run PC-DMIS in Offline mode, you can simulate a Directly Measured laser Auto feature, check the scan settings, and adjust them as needed without running the machine.

To simulate a Directly Measured laser Auto feature:

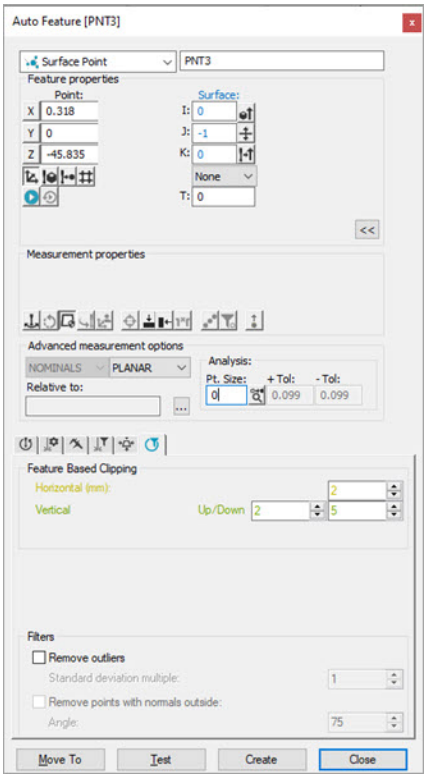
1. Start PC-DMIS in Offline mode.
2. Select the **DCC Mode** option from the **Probe Mode** toolbar (**View | Toolbars | Probe Mode**).
3. Open the **Auto Feature** dialog box (**Insert | Feature | Auto**) and select the feature you want to create.
4. Select the **Disabled** option from the **Reference** list.
5. Click the **Show/Hide Stripes** button  to view the simulated stripes.
6. Click the **Test** button to preview the internal scan stripes by projecting them as simulated scan stripes on the CAD model.



Example of a directly measured Laser Auto feature with simulated scan lines displayed offline

Laser Surface Point

Three methods exist to calculate the laser surface point: Planar, Spherical, and Extended Surface Point. For more information on these methods, see "Calculation Methods".



Auto Feature dialog box - Surface Point

To measure a laser surface point with a laser sensor, do the following:

1. From the menu, click **Insert | Feature | Auto** and select **Surface Point** to open the **Auto Features** dialog box for the surface point feature.
2. Do one of the following:
   - Click the CAD in the Graphic Display window to give the point a location and vector. Type in any remaining information.
   - From the Graphic Display window, use the **Laser** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Type in any remaining information.
   - Type in the theoretical information for X, Y, Z, I, J, K, and so on.
3. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.
4. Click the **Test** button to test the feature before you create it.

   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.
5. Click **Create** and then **Close**.

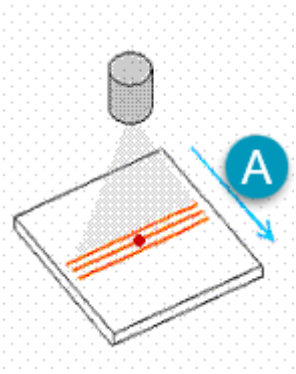
Surface Point Command Mode Text

The Auto Surface Point command in the Edit window in Command mode looks like this:

```
PNT1 =FEAT/LASER/SURFACE POINT,CARTESIAN
   THEO/<1.895,1.91,1>,<0,0,1>
   ACTL/<1.895,1.91,1>,<0,0,1>
   TARG/<1.895,1.91,1>,<0,0,1>
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THEO_THICKNESS,1
      MEASURE MODE=NOMINALS
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
      CONDITION INDICATOR=0,9
      FEATURE LOCATOR=NO,NO,""
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
      FILTER=NONE
```

Auto Surface or High Point Path

PC-DMIS determines the direction of the path for the Surface Point or High Point on the stripe.



Path direction of scan for surface or high point
(A) - Scan motion

Calculation Methods

There are three methods for calculating laser surface points:

- Planar
- Spherical
- Extended Surface Point

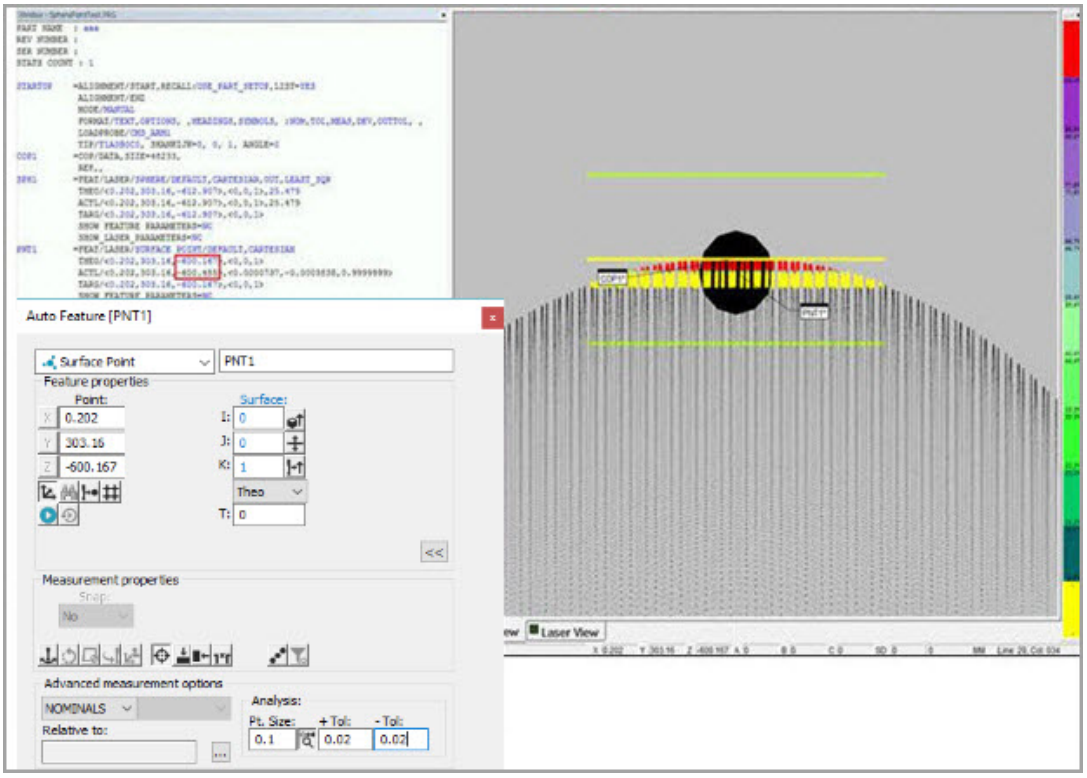
To change the calculation method, do one of the following:

- From the **Advanced measurement options** area in the Probe Toolbox, select the **PLANAR**, **SPHERICAL**, or **EXTENDED** option.

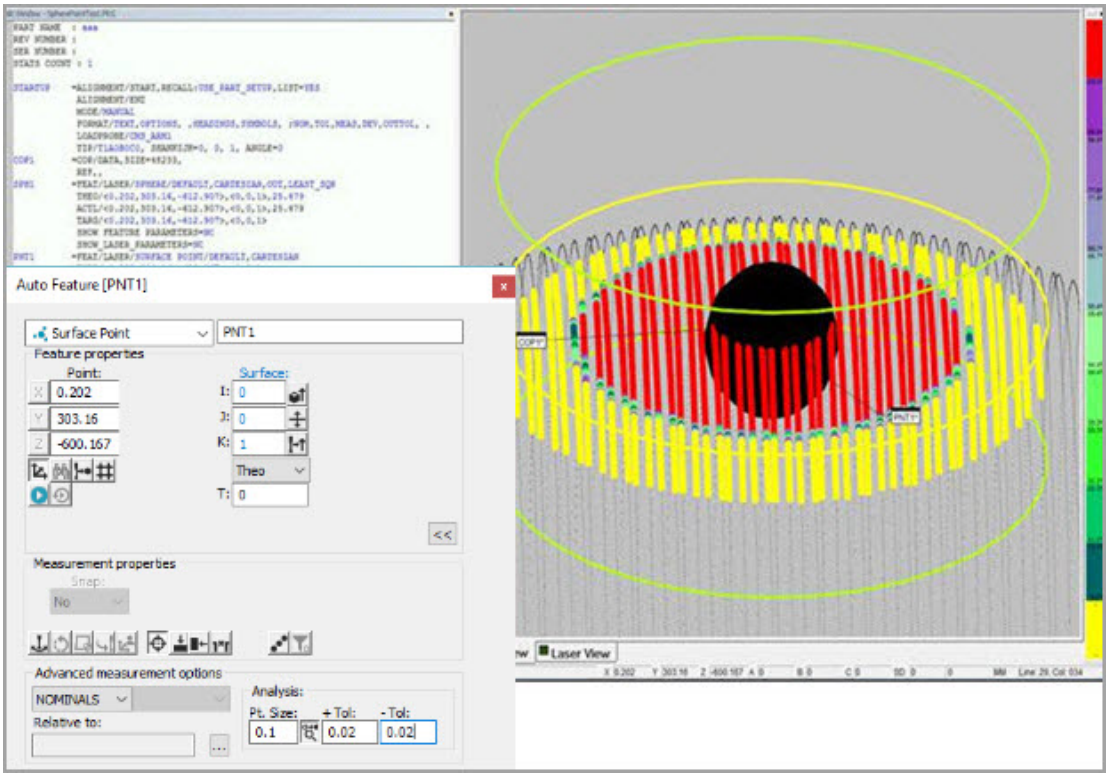
- In the **AutoFeatures** section of the PC-DMIS Settings Editor, modify the `Algorithm_Surface Point Laser` entry. For information about this entry, launch your PC-DMIS Settings Editor and press F1 to open its Help file. For more information, see the PC-DMIS Settings Editor documentation.

Planar Surface Point Calculation Method

This method calculates the laser surface point by fitting a local plane on the scan points within the circular area defined by the horizontal and vertical clipping parameters; this is the default method. The following is an example and its details:



Planar surface point example

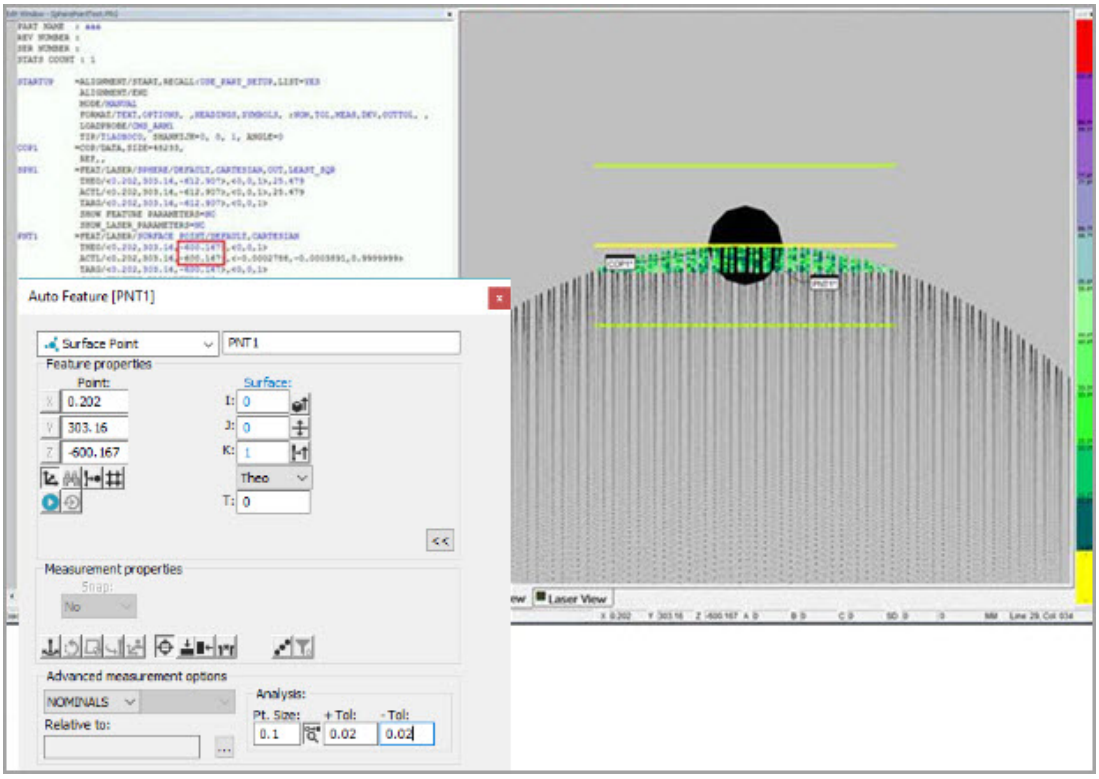


Planar surface point example - details

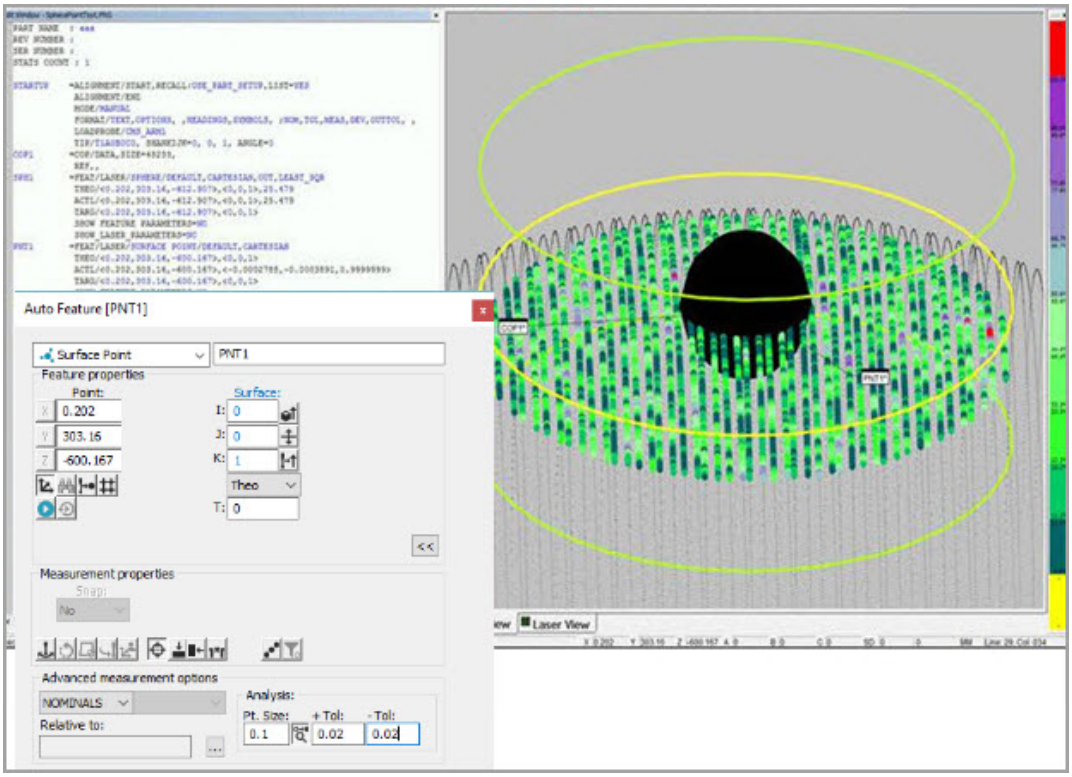
Spherical Surface Point Calculation Method

This method calculates the laser surface point by fitting a local sphere on the scan points within the circular area defined by the horizontal and vertical clipping parameters.

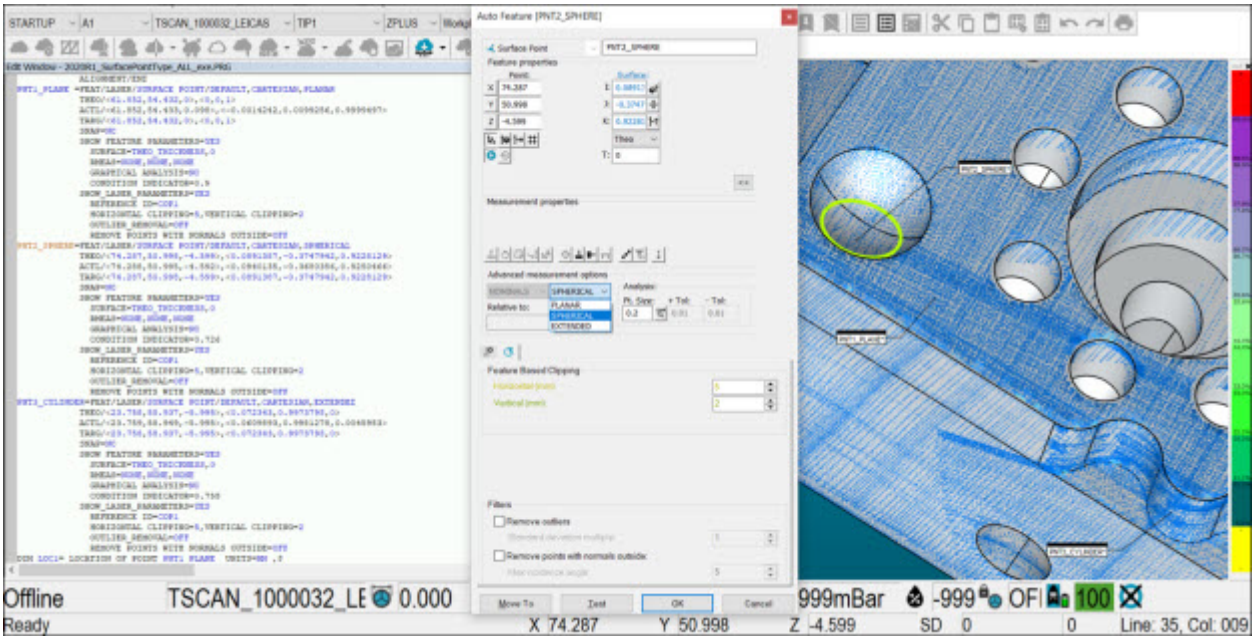
The following are examples with details:



Spherical surface point example



Spherical surface point example - details



Spherical surface point example - Condition Indicator

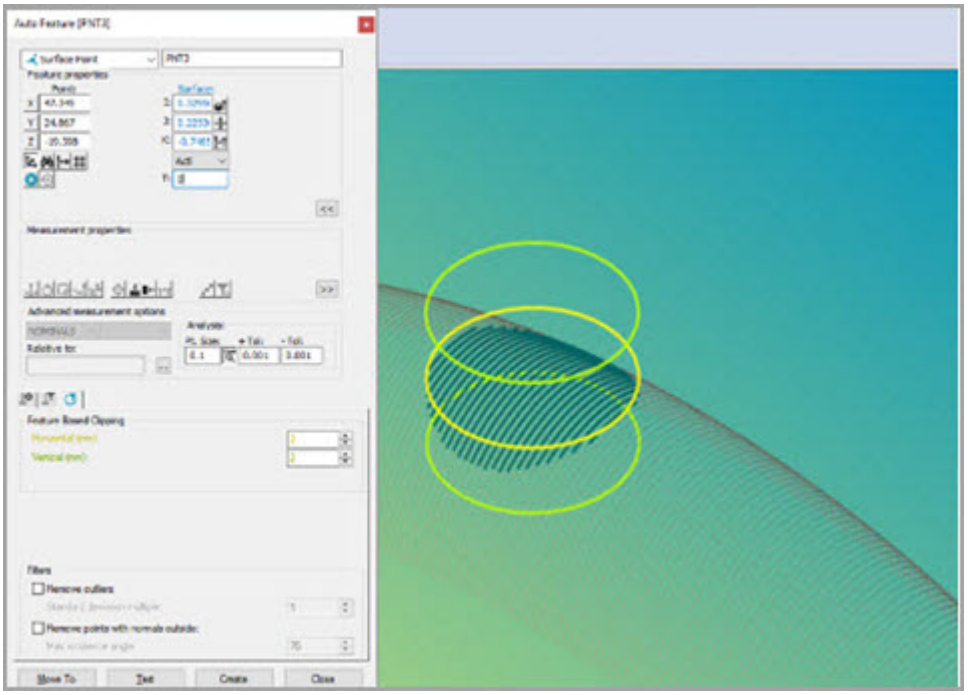
Extended Surface Point Calculation Method

This algorithm is able to calculate the Surface Point by fitting a local 2-curvatures manifold on the scan points within the circular area defined by the horizontal and vertical clipping parameters.

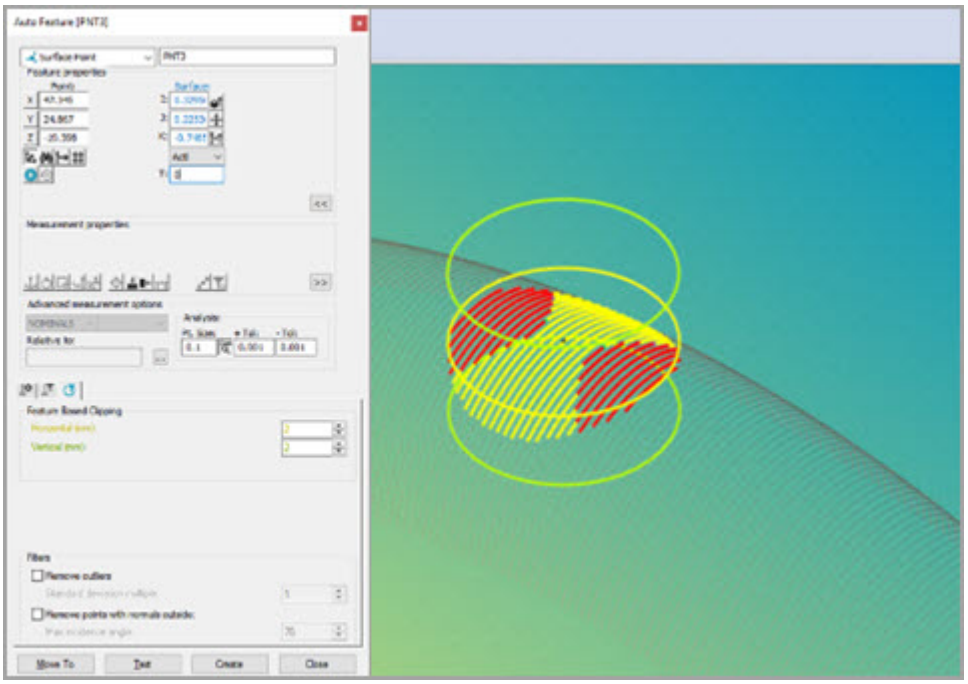
This method is especially useful when you need to calculate surface points on fillet surfaces.

The pictures below show the comparative results of algorithms applied to a point on a 2-curvatures filleted surface for the following:

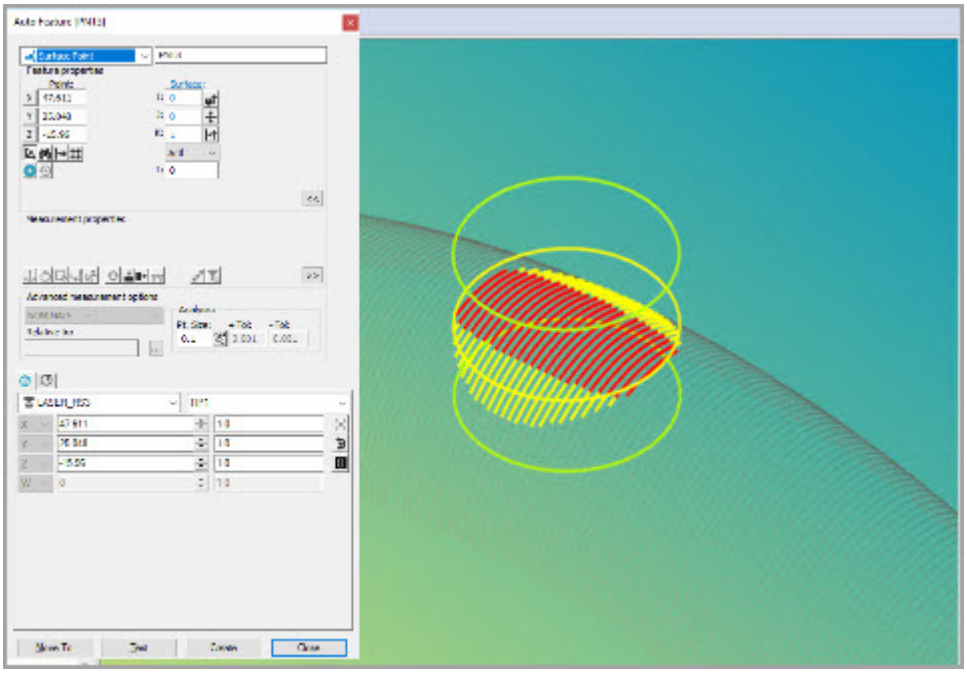
- Extended Surface Point
- Extended Spherical Surface Point
- Extended Planar Surface



Extended surface point details



Extended spherical surface point details



Extended planar surface point details

If a log file is enabled, additional results from the calculation of Extended Surface Points are available in the WaiFE_Debug.txt file found in the C:\ProgramData\Hexagon\PC-DMIS\2026.2\NCSensorsLogs\FeatureExtractor folder:

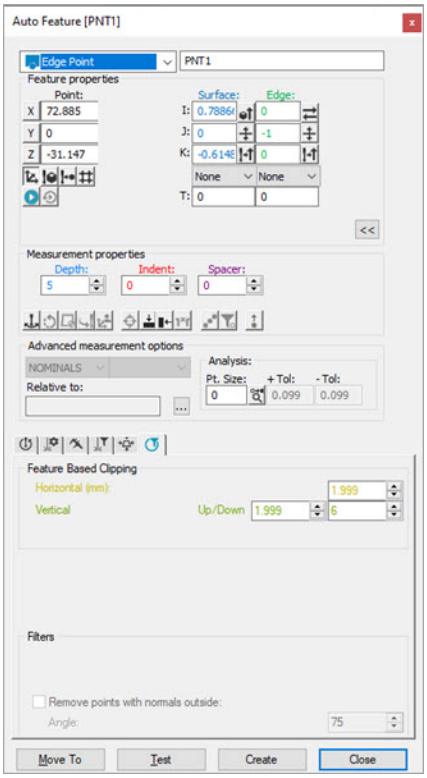
```
-------------- SURFACE POINT - begin: --------------

               TYPE: EXTENDED
               ACTUAL LOCAL CURVATURES: -0.028572 :  -0.200001
               ACTUAL SURFACE POINT:                i= 47.141291, j= 24.067065, k= -10.597570
               ACTUAL SURFACE VECTOR:               i= 0.553249557, j= 0.232507664, k= -0.799909441
               ACTUAL PRINCIPAL CURVATURE VECTOR: i= -0.832996099, j= 0.147852741, k= -0.533157637
               ACTUAL SECONDARY CURVATURE VECTOR: i= -0.005694434, j= 0.961290671, k= 0.275477440
               STANDARD DEVIATION: 0.000001
               CONDITION INDICATOR: 0.810149

-------------- SURFACE POINT - end --------------
```

The Condition Indicator (CI) value is a number from 0 (zero) to 1 inclusive. It indicates the quality of the points distribution. 0 (zero) indicates a poor distribution, and 1 indicates a good distribution. Generally, a value greater than 0.4 is considered to be acceptable.

Laser Edge Point



Auto Feature dialog box - Edge Point

To measure an edge point with a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Edge Point** to open the **Auto Features** dialog box for the edge point feature.
2. Do one of the following:
    - Perform clicks on the CAD to give the point a location and vector. Type in any remaining information.
    - From the Graphic Display window, use the **Laser** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Type in any remaining information.
    - Type in all of the theoretical information for X, Y, Z, I, J, K, and other parameters.
3. From the **Contact Path Properties** tab of the **Probe Toolbox**, specify values for **Depth**, **Indent**, and **Spacer**. PC-DMIS displays a corresponding graphical visualization of the change in the Graphic Display window.
4. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

5. Click the **Test** button to test the feature before you create it.

> **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.

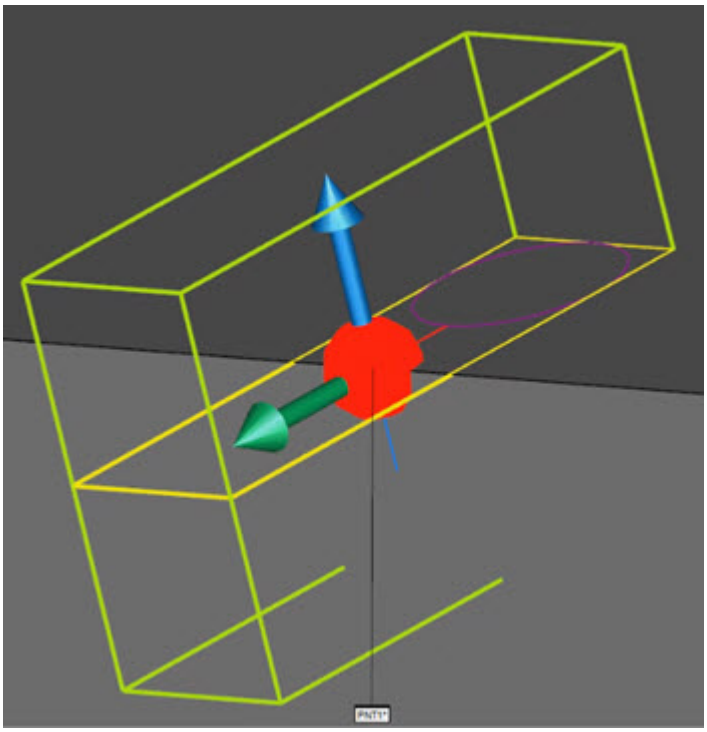
6. Click **Create** and then **Close**.

Edge Point Specific Parameters

Depth: This defines the depth to use when calculating the edge point. This corresponds to the blue graphical visualization in the Graphic Display window. A depth of 0 will cause 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 will cause it to be calculated at that depth.

Spacer: This controls the size of the area PC-DMIS uses to calculate the feature normal. This corresponds to the purple graphical visualization in the Graphic Display window.

Indent: This lets you define the location of the area PC-DMIS uses to calculate the feature normal. This corresponds to the red graphical visualization in the Graphic Display window.



Sample edge point with Depth, Spacer, and Indent graphical visualizations used in the Graphic Display window

Notes on Graphical Analysis and Feature Extraction of Edge Points

If you don't see some graphical analysis points computed to the edge plane, consider the following:

- **Edge Line Points** - All the edge line points on the reference plane returned by the feature extractor are displayed. For analysis, PC-DMIS computes the edge line points using the distance (**Indent** value) from the reference plane center (center of the circular surface area defined by the **Spacer** value) to the edge line.
- **Reference Plane Points** - If the Spacer value is 0.0 then the reference plane points are not displayed. If the Space value is not 0.0 then PC-DMIS extracts the reference plane points from the pointcloud, applying the following rules and using the plane statistical data returned by the feature extractor:
  - Rule 1: PC-DMIS discards all points that are outside of an imaginary cylinder.

    This cylinder is identified by using the following values:

    Center = Indent Center Point

    Vector = Surface Vector

    Radius = Spacer
  - Rule 2: PC-DMIS discards all points with a distance from an imaginary plane greater than the maximum plane error value.

    This plane is identified by using the following values:

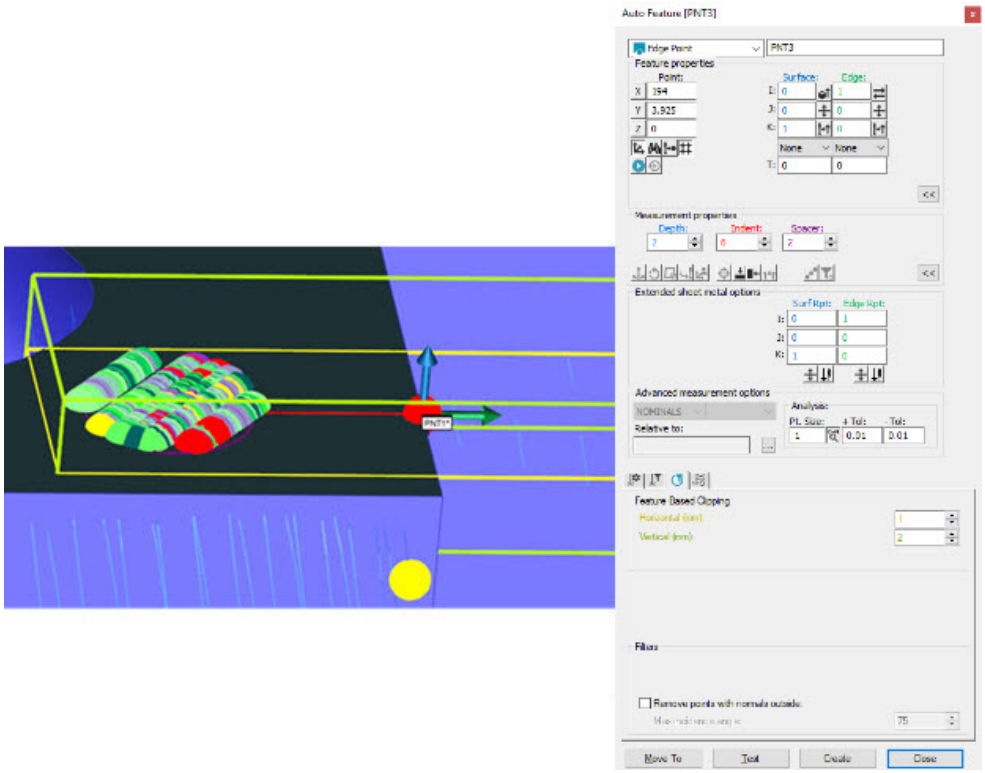
    Center = Measured Edge Point

    Vector = Measured Surface Vector
  - Rule 3: PC-DMIS uniformly reduces any remaining points that are higher than the allowed number (19900) to the allowed value.

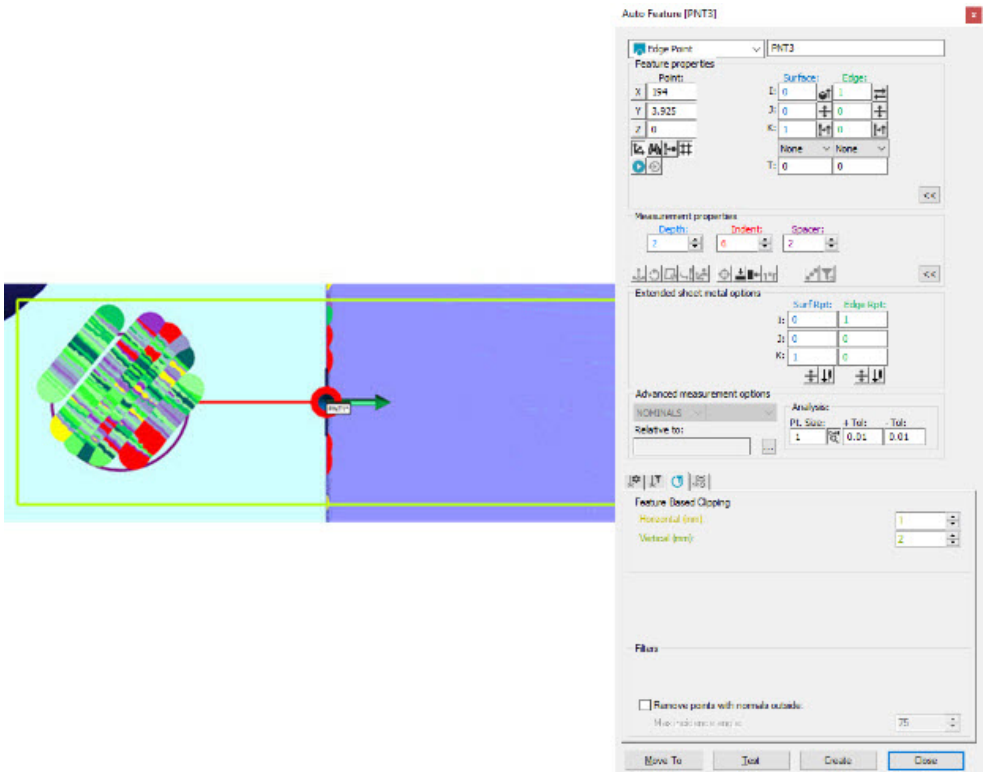
  For analysis, PC-DMIS computes each reference plane point using the distance from the reference plane and the measured surface plane.

The following two images show the Edge Point laser graphical analysis:

- Graphical Analysis Example - Side View



- Graphical Analysis Example - Top View



Edge Point Command Mode Text

The Auto Edge Point command in the Edit window in Command mode looks like this:

```
PNT2 =FEAT/LASER/EDGE POINT,CARTESIAN
   THEO/<1.895,1.91,1>,<0,1,0>,<0,0,1>
   ACTL/<1.895,1.91,1>,<0,1,0>,<0,0,1>
   TARG/<1.895,1.91,1>,<0,1,0>,<0,0,1>
   SHOW FEATURE PARAMETERS=YES
      SURFACE1=THEO_THICKNESS,1
      SURFACE2=THEO_THICKNESS,0
      MEASURE MODE=NOMINALS
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
      FEATURE LOCATOR=NO,NO,""
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
      FILTER=NONE
```

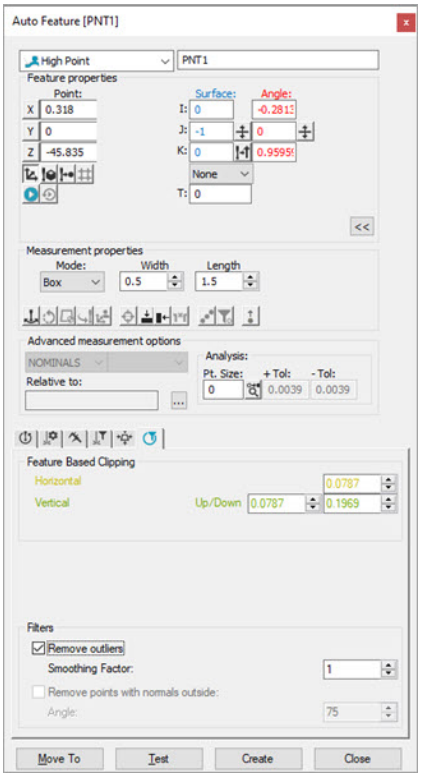
Laser High Point



Auto High Point button

Use the **Auto High Point** option to search a user-defined region on the part to locate the highest point in the current workplane. This samples the region for the highest point. It does not search existing points in your measurement routine.

When you execute a High Point feature, PC-DMIS searches for and then returns the highest point within the search region. The result of the search is a single point defined by its X, Y, and Z coordinates and approach vector.

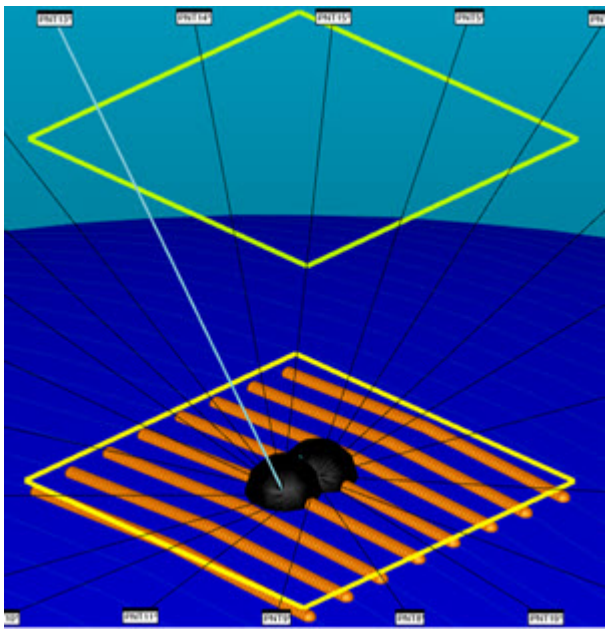


Auto Feature dialog box - High Point

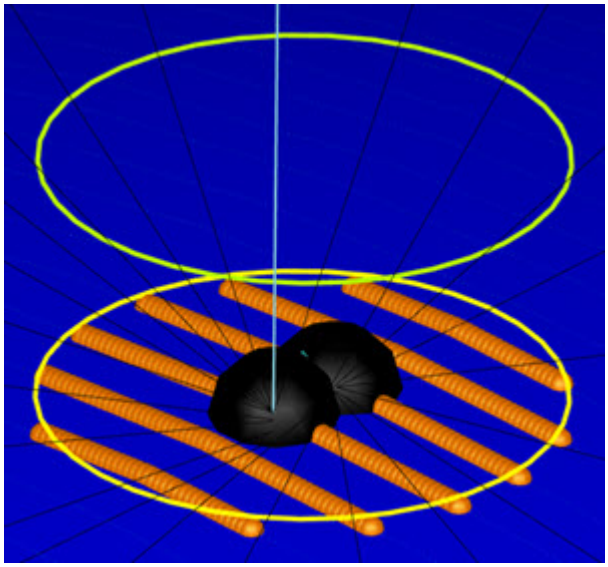
To measure a laser high point with a laser sensor, do the following:

1. From the menu, click **Insert | Feature | Auto** and select **High Point** to open the **Auto Features** dialog box for the high point feature.
2. Do one of the following:
    - Click the CAD in the Graphic Display window to give the point a location and vector. Type in any remaining information.
    - From the Graphic Display window, use the **Laser** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Type in any remaining information.
    - Type in the theoretical information for X, Y, Z, I, J, K, and so on.
3. From the Measurement properties area, select either **Box** or **Circular** from the **Mode** list. For details on this area, see the "Measurement Properties Area" section of the PC-DMIS Core documentation.

    If you select **Box** from the **Mode** list, PC-DMIS defines a rectangular search region to determine the high point feature. You can use the **Width** and **Length** boxes to define the region.



If you select **Circular** from the **Mode** list, PC-DMIS defines a circular search region to determine the high point feature.



4. Click the **View Segregated Points** button ( ) from the **Laser Scan Properties** tab of the Probe Toolbox to view the segregated points inside the search region as shown in the above two images.

5. From the **Reference** list, select the Pointcloud (COP) or Mesh feature to extract the high point from, or select **Disable** for direct measurement.

6. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

7. Click the **Test** button to test the feature before you create it.

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.

8. Click **Create** and then **Close**.

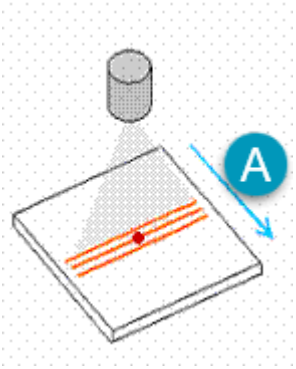
High Point Command Mode Text

The Auto High Point command in the Edit window in Command mode looks like this:

```
PNT13=FEAT/LASER/HIGH POINT/DEFAULT,CARTESIAN
   THEO/<0,0,30>,<0,0,1>
   ACTL/<-0.333,-0.031,29.984>,<0,0,1>
   TARG/<0,0,30>,<0,0,1>
   ANGLE VEC=<0.9999988,0,0.0015469>
   OX,WIDTH=5,LENGTH=5
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THICKNESS_NONE,0
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
   SHOW_LASER_PARAMETERS=YES.
```

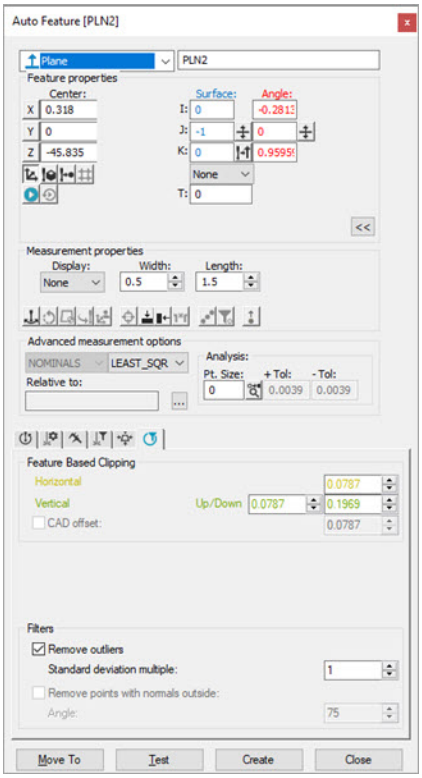
Auto Surface or High Point Path

PC-DMIS determines the direction of the path for the Surface Point or High Point on the stripe.



Path direction of scan for surface or high point
(A) - Scan motion

Laser Plane



Auto Feature dialog box - Plane

To create an auto plane using a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Plane** to open the **Auto Features** dialog box for the plane feature.

2. Do one of the following:

    - Click on the CAD to give the plane a location and vector. You can use the Ctrl key to select multiple planes if they are coplanar to each other. Type in any remaining information.

    - From the Graphic Display window, use the **Laser** tab to move the machine to the center of the plane location, and then click the **Read Point from Position** button ().

    - From the **Feature properties** area, type the **X**, **Y**, **Z**, **I**, **J**, **K**, Material Thickness (**T**), and any other parameters as necessary. From the **Measurement properties** area, type the **Display**, **Width**, **Length**, and any other parameters, as necessary.

3. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

4. Click the **Test** button to test the feature before you create it.

> **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.

5. Click **Create** and then **Close**.

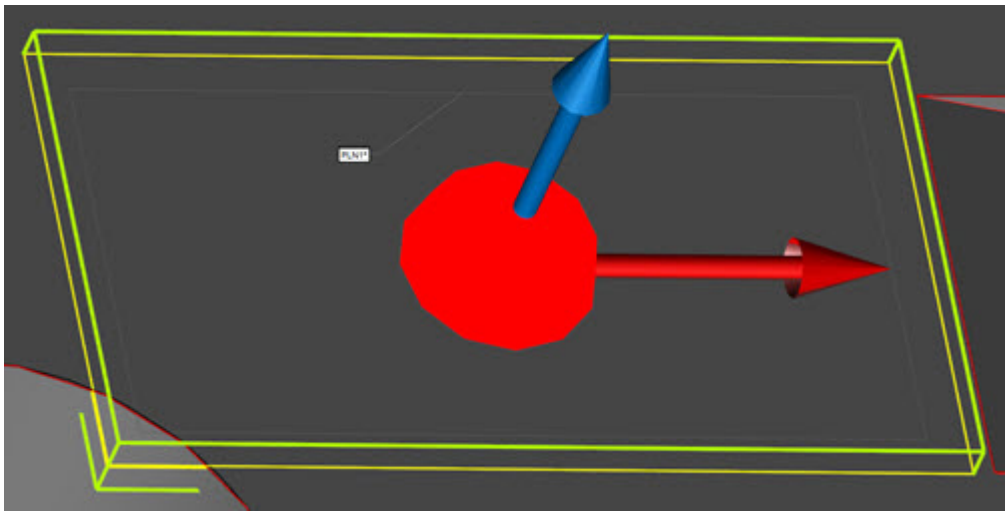
Plane Specific Parameters

Width: The value in this box determines the width of the measurement area of the plane.

Length: The value in this box determines the length of the measurement area of the plane.

Display: This list lets you choose how to present the plane inside the Graphic Display window. You can choose **NONE** or **TRIANGLE** or **OUTLINE**:

- If you choose **NONE**, the plane is not displayed.
- If you choose **TRIANGLE**, PC-DMIS displays the plane with a triangle symbol at the very center of the plane.
- If you choose **OUTLINE**, PC-DMIS displays an outline of the edges of the plane.



Sample plane in the Graphic Display window with:
***Outline** display (Gray dotted line)*
***Overscan** display (Yellow rectangle)*
***Vertical Clipping** (Green rectangular box)*

Plane Command Mode Text

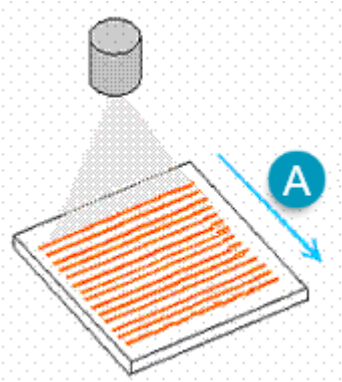
The Auto Plane command in the Edit window in Command mode looks like this:

```
PNT1 =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN,TRIANGLE
   THEO/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
   ACTL/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
   TARG/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
   DEPTH=4
   INDENT=7
   SPACER=1
   SHOW FEATURE PARAMETERS=YES
      SURFACE1=THEO_THICKNESS,0
      SURFACE2=THEO_THICKNESS,0
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=COP2
      HORIZONTAL CLIPPING=9,VERTICAL CLIPPING=9
```

Auto Plane Paths

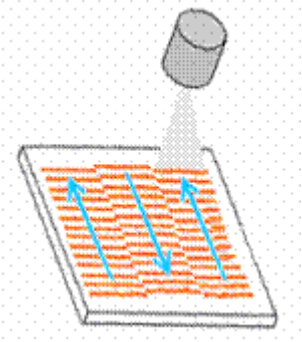
PC-DMIS provides two different paths for a plane. It automatically chooses the appropriate path based on the width and the size of the usable portion of the laser stripe. For auto planes, PC-DMIS always scans perpendicular to the direction of the stripe.

Path 1: Smaller Width



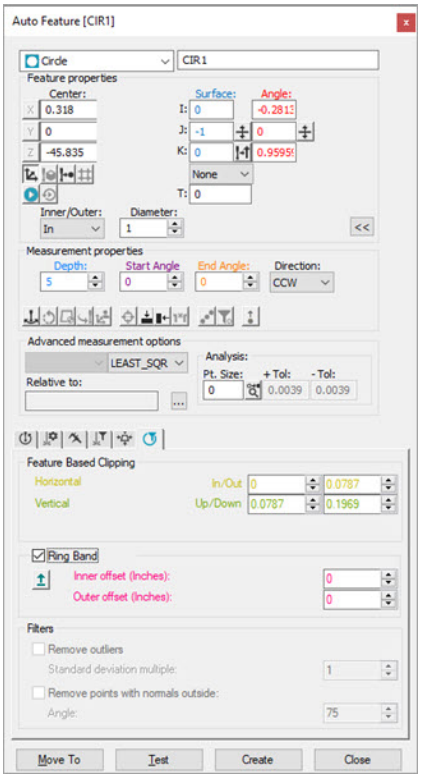
Planes with a width smaller than the usable portion of the stripe
(A) - Scan motion

Path 2: Larger Width



Planes with a width larger than the usable portion of the stripe

Laser Circle



Auto Feature dialog box - Circle

To create a laser auto circle:

1. From the menu, click **Insert | Feature | Auto** and select **Circle** to open the **Auto Features** dialog box for the circle feature.
2. Do one of the following:
    - Perform clicks on the CAD to give the circle a location and vector. Type in any remaining information.
    - From the Graphic Display window, use the **Laser** tab to move the machine to the circle location. Next, from the **Feature Properties** area, click **Read Point from Machine** . Type in any remaining information such as the diameter, depth, and other parameters.
    - Type in all of the theoretical information for X, Y, Z, I, J, K, diameter, depth, and other parameters.
3. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.
4. Click the **Test** button to test the feature before you create it.

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.

5. Click the **Create** button and then click **Close**.

Currently, you can only measure inner circles (holes) with laser sensors.

Circle-Specific Parameters

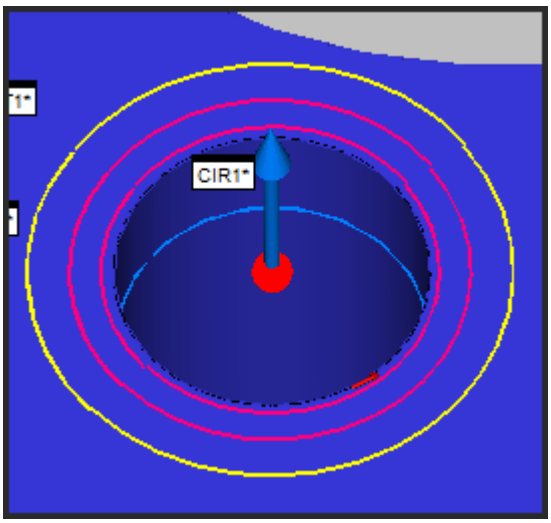
Diameter - This box specifies the circle's diameter. When you select a circle with the mouse in the Graphic Display window, PC-DMIS automatically places the circle's diameter from the CAD model in this box.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. Any other depth value causes PC-DMIS to calculate the feature using the points from the surface plane to the given depth value. Due to hardware limitations, for this feature type, if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data from 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you probably need to specify a depth to get accurate results.

Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample circle in the Graphic Display window that shows:
Depth (blue circle)
Ring Band (pink circles)
Overscan (yellow circle)

Start Angle and **End Angle** - These boxes let you change the default start and end angles on the feature. This is a user-supplied angle in decimal degrees. If you rotate the view of the feature so that you look down at its center, PC-DMIS shows the portion of the pointcloud to pass to the segregation routine.

Direction - Select the direction in which the hits are taken: **CW** (clockwise) or **CCW** (counterclockwise).

Circle Command Mode Text

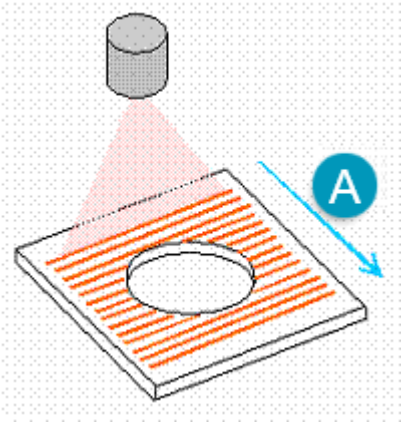
The Auto Circle command in the Edit window in Command mode looks like this:

```
CIR2 =FEAT/LASER/CIRCLE,CARTESIAN
   THEO/<1.895,1.91,1>,<0,0,1>,1.895
   ACTL/<1.895,1.91,1>,<0,0,1>,1.895
   TARG/<1.895,1.91,1>,<0,0,1>
   ANGLE VEC=<0,0,1>
   DEPTH=3
   SHOW FEATURE PARAMETERS=YES
      MEASURE MODE=NOMINALS
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
      FEATURE LOCATOR=NO,NO,""
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
      FILTER=NONE
```

Auto Circle Paths

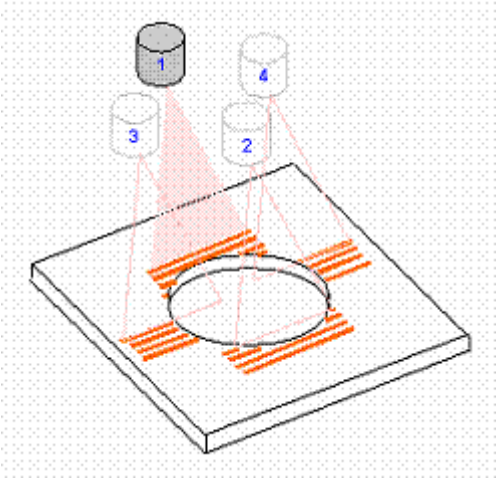
PC-DMIS provides two different paths for a circle. It automatically chooses the appropriate path based on the diameter and the size of the usable portion of the laser stripe. For auto circles, PC-DMIS always scans perpendicular to the direction of the stripe.

Path 1: Smaller Diameter



Circles with a diameter smaller than the usable portion of the stripe
(A) - Scan motion

Path 2: Larger Diameter



Circles with a diameter larger than the usable portion of the stripe

The method for measuring circles with a larger diameter has been improved to measure the four passes at 1:30, 4:30, 7:30, and 10:30 rather than 12:00, 3:00, 6:00, and 9:00 as depicted in the image.

Auto Circle Fractions

You can measure a fraction of a laser auto circle. You may need to measure a fraction of a circle in these cases:

- The start and end angles are already defined on the CAD model.

- A complete circle is defined on the CAD model, but the actual pointcloud data is inconsistent.
- The start and end angles may be defined the wrong way on the CAD model.

These are the features and parameters that you can use to measure a fraction:

- Single-click on the CAD model to enable PC-DMIS to learn the start and end angles.
- Define the start and end angles. You can type the angles in the **Start Angle** and **End Angle** boxes in the **Measurement Properties** area in the **Auto Feature** dialog box. For more information about these boxes, see "Circle-Specific Parameters".
- Edit all of the laser auto circle's features, including the direction (clockwise or counterclockwise). For more information, see "Circle-Specific Parameters".

A pointcloud that represents an entire part might contain millions of points. PC-DMIS passes all the parameters to the segregation routine. A segregation routine is a piece of code that creates a subset of the given pointcloud. The horizontal and vertical clipping parameters limit the subset. PC-DMIS then extracts the geometric element from this subset. This is easier and faster than from an entire pointcloud.

Command Mode Example

Here is an example of the Edit window in Command Mode:

```
CIR_PARA_1 =FEAT/LASER/CIRCLE/DEFAULT,CARTESIAN,IN,LEAST_SQR
   THEO/<-26,51.094,-11.344>,<-1,0,0>,4
   ACTL/
<-26.013,50.113,-11.55>,<-0.9578033,0.0757769,-0.2772556>,2.
177
   TARG/<-26,51.094,-11.344>,<-1,0,0>
   DEPTH=0,START ANG=200,END ANG=340
   ANGLE VEC=<0,0,1>
   DIRECTION=CCW
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THICKNESS_NONE,0
      RMEAS=NONE,NONE,NONE
      GRAPHICAL ANALYSIS=YES,0.5,0.01,0.01
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=COP1
      HORIZONTAL CLIPPING=2,VERTICAL CLIPPING=1
      RINGBAND=ON,INNER OFFSET=0.5,OUTER OFFSET=1.5
      OUTLIER REMOVAL=OFF
      REMOVE POINTS WITH NORMALS OUTSIDE=OFF
```

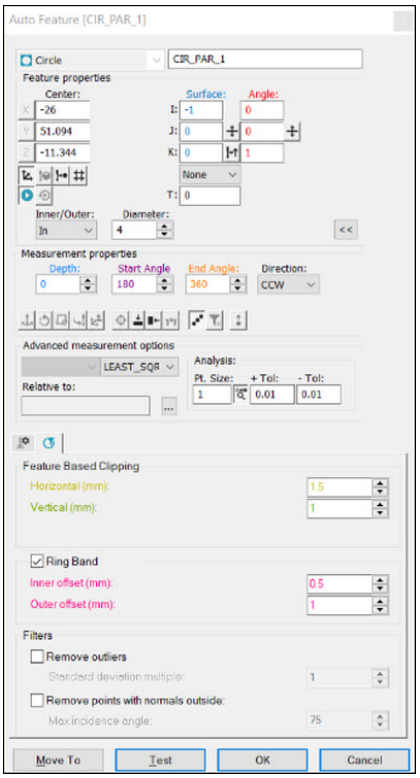
Summary Mode Example

Here is an example of the Edit window in Summary Mode:

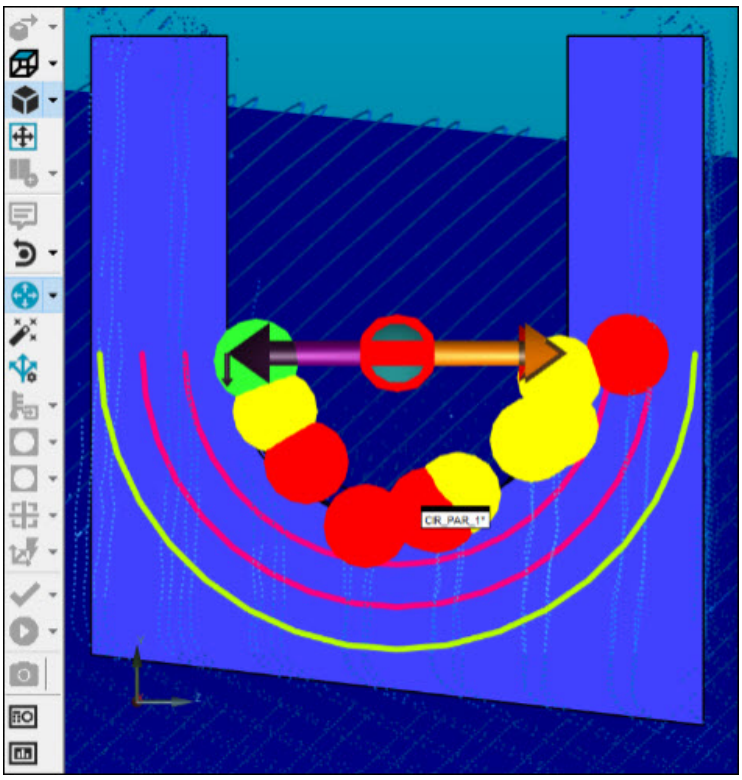
```
FEAT/LASER/CIRCLE/DEFAULT,CARTESIAN,IN,LEAST_SQR
   THEO/<-26,51.094,-11.344>,<-1,0,0>,4
   ACTL/
<-26.013,50.113,-11.55>,<-0.9578033,0.0757769,-0.2772556>,2.
177
   TARG/<-26,51.094,-11.344>,<-1,0,0>
   DEPTH=0,START ANG=200,END ANG=340
   ANGLE VEC=<0,0,1>
   DIRECTION=CCW
```

CAD Selection Example

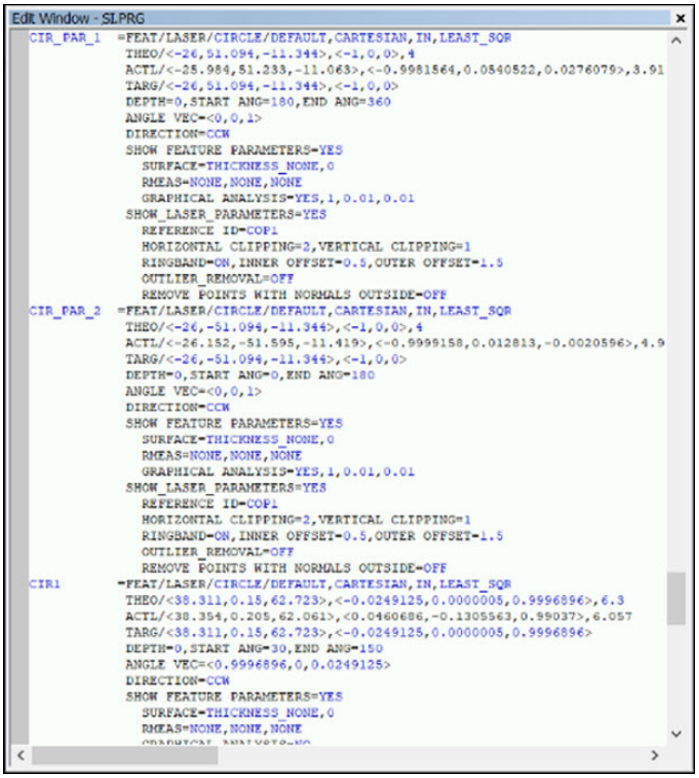
Here is an example of the **Auto Feature** dialog box, CAD model, and Edit window:



Auto Feature dialog box



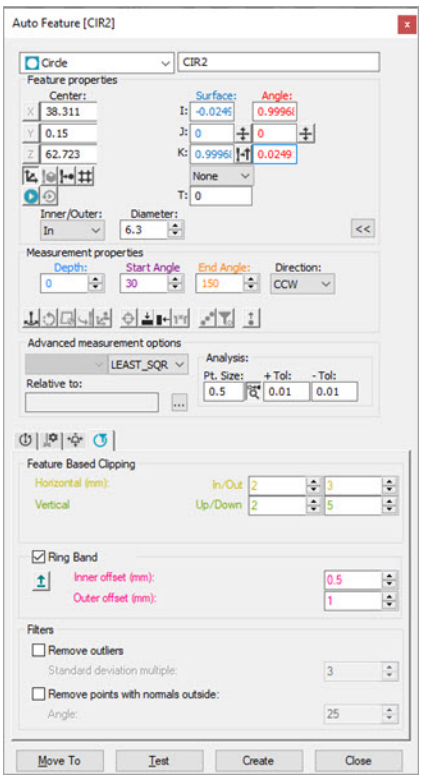
CAD model



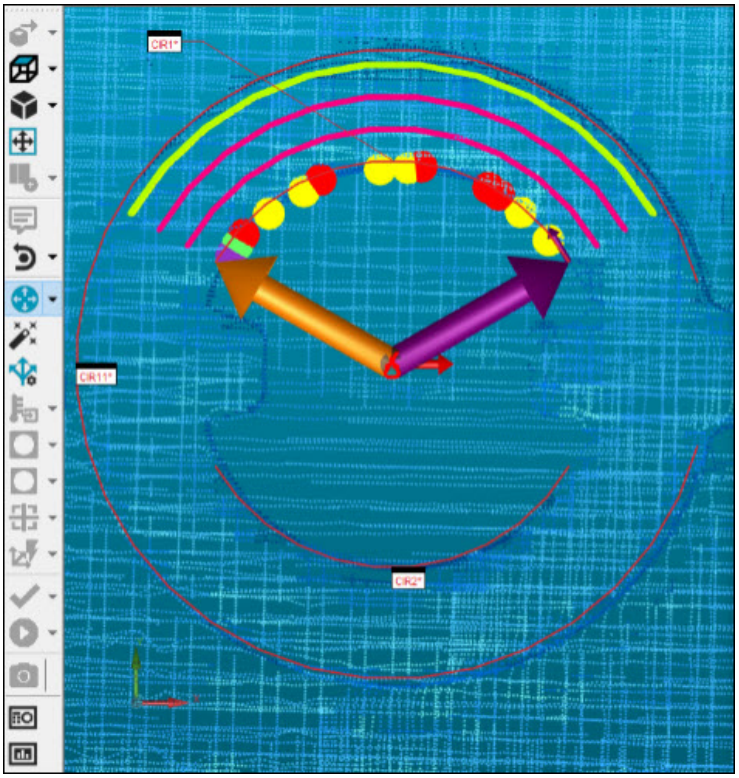
Edit window

Part Variance Example

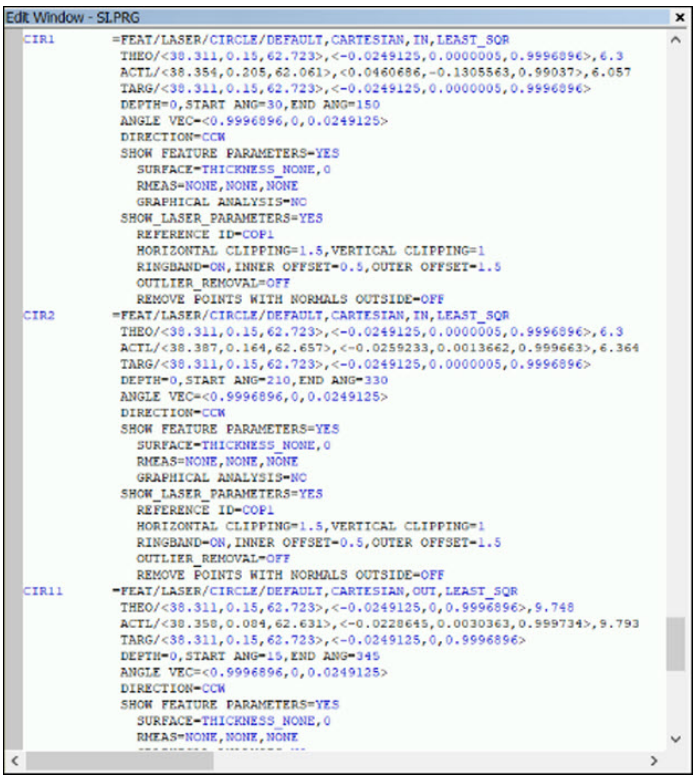
Here is an example of the **Auto Feature** dialog box, CAD model, and Edit window for removing the part variance:



Auto Feature dialog box

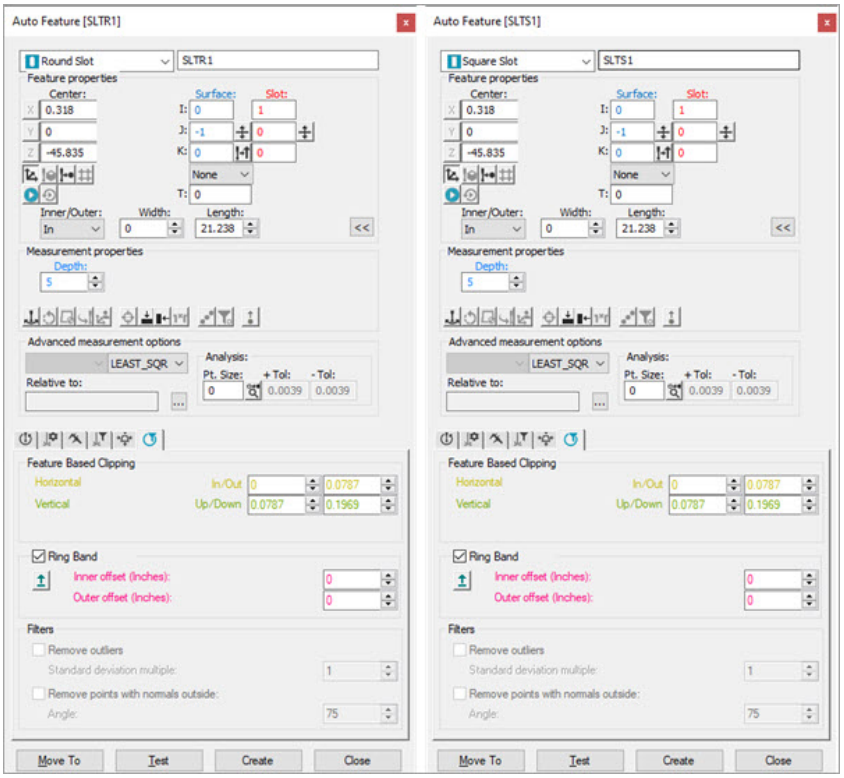


CAD model



Edit window

Laser Slot



Auto Feature dialog box - Round Slot (left) and Square Slot (right)

To measure a slot with a laser sensor:

1. Open the **Auto Features** dialog box (**Insert | Feature | Auto**), and select **Round Slot** or **Square Slot**.

2. Do one of the following:

   A. Click on the CAD to collect the X, Y, Z, I, J, K information. If PC-DMIS doesn't automatically recognize the feature, do this:

      For Round Slots:

      a. Click one of the rounded edges of the slot in the Graphic Display window. PC-DMIS prompts you to click two more times on the same rounded edge.

      b. Click twice on this edge. PC-DMIS then prompts you to click on the other rounded edge.

      c. Click on the other rounded edge. PC-DMIS prompts you to click two more times on that same rounded edge.

      d. Click twice on the second rounded edge. PC-DMIS establishes the orientation of the round slot.

      For Square Slots:

      a. Click on one of the long edges of the slot in the Graphic Display window. PC-DMIS prompts you to click another location on the same edge to determine the direction.

      b. Click on a second edge, 90 degrees from the first.

      c. Click on the third edge, 90 degrees from the second. This sets the width.

      d. Click on the fourth edge and final edge. This sets the length.

   B. From the Graphic Display window, use the **Laser** tab, and move the machine to the slot location.

   C. From the **Feature Properties** area, click the **Read Point from Position** button ().

3. Type in the theoretical X, Y, Z, I, J, K, width, length, depth, height, and other parameters.

4. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

5. Click the **Test** button to test the feature before you create it.

> **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.

6. Click **Create** and then **Close**.

Slot Specific Parameters

Inner/Outer - This list lets you choose whether or not the slot is an Inner slot (a hole) or an Outer slot (a stud).

Width - The value in this box determines the slot's width.

Length - The value in this box determines the slot's length.

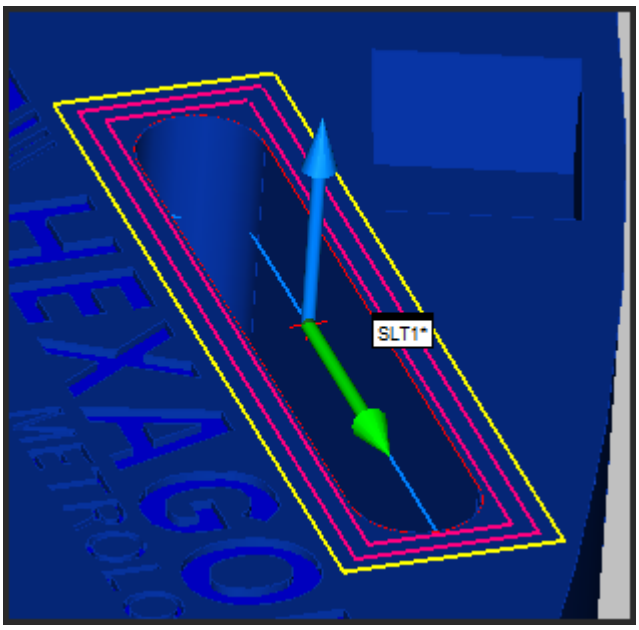
Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you should specify a depth to get accurate results.

> Even if you specify a depth greater than zero, PC-DMIS always projects the measured results into the plane where the feature resides.

> The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

Slot (Vector) - These boxes define the slot's orientation.



Sample round slot in the Graphic Display window showing:
Depth (blue slot line)
Ring Band (pink rectangles)
Overscan (yellow rectangle)

Slot Command Mode Text

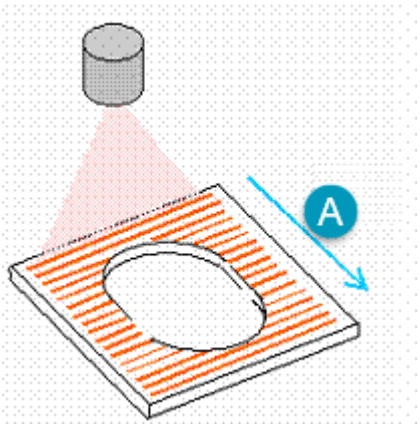
The Auto Slot command in the Edit window in Command mode looks like this:

```
SLT1 =FEAT/LASER/SQUARE SLOT,CARTESIAN
   THEO/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
   ACTL/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
   TARG/<1.895,1.91,1>,<0,0,1>
   DEPTH=3
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THEO_THICKNESS,1
      MEASURE MODE=NOMINALS
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
      FEATURE LOCATOR=NO,NO,""
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
      FILTER=NONE
```

Auto Round Slot Paths

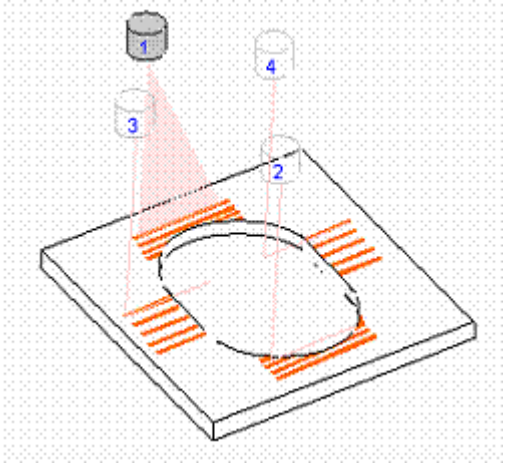
Depending on the width of the round slot, PC-DMIS takes one of these paths when it performs the measurement:

Path 1: Narrow Width



Round slots with a width less than the usable portion of the stripe
(A) Scan motion

Path 2: Larger Width

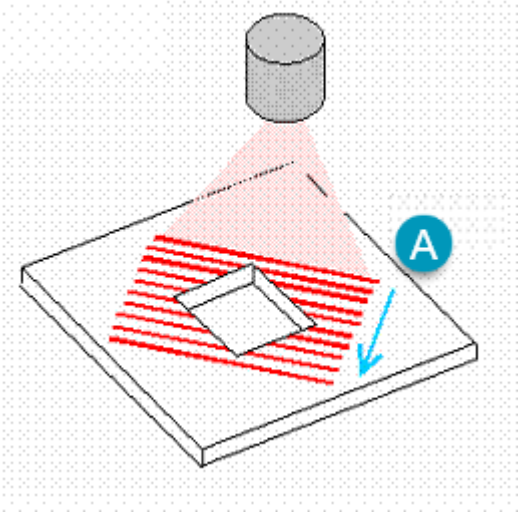


Round slots with a width larger than the usable portion of the stripe

Auto Square Slot Paths

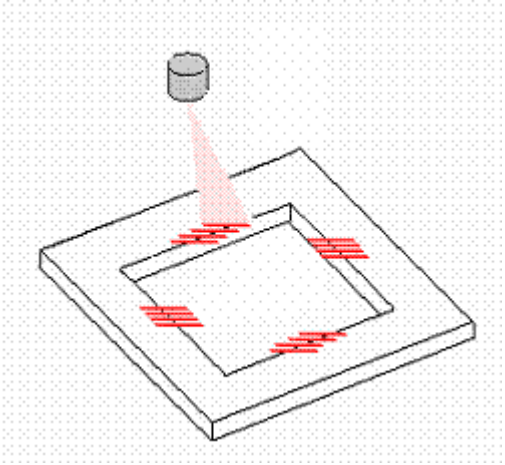
PC-DMIS must measure Auto Square slots at a 45 degree angle to the slot (see the illustrations below). Depending on the size of the slot, PC-DMIS takes one of these two paths.

Path 1: Small Slot - Measured with a single pass of the laser sensor



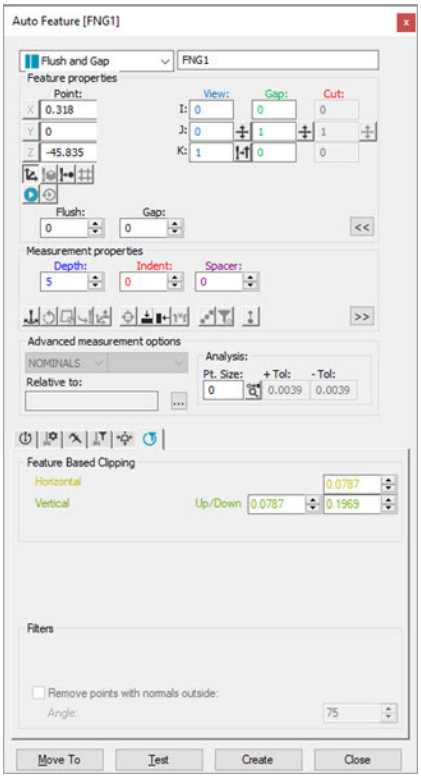
Small square slots require a single pass of the laser sensor's stripe
(A) - Diagonal scan motion

Path 2: Large Slot - Measured with multiple passes of the laser sensor



Large square slots require multiple passes of the laser sensor's stripe

Laser Flush and Gap



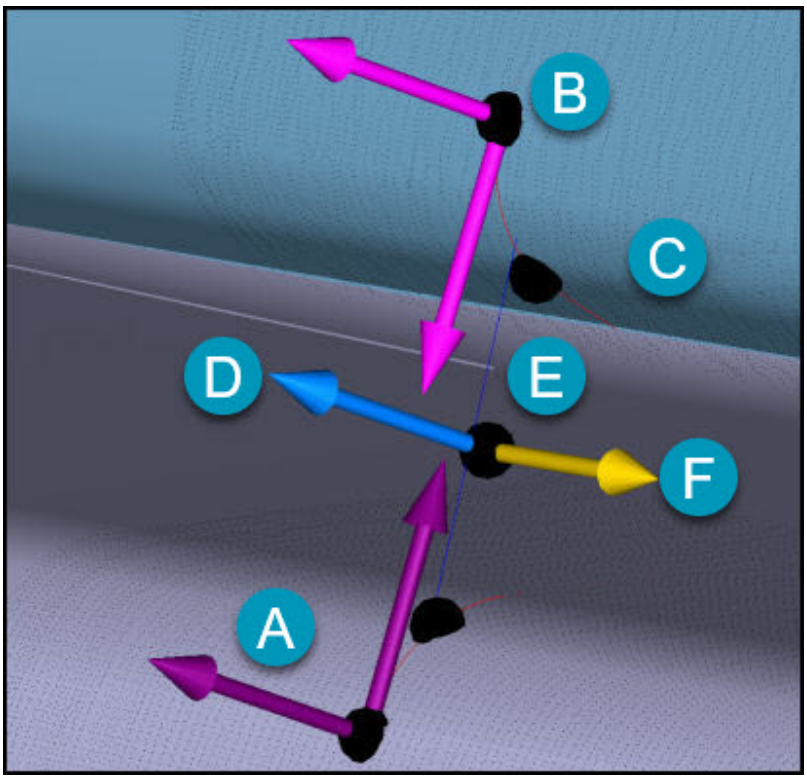
Auto Feature dialog box - Flush and Gap

Flush and Gap measures the height difference between two mating sheet metal parts (the Flush) and the distance between two mating parts (the Gap).

To measure a Flush and Gap using a laser sensor, from the menu, click **Insert | Feature | Auto** and select **Flush and Gap** to open the **Auto Features** dialog box for the flush and gap feature. The dialog box automatically expands the **Extended sheet metal options** area. This area provides **XYZ** position boxes and **IJK** vector boxes for the Master and Gauge points. Follow one of the procedures below.

With CAD Data

1. Load a CAD model.
2. Click on the Master side.
3. Click on the Gauge side.



A *- Master*
B *- Gauge*
C *- CAD learned curves*
D *- View vector*
E *- Depth line*
F *- Cut vector*

These points must be on the reference "flat" surfaces, where PC-DMIS sets the planes used to calculate flush, and not on the curves.

PC-DMIS performs these calculations in the background:

- The software learns the theoretical Flush.
- The software learns the curves from the CAD model.
- The software learns the point coordinate and vectors for both the Master and Gauge sides of the gap.
- The software applies the defined Depth value, and, after piercing the curves, it calculates the theoretical gap at the specified depth.
- The software also calculates the cut vector (along the rail) and the Gap direction (crossing the rail).

4. Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.
5. Set other parameters as needed. See "Flush and Gap Specific Parameters".

6. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

7. Click the **Test** button to test the feature before you create it.

> **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.

8. Click the **Create** button and then **Close**.

Flush and Gap CAD Selection Capability

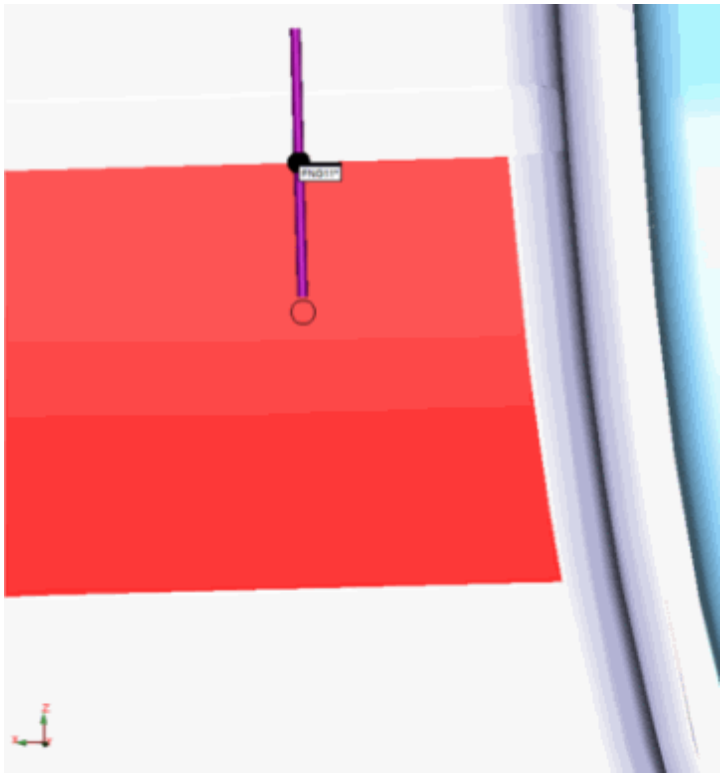
The ability to re-click the first CAD point on a selected surface is often a requirement when you define or re-define a measurement routine.

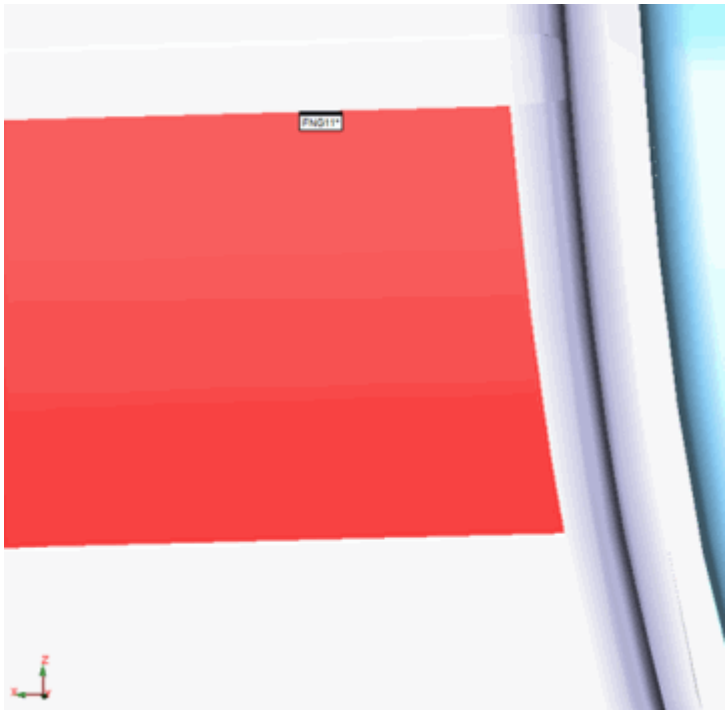
Other than the Master Side Point and the Edge Vector, PC-DMIS displays the first point you click in the Graphic Display window, as a black circle centered in the picked point. The software also highlights the selected surface.

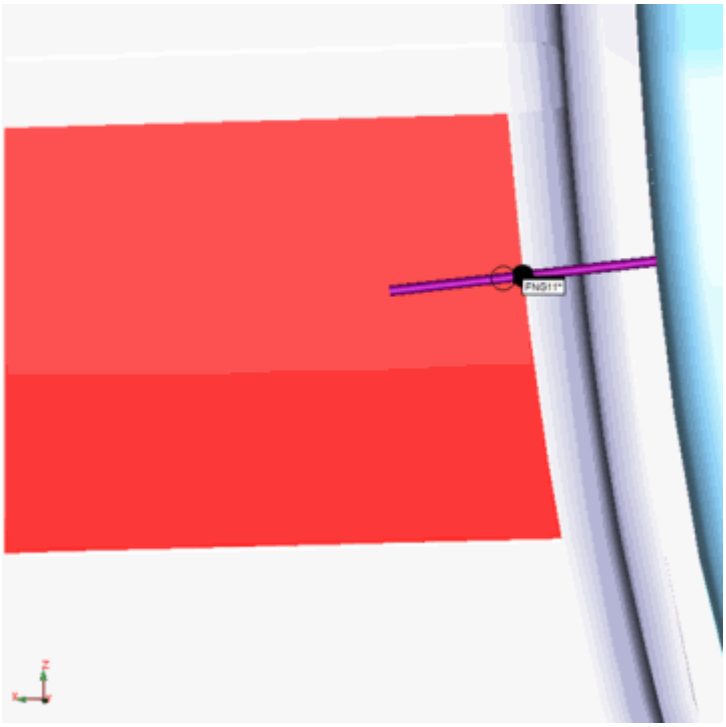
Sometimes, the Master Side Point found is in a wrong surface boundary location, and the software requires you to click the point again. The following describes two ways to do this:

1. If the desired Master Side Point is on the edge of the highlighted surface, it's sufficient to just re-click a point on the surface that is very close to the edge.

2. If the desired Master Side Point doesn't lie on the highlighted surface, you can click the drawn circle area to reset the interface. PC-DMIS is then ready for you to re-take the first point. To help re-define the new surface selection, PC-DMIS leaves the previous surface highlighted. See the example pictures below:







Example of Flush and Gap Auto feature CAD selection capability

Without CAD Data

1. Use the **Laser** tab of the Graphic Display window to move the machine to the gap location.
2. Click the **Read Point from Position** button.
3. Type all of the theoretical XYZ and IJK values. These include the Flush and Gap **Point**, the **View Vector**, **Gap Dir** (gap direction), **Master Pnt** (master point), **Gauge Pnt** (gauge point), **Master Vec** (master vector), and **Gauge Vec** (gauge vector).

   Note that when you change some parameters, and you don't have any CAD data, PC-DMIS adjusts some parameter values automatically. For information, see "Automatically Adjusted Flush and Gap Values".
4. Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.
5. Set other parameters as needed. For information, see "Flush and Gap Specific Parameters".
6. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.
7. Click the **Test** button to test the feature before you create it.

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.

8. Click the **Create** button and then **Close**.

Flush and Gap Specific Parameters

For a visual example of these parameters, consult the diagrams below.

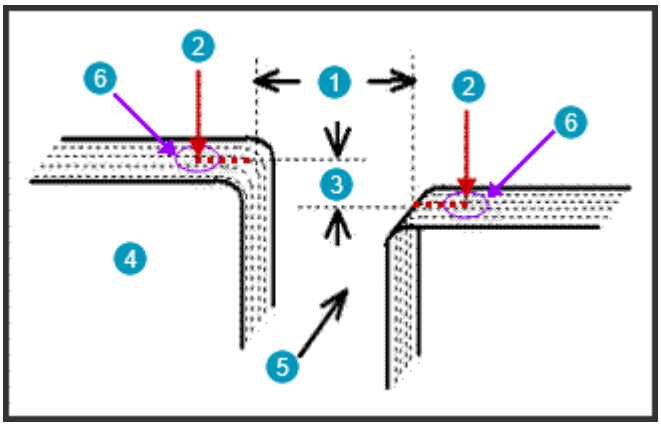
Flush - This box determines the height difference between two mating sheet metal parts. Whether or not the flush value is positive or negative depends on whether it's higher or lower than the "Master" side.

Gap - This box determines the distance (in the same plane) between two mating sheet metal parts.

Indent - The indent specifies the distance from the gap's edge at which PC-DMIS measures the flush.

Spacer - This is a circle at the indent point used to calculate the surface normals used in the calculation.

Gap Dir (Vector) - These boxes in the **Feature Properties** area define the direction of the gap.



Flush and gap diagram

Key:

1 - Gap

2 - Indent

3 - Flush (negative flush is shown at left)

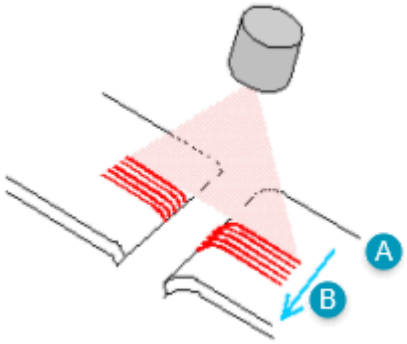
4 - Master side

5 - Cut Vector

6 - Spacer

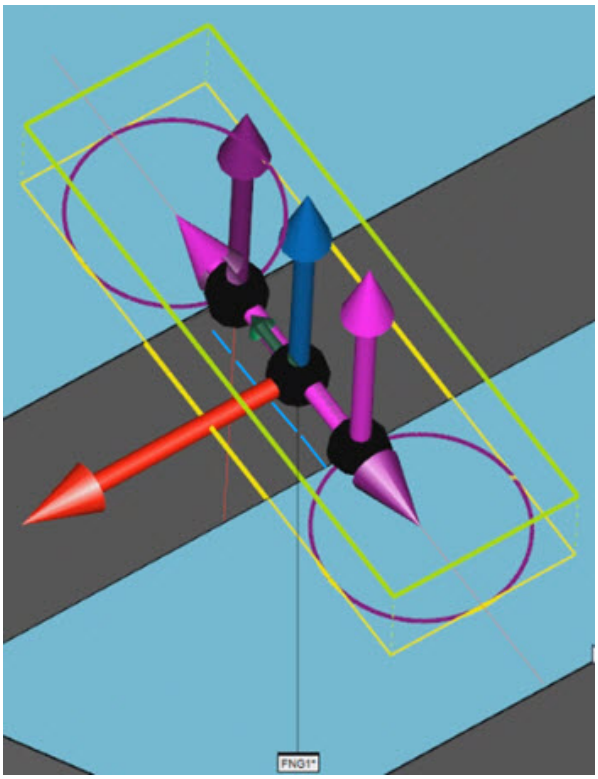
The "Master" side is always to the left of the scan/gap direction.

The direction of the scan is controlled by the specified Cut Vector and not the direction of the laser stripe.



Direction of scan
(A) - Masterside (B) - Scan motion

The "Master" side is always to the left of the Cut Vector.



Sample Flush and Gap Auto feature in the Graphic Display window. This shows the indent (red lines), spacer (purple circles), depth (blue line), horizontal clipping region (yellow lines), vertical clipping region (green), the view vector (blue arrow), and the cut vector (red arrow).

Flush and Gap Command Mode Text

The Auto Flush and Gap command in the Edit window in Command mode looks like this:

```
FNG2 =FEAT/LASER/FLUSH AND GAP/DEFAULT,CARTESIAN
   THEO/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
   ACTL/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
   TARG/<124.012,13.241,0>,<0,0,1>
   MASTER SIDE POINT
   THEO/<128,13.241,0>,<0,0,1>
   ACTL/<0,0,0>,<0,0,0>
   GAUGE SIDE POINT
   THEO/<120,13.241,0>,<0,0,1>
   ACTL/<0,0,0>,<0,0,0>
   CUT PLANE VECTOR<0,1,0>,<0,1,0>
   Depth=1
   INDENT=3
   SPACER=1.5
   SHOW FEATURE PARAMETERS=NO
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      ZOOM=2A,GAIN=NORMAL,OVERLAP=1
      OVERSCAN=5
      REDUCTION FILTER=OFF
      FILTER LINES=Disabled
      CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
      SOUND=ON
      HORIZONTAL CLIPPING=2,VERTICAL CLIPPING=5
```

Flush and Gap Graphical Analysis

The Flush and Gap analysis comprises these three regions. Consult the diagram at the bottom of this topic:

1. **Gap region** - In the Gap region, the points that are analyzed are in a box centered on the gap point and oriented along the Gap vector. The box's height is 60% of Gap length value. The width is 130% of Gap length value.

2. **Master Flush region** - In the Master Flush region, the points are analyzed in an area that begins at the Master Side point in a direction opposite from the Master Edge Vector. It has a length that is 60% of the Gap length value.

3. **Gauge Flush region** - In the Gauge Flush region, the points are analyzed in an area that begins at the Gauge Side point in the direction opposite from the Gauge Edge Vector. It has a length that is 60% of the Gap length value.

The Flush and Gap analysis is performed using these measured items.

- Gap Point and Vector

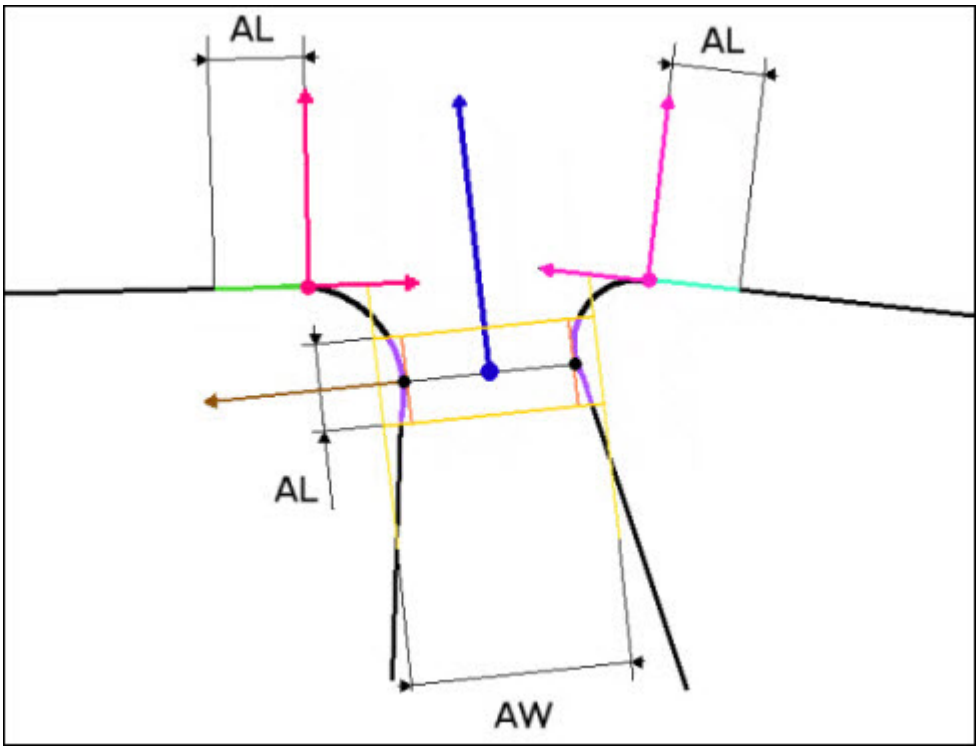
- Master Side Point
- Master Side Surface and Edge Vectors
- Gauge Side Point
- Gauge Side Surface and Edge Vectors

PC-DMIS computes the Flush and Gap measured point's distance from these four measured reference planes:

- The first two planes are the Gap analysis reference planes defined from the two measured Minimum Distance Points (where the Gap distance is calculated) and the measured Gap Vector.
- The third plane is the measured Master Side analysis reference plane. It is defined by the measured Master Side Point and the measured Master Side Surface Vector.
- The fourth plane is the measured Gauge Side analysis reference plane. It is defined by the measured Gauge Side Point and the measured Gauge Side Surface Vector.

To reduce the analysis time, PC-DMIS only uses the points closest to the cut plane (less than 0.5 mm or 0.19685 inches).

Graphical Analysis Diagram:



Key:

AL - Analysis Length. It is 60% of the Gap length value.

AW - Analysis Width. It is 130% of the Gap length value.

● - Minimum Distance Points

 - Gap Vector

 - Gap Point and View Vector

 - Gauge Side Point and Vectors

 - Master Side Point and Vectors

 - Master Side Flush analysis region. Reference Plane.

 - Gauge Side Flush analysis region. Reference Plane.

 - Gap analysis region

 - Gap analysis reference plane

Automatically Adjusted Flush and Gap Values

Note that when you change some Flush and Gap parameters, and you don't have any CAD data, PC-DMIS will adjust some parameters' values automatically. This topic details what gets changed and how the software computes those automatic values.



Key: Use these abbreviations when viewing the equations below:

- **CPV** = Cut Plane Vector
- **VV** = View Vector
- **x** = Cross Product
- **GV** = Gap Vector
- **GD** = Gap Distance
- **GP** = Gap Point
- **GPV** = Gap Point Vector

When Typing in a Gap Point Value or Modifying it by Read Position...

- The current probe vector is used as the View Vector.
- The current stripe vector is used as the Gap Vector.
- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- Master Side Point and Gauge Side Point are ESTIMATED at $\frac{GD}{2}$ from the new Gap Point along the Gap Vector.

    - If the Flush distance is positive, the Master Side Point is translated along the View Vector of the flush value.
    - If the Flush distance is negative, the Gauge Side Point is translated along the View Vector of the flush value.

- The Master Side Surface Vector and the Gauge Side Surface Vector are set with the View Vector.

When Typing in a View Vector Value...

- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The Gap Vector is computed to be orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- The Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Gap Vector Value...

- The new cut plane is located in the Gap point and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The View Vector is computed to be orthogonal to the new Gap Vector: $VV = GV.x(CPV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Master Side Point Value or Modifying it by Read Position...

- The new cut plane is computed orthogonal to the View Vector and the Master Side Point minus the Gap Point: $CPV = VV.x(MSP - GP)$
- The Gap Vector is computed orthogonal to the new View Vector. $GV = CPV.x(VV)$
- The Master Side Surface Vector, the Gauge Side Surface Vector and Gauge Side Point are translated onto the new cut plane.

When Typing in a Gauge Side Point Value or Modifying it by Read Position...

- The new cut plane is computed centered on the new Master Side Point and orthogonal to the View Vector and the Master Side Point minus the Gauge Side Point: $CPV = VV.x(MSP - GSP)$

- The Gap Vector is computed orthogonal to the new View Vector:
  $GV = CPV.x(VV)$

- The Master Side Surface Vector, the Gauge Side Surface Vector, and Gap Point are translated onto the new cut plane.

When Typing in a Flush Distance Value...

- The Master Side Point and/or the Gauge Side Point are translated according to the new flush value along the Master or Gauge Side Surface Vector.

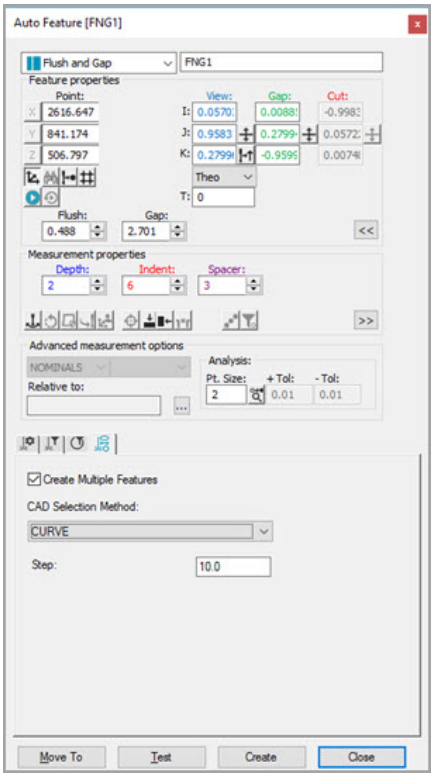
When Typing in the Distance Value...

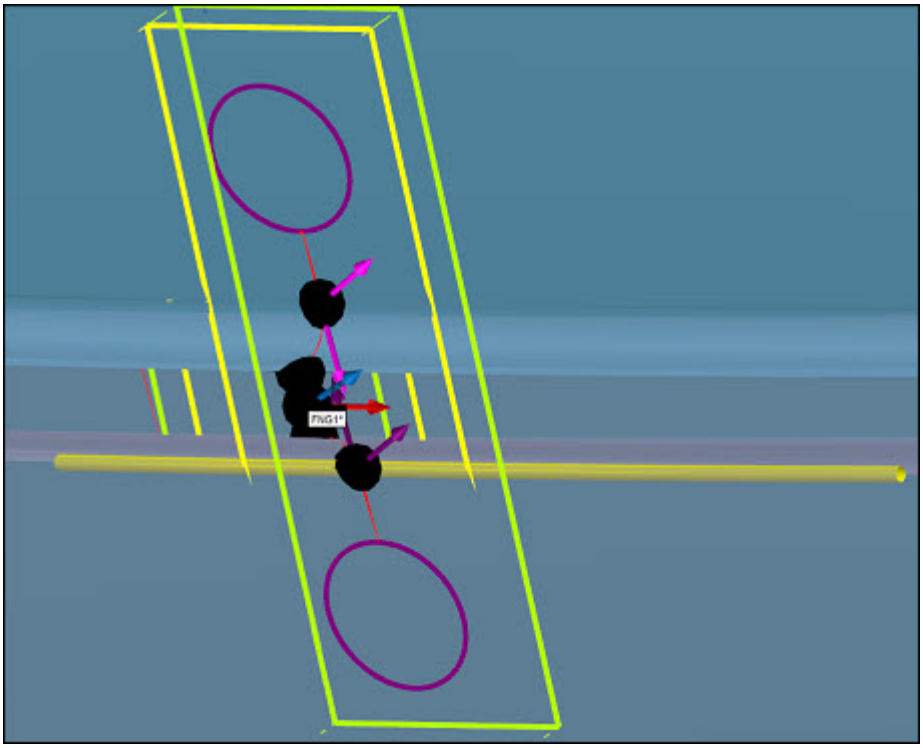
- The Master Side Point and/or the Gauge Side Point are translated according to the new gap value along the Gap Vector.

Flush and Gap Features Around a Defined Contour

You can extract a series of Flush and Gap features around a defined contour. See the following examples:

First Curve Selection

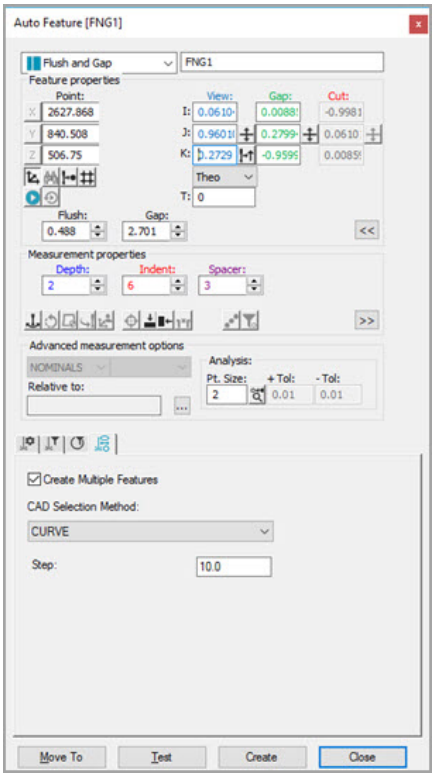


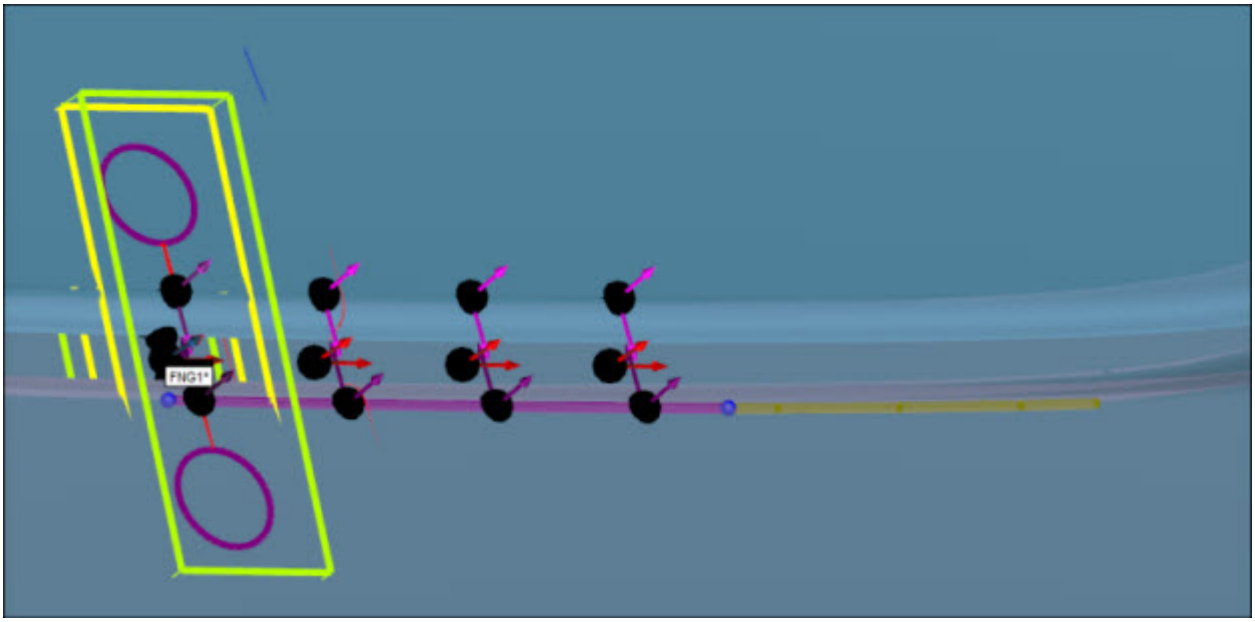


Flush and Gap Auto feature - First curve selection

Additional Curve Selection with Ctrl Key

To select additional curves, press and hold the Ctrl key:

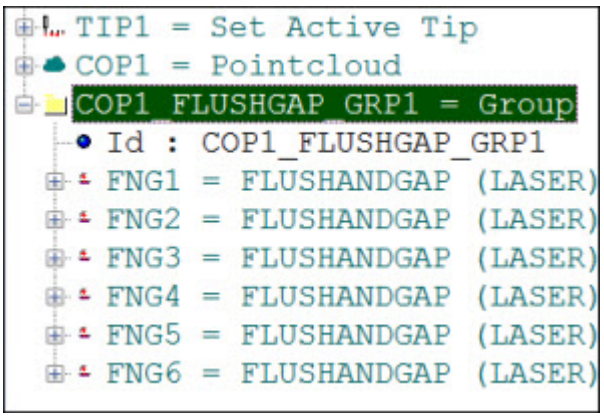


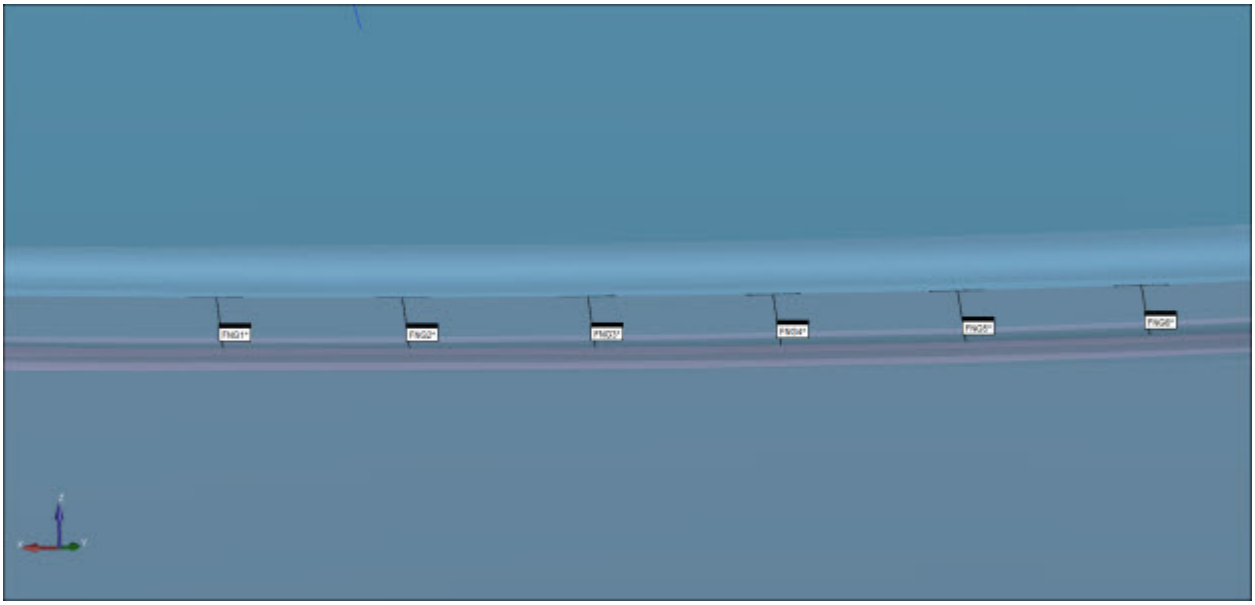


Flush and Gap Auto feature - Additional curve selection

To select additional curves, continue to press and hold the Ctrl key to create the Flush and Gap features.

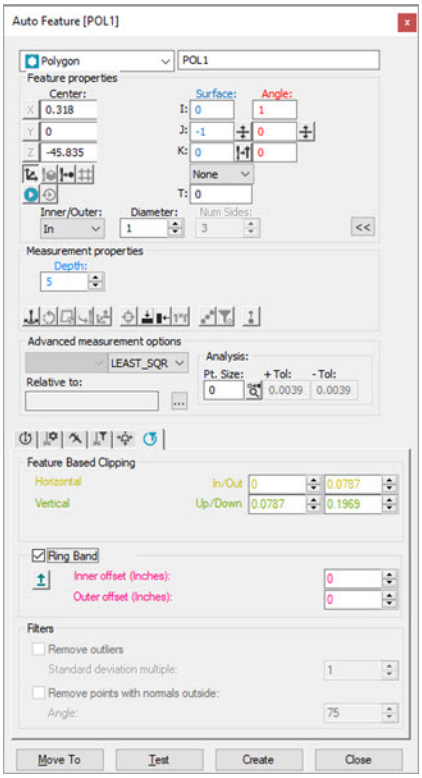
Result





Flush and Gap Auto feature - Result

Laser Polygon



Auto Feature dialog box - Polygon

You can only use this dialog box to measure a hexagonal feature (a polygon with 6 sides).

To measure a hexagonal feature with a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Polygon** to open the **Auto Features** dialog box for the polygon feature.
2. Do one of the following:
   - Perform clicks on the CAD to give the polygon a location and vector. Type in any remaining information.
   - Move the machine to the sphere location using the **Laser** tab of the Graphic Display window. Click the **Read Point from Position** button ( ). Type in any remaining information, such as the diameter, as needed.
   - Type in all of the theoretical information for X, Y, Z, I, J, K, Diameter, and other parameters.
3. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

4. Click the **Test** button to test the feature before you create it.

> **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.

5. Click **Create** and then **Close**.

Polygon Specific Parameters

Num Sides - This parameter defines the number of sides used on the polygon. For laser devices, the number of sides for the Auto feature Polygon is fixed at 6.

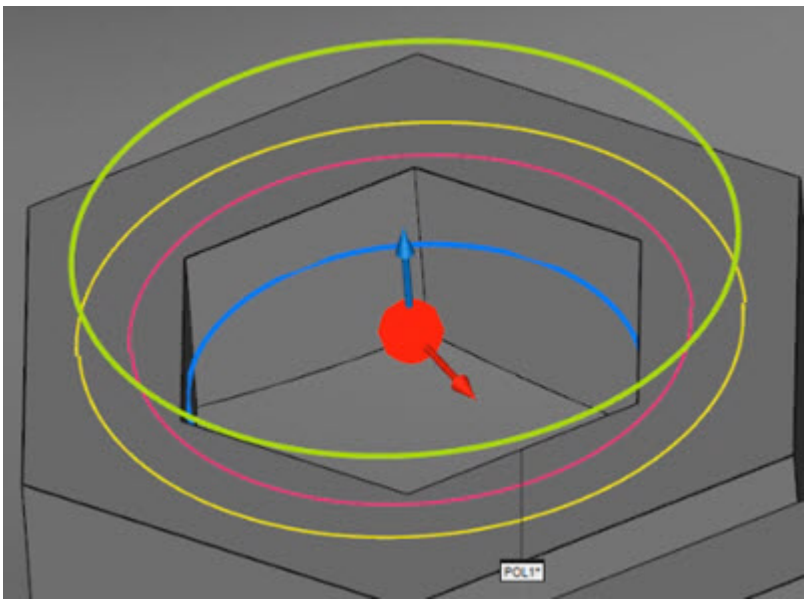
Diameter - The value in this box defines the polygon's diameter.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 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 it to be calculated at that depth. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

> The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you probably need to specify a depth to get accurate results.

> Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample Polygon Auto feature in the Graphic Display window showing:

- *the ring band (pink circles)*
- *the horizontal overscan (yellow circle)*
- *the vertical overscan (green circles)*
- *the depth (blue)*

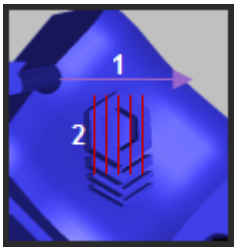
Polygon Command Mode Text

The Auto Polygon command in the Edit window in Command mode looks like this:

```
POL1 =FEAT/LASER/POLYGON,CARTESIAN
   THEO/
<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-0.5,0>,0.5118
   ACTL/
<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-0.5,0>,0.5118
   TARG/<1.0379,1.9488,0.5906>,<0,0,1><0.8660254,-0.5,0>
   NUMSIDES=6
   DEPTH=0
   SHOW FEATURE PARAMETERS=NO
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=30,OVERLAP=0.0394
      OVERSCAN=0.0787,EXPOSURE=35
      FILTER=NONE
      PIXEL LOCATOR=GRAY SUM,Min=30,Max=300
      CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
      RINGBAND=OFF
```

Auto Polygon Paths

PC-DMIS uses the **Angle** IJK vector to determine the direction of the scan.



The feature's scan lines or laser stripes (shown in 2) are perpendicular to the feature's angle vector (shown in 1)

Laser Cylinder



Auto Feature dialog box - Cylinder

To measure a cylinder with a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Cylinder** to open the **Auto Features** dialog box for the cylinder feature.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:
    - Perform clicks on the CAD to give the cylinder location and vector. Type in any remaining information.

- Type in all of the theoretical information for X, Y, Z, I, J, K, Inner/Outer value, diameter, length, depth, and other parameters.
- From the Graphic Display window, use the **Laser** tab to move the machine to the cylinder location. Next, from the **Feature properties** area, click **Read Point from Machine**  . Type in any remaining information, such as the Inner/Outer value, diameter, length, and other parameters.
- In the Measurement properties area, type in the **Center Offset**, **Search Length**, **Start Angle**, **End Angle** and **Direction**.

  The **Start Angle** and **End Angle** properties are helpful when you need to measure fractions of a cylinder. This is especially so when:

  - The Start Angle and End Angle are already defined in the CAD.
  - The CAD model already defines a complete cylinder, but the actual pointcloud data is not complete.
  - The CAD model defines the Start Angle and End Angle the wrong way.

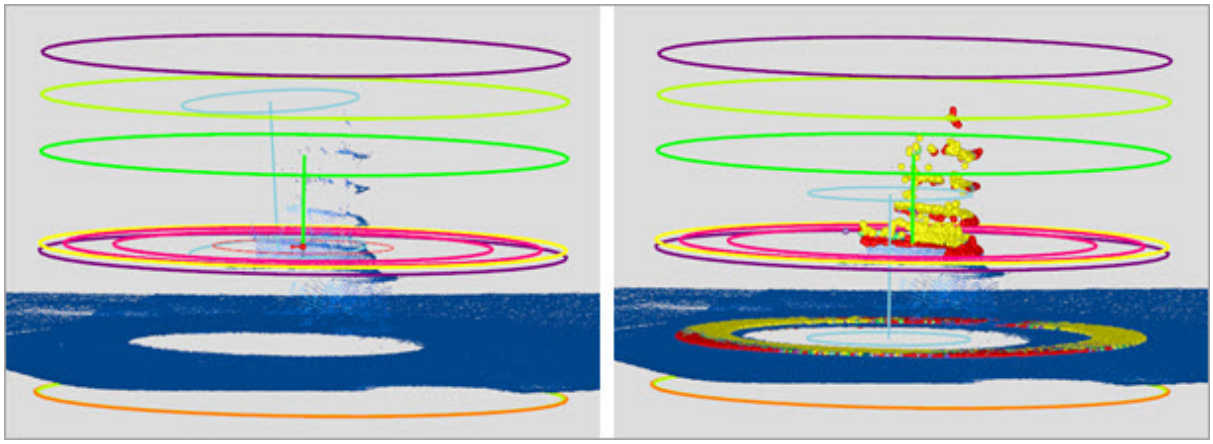
  To define the **Start Angle** and **End Angle** properties, do one of these:

  - Learn them from the CAD model with a single click.
  - Use the dialog box to type in the values or use the spinners to increment or decrement the values.
  - You can edit the existing values and select the **Direction** value in the dialog box.

  These parameters are then passed into PC-DMIS to extract the feature.

4. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.
5. The Probe Toolbox **Feature Extraction** tab has an additional feature called **Constrained Axis** that you can use to constrain the Cylinder feature to the vector of the plane surrounding the cylinder. The plane is defined by the Ring Band area. The **Constrained Axis** option is only available for the Cylinder feature and is only available when you select the **Ring Band** option.

   You can see the before and after effects of the **Constrained Axis** option in the images below. Even with a sparse set of data as shown below, this option calculates a more stable cylinder feature constrained to the plane's vector.



Examples with the Constrained Axis not selected (left) and selected (right)

6. Click the **Test** button to test the feature before you create it.

> **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.

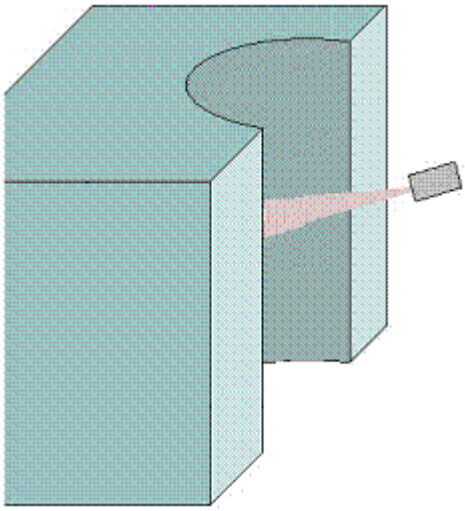
> The location and direction vector of the feature defines the center axis of the cylinder.

Cylinder Specific Parameters

Diameter - The value in this box defines the cylinder's diameter.

Length - The value in this box provides the length (height) of the cylinder's axis. The length parameter is only valid as a nominal. The software does not actually measure the length.

Inner/Outer - This parameter defines whether or not the cylinder is an inner cylinder (hole) or an outer cylinder (including a stud).



Unlike most other laser Auto features, for the **Overscan** value on the **Laser Scan Properties** tab of the **Probe Toolbox**, you should use negative values.

This limits measurement in the cylindrical region along the cylinder's axis.

Depth - 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 because you can specify how far or close the laser is to the cylinder's surface. A depth of 0 for an internal feature means that the center of the laser sensor is on the cylinder's center axis. For an external feature, it is on the surface of the outer cylinder.

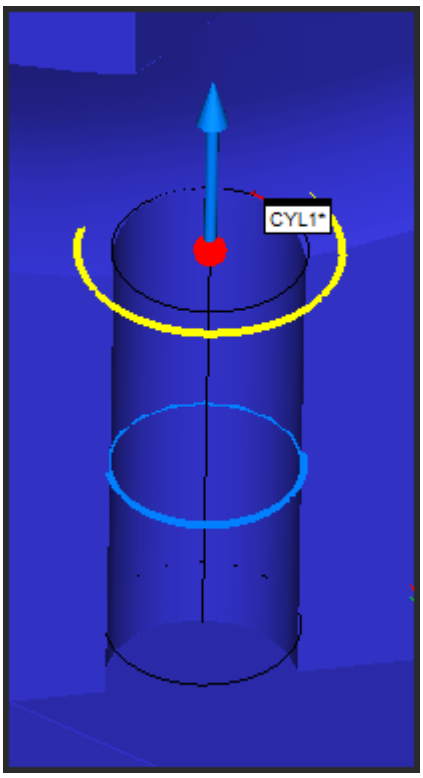
- A negative depth value moves the laser sensor's center away from the cylinder's surface.
- A positive depth value moves the laser sensor's center closer to the cylinder's surface.

Center Offset - This value identifies the center of the cylinder portion of the stud.

Search Length - This value identifies the length of the cylinder portion.

The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

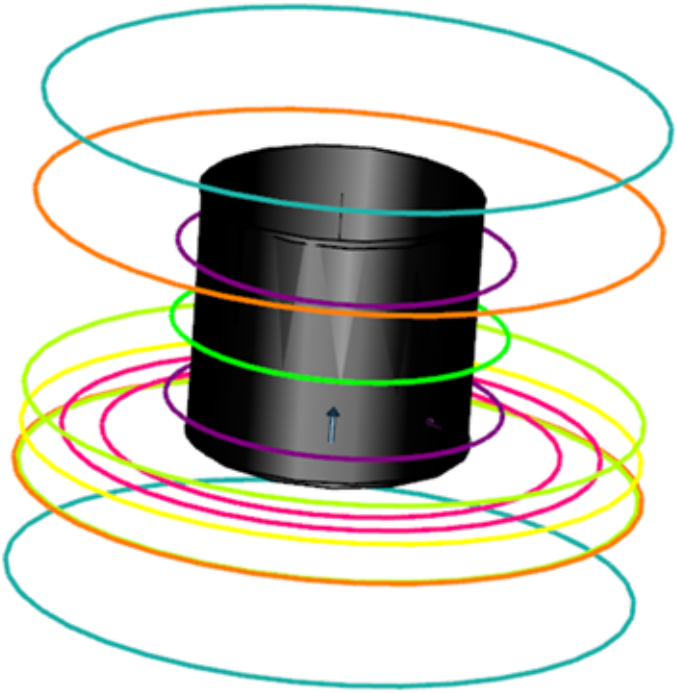
Sample Inner Cylinder



Sample Inner Cylinder that shows:

- *The **Depth** (blue circle)*
- *The **Length** (bottom black circle)*
- *The **Center Point** (yellow circle)*

Sample Outer Cylinder



Sample Stud Cylinder that shows:

- *The **Search Length** (purple circles)*
- *The **Center Offset** (lime green circle)*
- *The **Point Segregation** (orange circles)*
- *The **Center Point** (yellow circle)*
- *The **Clipping Plane** (light green circles)*
- *The **Overscan** (sea green circles)*
- *The **Ring Band** (pink circles)*

Cylinder Command Mode Text

The Auto Cylinder command in the Edit window in Command mode looks like this:

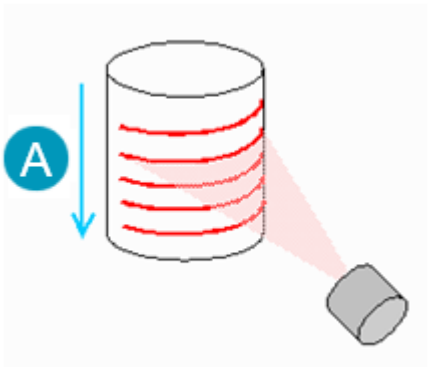
```
CYL1=FEAT/LASER/CYLINDER/DEFAULT,CARTESIAN,OUT,LEAST_SQR
   THEO/<79.82,69.948,0>,<0,0,-1>,7.3,6
   ACTL/
<79.807,70.181,0.031>,<0.0201382,-0.0170408,0.999652>,6.66,6
.35
   TARG/<79.82,69.948,0>,<0,0,1>
   ANGLE VEC=<1,0,0>
   DEPTH=0
   CENTER OFFSET=3.175
   SEARCH LENGTH=4
   START ANG=0,END ANG=360
   DIRECTION=CCW
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THICKNESS_NONE,0.7
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=YES,0.2,0.01,0.01
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=COP1
      HORIZONTAL CLIPPING=5
      UP VERTICAL CLIPPING=2,DOWN VERTICAL CLIPPING=2
      RINGBAND=ON,INNER OFFSET=2,OUTER OFFSET=4,CONSTRAINED
AXIS=OFF
      OUTLIER_REMOVAL=OFF
      REMOVE POINTS WITH NORMALS OUTSIDE=OFF
```

Auto Cylinder Paths

Cylinder Measurements

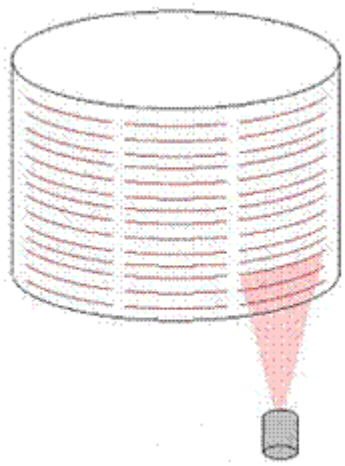
Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. The laser plane must be roughly normal to the cylinder axis (< 30 degree deviation). Depending on the diameter of the cylinder, PC-DMIS takes one of these paths when performing the measurement:

Path 1: Single Scan



Cylinders with a diameter less than the usable portion of the stripe. A is the scan motion.

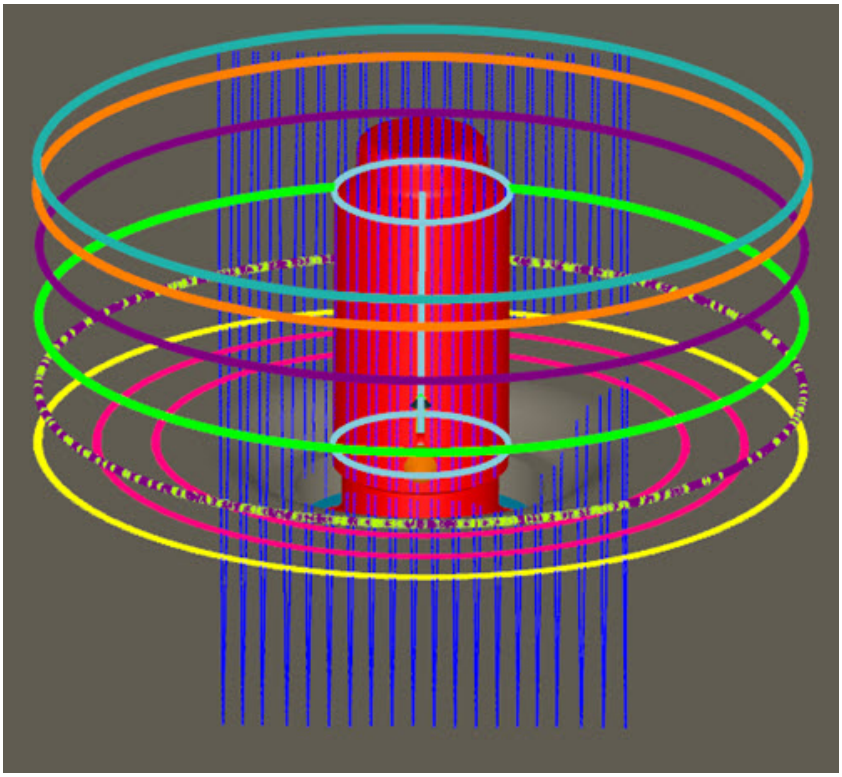
Path 2: Multiple Scans



Cylinders with a diameter larger than the usable portion of the stripe

When you perform a laser scan of a feature 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.

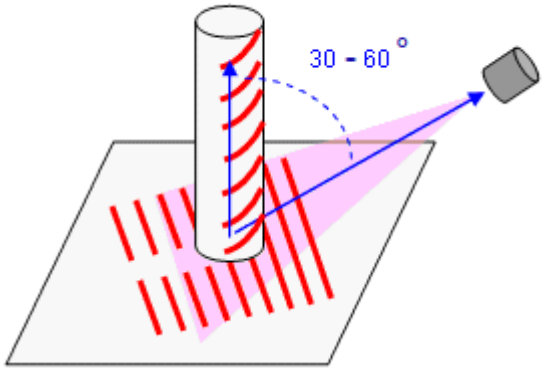
Example path creation for directly measured Cylinder laser feature when using an Auto Wrist:



Stud Measurements

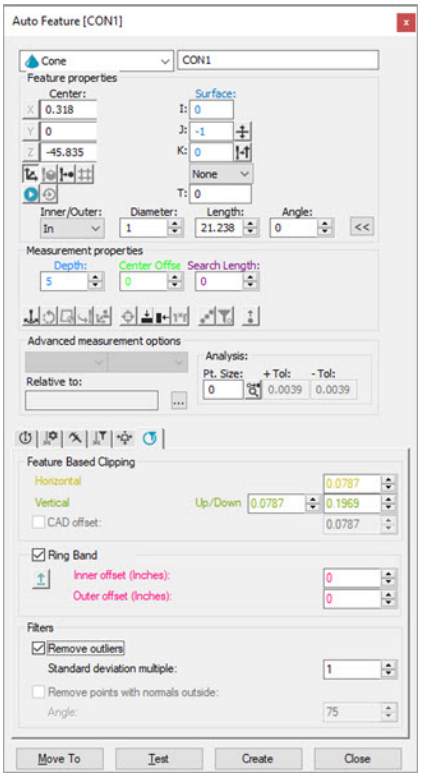
Single Scan

Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. The laser plane must be roughly 30-60 degrees to the cylinder's axis. The scan must capture the region on the base plane of the stud where the cylinder is mounted.



Single pass laser scan on stud cylinder

Laser Cone



Auto Feature dialog box - Cone

To measure a cone with a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Cone** to open the **Auto Features** dialog box for the cone feature.

2. From the **Inner/Outer** box, select **In** or **Out**.

3. Do one of the following:

   - Click on the CAD to give the cone location and vector. Type in any remaining information.
   - From the Graphic Display window, use the **Laser** tab to move the machine to the cone location. Next, from the **Feature Properties** area, click the **Read Point from Position** button (). Type in any remaining information such as the inner or outer values, diameter, length, and other parameters.
   - Type in the theoretical information for X, Y, Z, I, J, K, inner or outer values, diameter, length, depth, and other parameters.

4. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.

5. Click the **Test** button to test the feature before you create it.

   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.

6. Click **Create** and then **Close**.



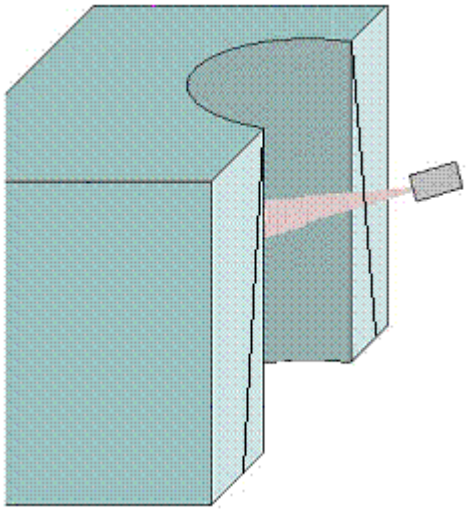
The location and direction vector of the feature define the center axis of the cone.

Cone Specific Parameters

Diameter - The value in this box defines the cone's diameter.

Length - The value in this box provides 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.

Inner/Outer - This parameter defines whether or not the cone is an inner cone (hole) or an outer cone (stud).



Unlike most other laser Auto features, for the **Overscan** value on the **Laser Scan Properties** tab of the **Probe Toolbox**, you should use negative values.

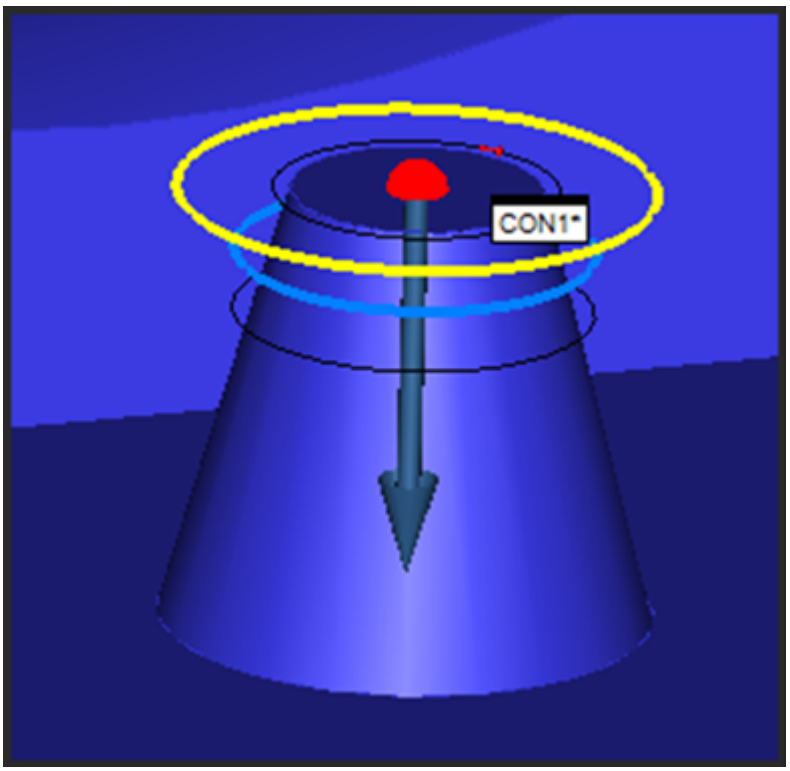
This limits measurement in the conical region along the cone axis.

Depth - 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.

Center Offset - This value identifies the center of the cone portion of the stud.

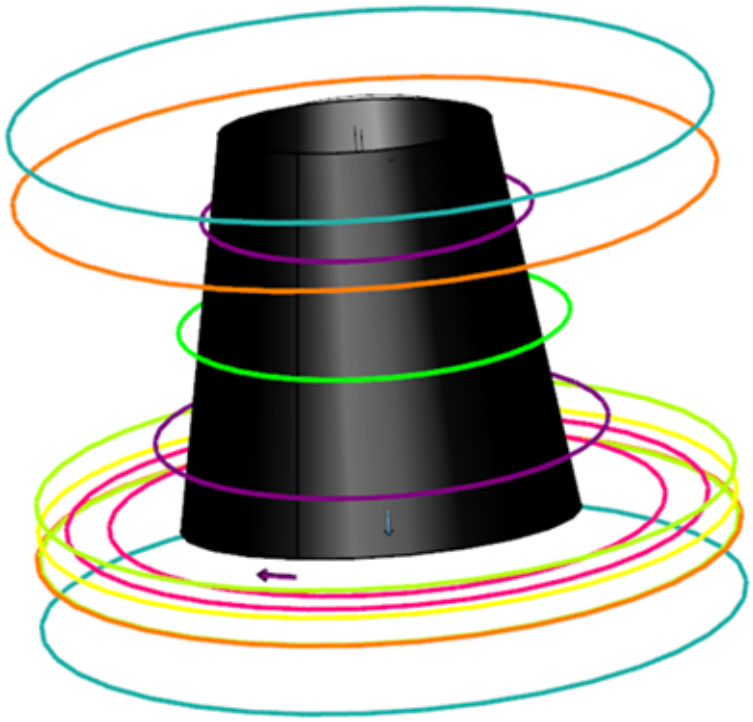
Search Length - This value identifies the length of the cone portion.

The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.



Sample external cone in the Graphic Display window showing:

- *The **Diameter** (top black circle)*
- *The **Length** (bottom black circle)*
- *The **Depth** (blue circle)*
- *The **Center Point** (yellow circle)*



Sample external stud cone in the Graphic Display window showing:

- *The **Search Length** (purple circles)*
- *The **Center Offset** (lime green circle)*
- *The **Point Segregation** (orange circles)*
- *The **Center Point** (yellow circle)*
- *The **Clipping Plane** (light green circle)*
- *The **Overscan** (sea green circles)*
- *The **Ring Band** (pink circles)*

Cone Command Mode Text

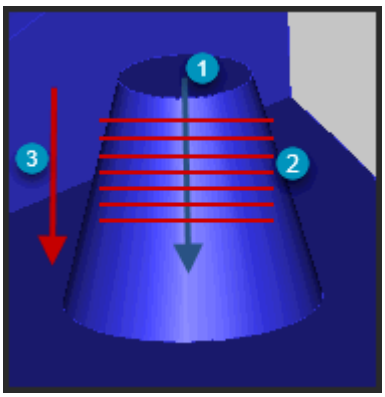
The Auto Cone command in the Edit window in Command mode looks like this:



```
CON1 =FEAT/LASER/CONE/DEFAULT,CARTESIAN,OUT
   THEO/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
   ACTL/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
   TARG/<3.1425,2.7539,0>,<0,0,1>
   DEPTH=0
   CENTER OFFSET=3
   SEARCH LENGTH=2
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THEO_THICKNESS,0
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=YES
      GRAPHICAL ANALYSIS=NO
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=COP1
         SOUND=OFF
      HORIZONTAL CLIPPING=0.0787,VERTICAL CLIPPING=0.0787
   RINGBAND=ON,INNER OFFSET=0.5,OUTER OFFSET=2
   OUTLIER_REMOVAL=ON,1
```

Auto Cone Paths

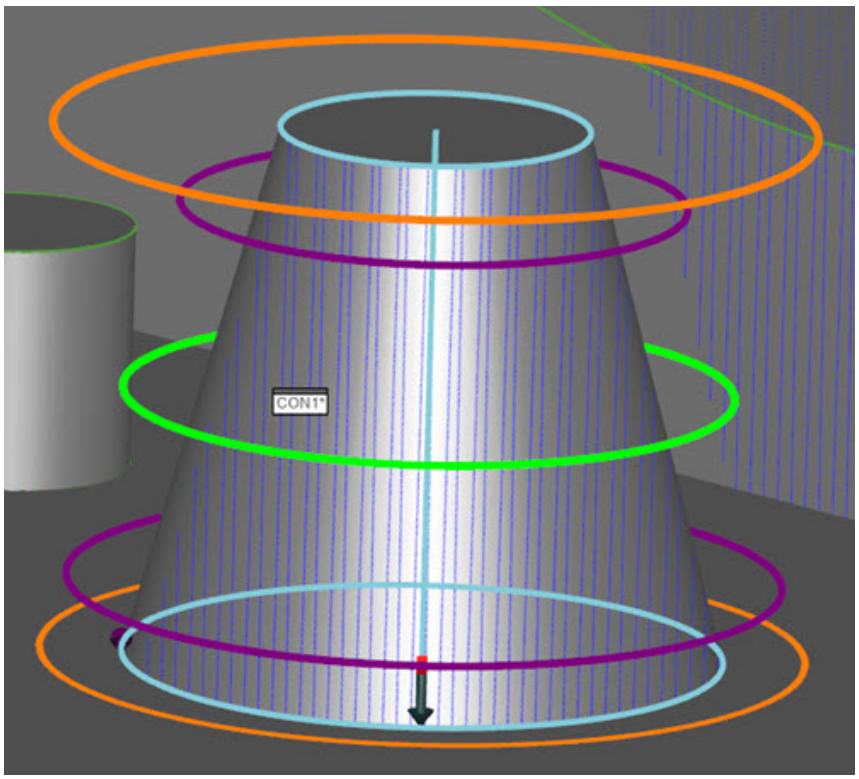
The laser sensor scans along the length of the cone. It moves in the direction of the cone's vector. The laser must be nearly perpendicular to that vector.



1 - The feature's vector.
2 - The feature's scan lines or laser stripes are perpendicular to the feature's vector.
3 - Scan direction follows the feature's vector

When you perform a laser scan of a feature 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.

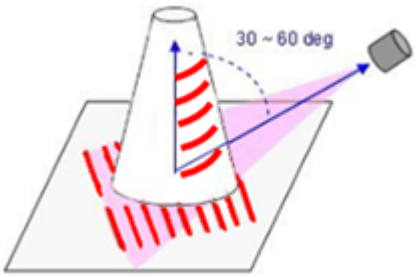
Example path creation for directly measured Cone laser feature when using an Auto Wrist:



Stud Measurements

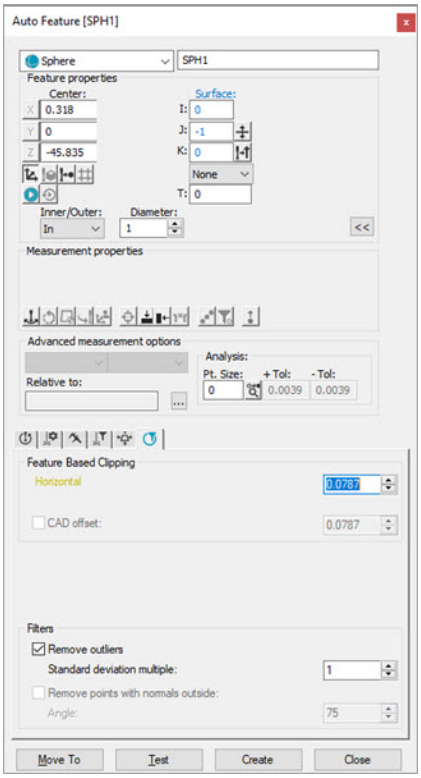
Single Scan

Adjust the processing window in the Laser View to include as much of the cone's surface as possible. The laser plane must be roughly 30-60 degrees to the cone axis. The scan must capture the region on the base plane of the stud where the cone is mounted.



Single pass laser scan on stud cone

Laser Sphere



Auto Feature dialog box - Sphere

To measure a sphere with a laser sensor:

1. From the menu, click **Insert | Feature | Auto** and select **Sphere** to open the **Auto Features** dialog box for the sphere feature.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:
    - Click on the CAD to give the sphere a location and vector. Type in any remaining information.
    - From the Graphic Display window, use the **Laser** tab to move the machine to the sphere. Next, from the **Feature Properties** area, click the **Read Point from**

Position button ( ). Type in any remaining information, such as the Inner/Outer value, diameter, and other parameters.

- Type in the theoretical information for X, Y, Z, I, J, K, Inner/Outer value, diameter, and other parameters.

4. Cycle through each of the Probe Toolbox tabs and type in the necessary information. For details on a specific Probe Toolbox tab, see the appropriate topic in the "Using the Probe Toolbox in PC-DMIS Laser" section of the PC-DMIS Laser documentation.
5. Click the **Test** button to test the feature before you create it.

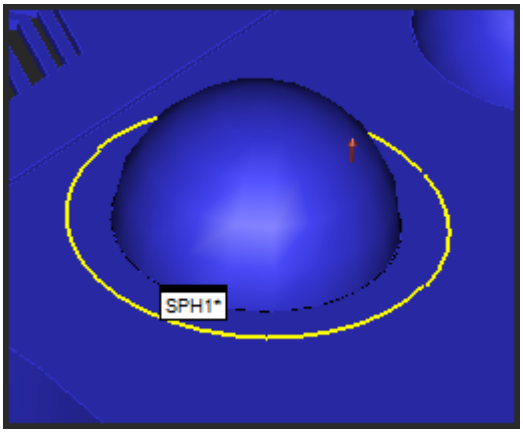
> **WARNING:** When you do this, the machine moves. To avoid injury, stay clear of the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

Sphere Specific Parameters

Inner/Outer: This parameter defines whether or not the sphere is an inner sphere (concave) or an outer sphere (convex).

Diameter: The value in this box defines the sphere's diameter.



Sample outer sphere in the Graphic Display window showing the overscan (yellow circle)

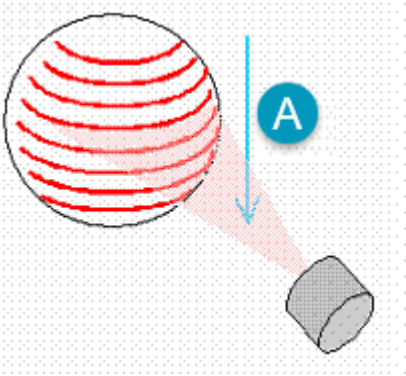
Sphere Command Mode Text

The Auto Sphere command in the Edit window in Command mode looks like this:

```
SPH1 =FEAT/LASER/SPHERE,CARTESIAN,IN,LEAST_SQR
   THEO/<1.895,1.91,1>,<0,0,1>,1.895
   ACTL/<1.895,1.91,1>,<0,0,1>,1.895
   TARG/<1.895,1.91,1>,<0,0,1>
   START ANGLE 1=0,END ANG 1=0
   START ANGLE 2=0,END ANG 2=0
   SHOW FEATURE PARAMETERS=YES
      SURFACE=THEO_THICKNESS,0
      MEASURE MODE=NOMINALS
      RMEAS=NONE,NONE,NONE
      AUTO WRIST=NO
      GRAPHICAL ANALYSIS=NO
      FEATURE LOCATOR=NO,NO,""
   SHOW_LASER_PARAMETERS=YES
      REFERENCE ID=DISABLED
      SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
      FILTER=NONE
```

Auto Sphere Path

The direction of the path is determined based on the stripe.



Path direction of scan
(A) Scan motion

Clearing Auto Feature Scan Data

PC-DMIS's laser Auto features sometimes store scanned data as an internal pointcloud following their creation. This occurs if the Pointcloud parameter on the Laser Scan Properties tab is set to **Disabled**.

Two menu items exist to clear out this internal data based on your needs. Located under the **Operations | Laser Autofeatures** sub menu, these menu items remove the internal data, thereby helping to reduce measurement routine size:

- **Clear All Scan Data Now** - When you select this check box, PC-DMIS immediately deletes all the internal pointclouds from all the laser Auto features in the measurement routine.
- **Clear All Scan Data after Execution** - When you select this check box, any time you execute a measurement routine that has a laser Auto feature, PC-DMIS deletes its internal pointcloud data following execution. By default, this menu item is unmarked.



This only operates on internal pointclouds from Auto features. It does not affect COP commands in the measurement routine.

Scanning Your Part Using a Laser Sensor

When you scan the surface of the part with a laser sensor, you can define an area of measurement. The software collects a group of point data that it passes to the reference Pointcloud object in the measurement routine. Note that when you work with pointclouds and scans, the scans themselves DON'T contain any data. They only define the machine movement. The Pointcloud object always stores the point data.

The main topics in this section discuss the scanning options available from the **Insert | Scan** submenu when you use a laser sensor:

- Introduction to Performing Advanced Scans
- Common Functions of the Scan Dialog Box
- Changing the Scan Speed
- Performing a Linear Open Advanced Scan
- Performing a Patch Advanced Scan
- Performing a Perimeter Advanced Scan
- Performing a Freeform Advanced Scan
- Performing a Grid Advanced Scan
- Performing a Manual Laser Scan on DCC Machines
- Setting Machine Speed for Scanning
- CWS Parameter Tab

Introduction to Performing Advanced Scans

Advanced scans are DCC continuous-motion scans that follow a predefined path. PC-DMIS follows the predefined path regardless of the shape of the actual part. The path can be defined in several ways that are explained later.

These advanced scans use a laser scanning probe. This enables you to automatically digitize surfaces.

To perform an advanced scan:

1. Specify the necessary parameters for the DCC scan that you selected.
2. Click the **Generate** button. PC-DMIS generates the scan.
3. Once it finishes, click the **Create** button. The PC-DMIS scanning algorithm then takes control of the measurement process.

The types of advanced scans that PC-DMIS supports include:

- Linear Open Scan
- Patch Scan
- Perimeter Scan
- Freeform Scan
- Grid Scan
- Manual Laser Scan on DCC Machines

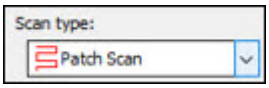
This document first covers the common functions that are available in the **Scan** dialog box (the dialog box that you use to perform these scans). It then describes how to perform the available advanced scans.

For information on setting your machine's scan speed, see "Setting the Machine Speed for Scanning".

Common Functions of the Scan Dialog Box

Many of the functions described below are common to both DCC and Manual scans. Functionality that relates specifically to one scan mode is appropriately indicated.

Scan Type



Scan type list

Use the **Scan type** list in the **Scan** dialog box to change scan types without closing the dialog box and selecting a different scan type.

ID

The **ID** box in the **Scan** dialog box displays the ID of the scan to be created.

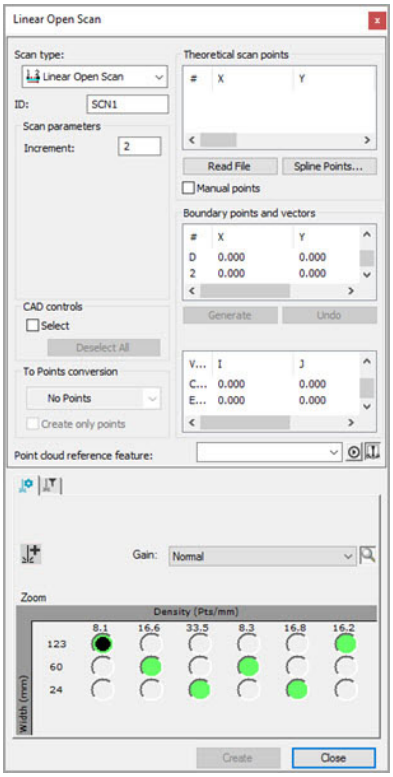
Scan Parameters Area

The **Scan parameters** area in the **Scan** dialog box provides different controls depending on the type of scan that is being performed. See the specific topics located under each scan type:

- Linear Open Scan Parameters
- Patch Scan Parameters
- Perimeter Scan Parameters
- Grid Scan Parameters

CAD Controls Area

Click the **Advanced >>** button in the **Scan** dialog box to display the full dialog box if necessary.



Scan dialog box for Linear Open Scan

Click the **Graphics** tab to display the **CAD controls** area. You can use this area to specify the CAD surface elements that define the "Theoretical Points".



CAD controls area

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 to use to generate the scan. 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 scan generation is successful.

To use this functionality to select CAD surfaces, you must have the ability to import and use CAD surface data. Ensure that you select the **Draw Surfaces** button ( ). If you don't, when you click on the CAD model, the nearest wire gets selected instead of the selected surface.

To scan specific surfaces:

1. Select the **Select** check box.

2. Click on the appropriate surfaces. Once a CAD surface is selected, it is highlighted in the Graphic Display window. The status bar displays the number of surfaces that you selected.

If you mistakenly select a surface, press Ctrl, and click on that surface a second time. This deselects the surface. Clicking the **Deselect All** button deselects all highlighted surfaces at once.

Once you are done selecting surfaces, clear the **Select** check box. The selected surfaces are kept.

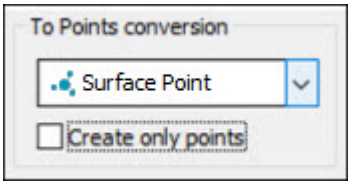
If you clear the **Select** check box, PC-DMIS assumes any clicks on the surface to be those that create the scan path.

The following options are available:

Select check box - Enables you to select the CAD surface and wireframe elements used to find the nominal.

Deselect All button - Deselects all of the highlighted surfaces at once that were created with the **Select** check box.

To Points Conversion Area



To Points conversion area

The **To Points conversion** area in the **Scan** dialog box enables you to create laser Point commands. The commands start from the points that make up the scan.

Hit Type List

The default setting is **No Points**.

For a Perimeter scan, you can select either Surface Point or Edge Point in the list. For all other types of scans, you can select only Surface Point.

The points are collected in a collapsed `GROUP` command. The name of the command includes the name of the related scan, the pointcloud associated with it, and the point ID preceded by "Edge" (if you selected Edge Point).

Surface Point Group Command Mode Text

The following is an example of a collapsed `GROUP` command collecting Surface Points:

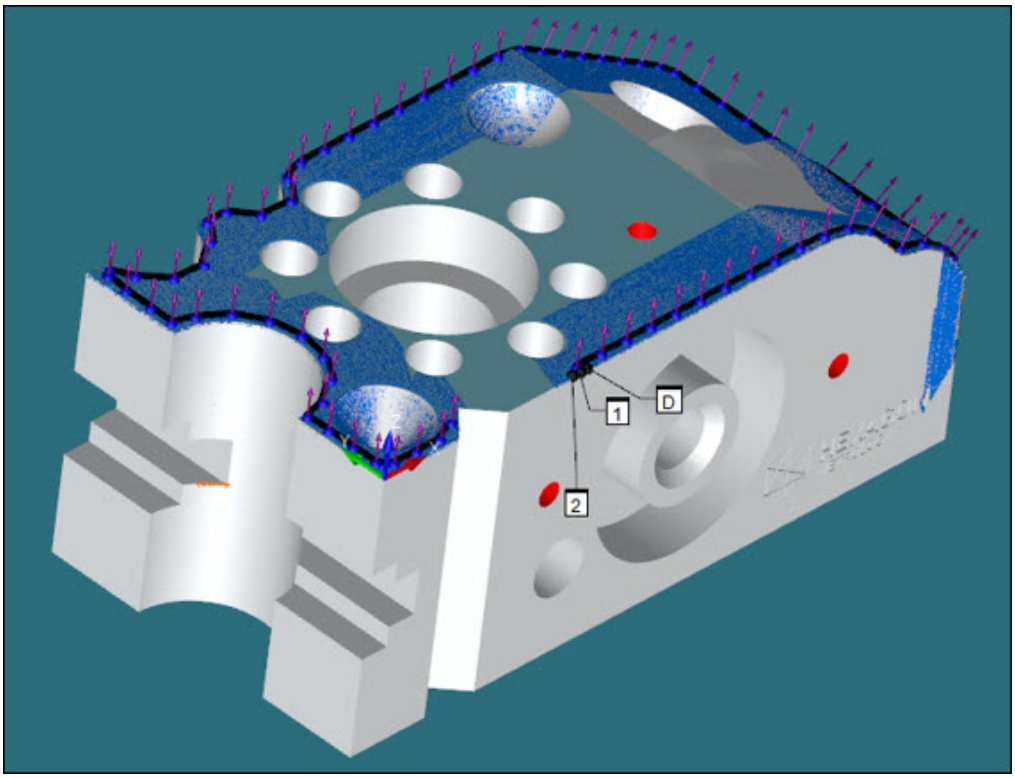
```
COP             = COP/DATA,TOTAL SIZE=468492,REDUCED
SIZE=468492,
                  FINDNOMS=NO,REF,SCN1,,
SCN1              =  FEAT/SCAN,PERIMETER,NUMBER OF HITS=4,
                  SHOW
HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
                  MEAS/SCAN
                  BASICSCAN/PERIMETER,NUMBER OF HITS=4,
                  SHOW HITS=NO,SHOWALLPARAMS=NO
                  ENDSCAN
                  ENDMEAS/
SCN1_COP_PNT_GRP1=GROUP/SHOWALLPARAMS=NO
                  EXECUTION CONTROL=AS MARKED
                  ENDGROUP/ID=SCN1_GRP1
```

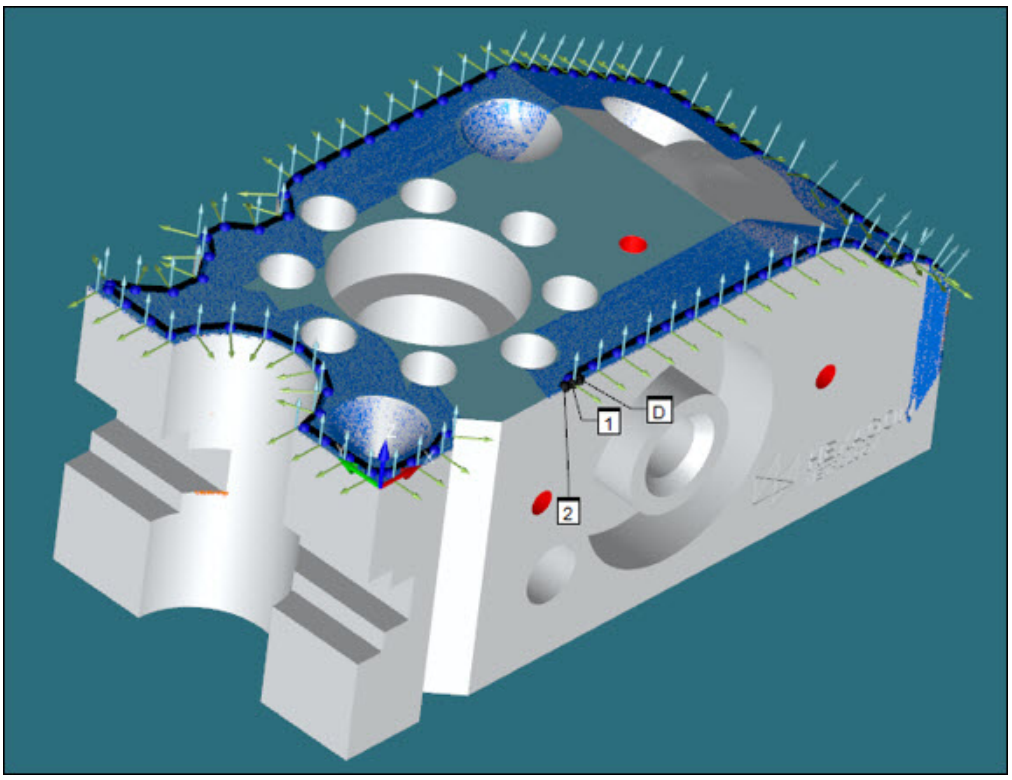
The following is an example of a `GROUP` command collecting Edge Points:

```
SCN2       =FEAT/SCAN,PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
            MEAS/SCAN
            BASICSCAN/PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO
            ENDSCAN
            ENDMEAS/
SCN2_COP_EDGEPNT_GRP2=GROUP/SHOWALLPARAMS=YES
            EXECUTION CONTROL=AS MARKED
PNT5         =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<133.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<133.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<133.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
            SPACER=0.5
            SHOW FEATURE PARAMETERS=NO
            SHOW_LASER_PARAMETERS=YES
              REFERENCE ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
PNT6       =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<138.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<138.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<138.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
            SPACER=0.5
            SHOW FEATURE PARAMETERS=NO
            SHOW_LASER_PARAMETERS=YES
              REFERENCE ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
PNT7       =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<143.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<143.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<143.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
            SPACER=0.5
            SHOW FEATURE PARAMETERS=NO
            SHOW_LASER_PARAMETERS=YES
              REFERENCE ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
            ENDGROUP/ID=SCN2_COP_EDGEPNT_GRP2
```

Surface Points and Edge Points are extracted from the COP that you specified in the scan.

Consider the following figures that show Surface Points and Edge Points extracted from a COP using the **Scan** dialog box for a Perimeter scan:





Create only points

If you select the **Create only points** check box, PC-DMIS does not create the scan command. In this case, the `GROUP` command does not contain the name of the scan.



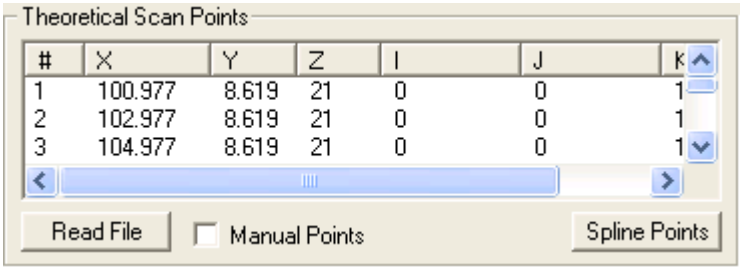
The `SCAN` command precedes the `GROUP` command in the Edit window if you create both commands.

Theoretical Scan Points Area

You can define the Theoretical Points of a scan through any of these actions:

- Reading them from a file
- Reading machine positions
- Generating them from the defined boundary points
- Using CAD data

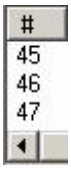
These topics are discussed in greater detail later in this section.



Theoretical Scan Points area

Editing Theoretical Points

To edit theoretical points, double-click the number of the desired point in the # column.



column

This displays the **Edit Theo Data** dialog box. Use this dialog box to edit the X, Y, Z, I, J, K values. The dialog box's title bar displays the ID of the point you are editing.



Edit Theo Data dialog boxes depicting Next, Previous, and Flip Vector buttons

Click the **Next** or **Previous** buttons to cycle between the theoretical points.

Click the **Flip Vector** button to flip the vector for the selected point.

Deleting Theoretical Points

You can easily clear the **Theoretical Points** list of any of the scan types. Right-click inside the **Theoretical Points** list. A **Reset Theoretical Points** prompt appears. Click the prompt to clear any points from the list.

Read File

The **Read File** button tells PC-DMIS to read the Theoretical points in from a text file. The points must be in X,Y,Z,I,J,K comma delimited format. A blank space between points denotes the start of a new scan line.

Manual Points

By selecting the **Manual Points** check box, you can manually add points into the **Theoretical Points** list. To take these points, move the probe to the desired location and click the **Probe Enable** button on your jog box, or click points on the CAD file.

New Line

The **New Line** check box functions only for Patch Scans. By selecting the **New Line** check box, you tell PC-DMIS that manual points you take should begin a new line.

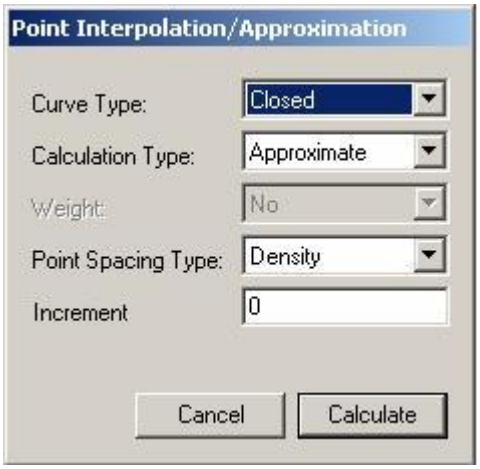
Spline Points

When you take manual points, the spacing and path are usually inconsistent. With the **Spline Points** button, however, you can construct a spline curve along a path through a list of manual points and create a smooth evenly-spaced path. For a Linear Open scan, PC-DMIS places all the points on the Cut Plane. For a Patch Scan, it places the points for each scan line on the Cut Plane for that scan line.



The **Spline Points** button is not available for a Perimeter scan.

Clicking the **Spline Points** button displays the **Point Interpolation / Approximation** dialog box.



Point Interpolation/Approximation dialog box

Curve Type

There are three types of curves that can be constructed with the spline routines:

Open - This option creates an open-ended curve. This means the curve starts in one location and ends in another.

Closed - This option creates a closed-ended curve. This means the curve starts and ends in the same location.

Line - This option differs from the **Open** or **Closed** options. It doesn't use theoretical points but instead uses boundary points and creates straight lines within the boundary points, following the boundary points' direction rules.

Calculation Type

There are two calculation types you can use in spline routines.

Approximate: This option allows the path to deviate from the actual input point by a small amount in order to produce a smooth curve from which the new points are taken.

Interpolate: This option forces the curve to go through each of the input points exactly.

Weight

This list becomes available when you select the **Approximate** calculation type. When constructing the curve, it allows more weight to be given to points that are further apart. The two choices for the option are **YES** and **NO**.

Point Spacing Type

This option allows you to control the output points of the spline routine.

Density: This option lets you specify the incremental distance between each output point. PC-DMIS determines the number of output points by the length of the curve and the user supplied increment.

Number of Hits: This option lets you specify how many points you want in the output. No matter the length of the curve, PC-DMIS evenly spaces the user supplied points over the length of the curve.

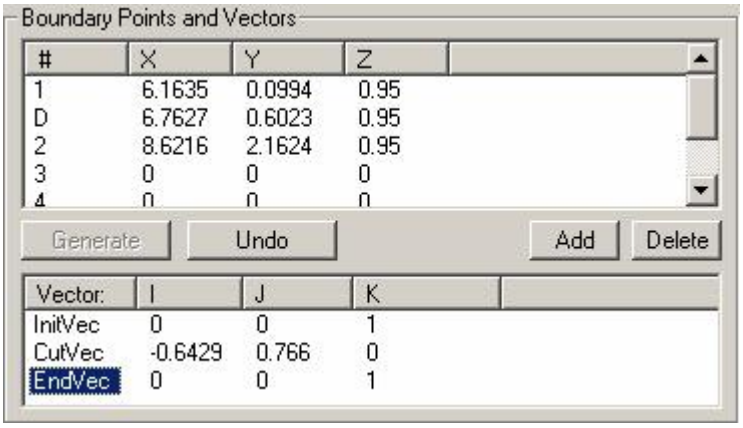
Increment

This box holds the increment value for the Point Spacing Type; either **Density** or **Number of Hits**.

Boundary Points Area

PC-DMIS lets you define the boundary of a scan. You can do this in these ways:

- Type the XYZ values for the individual boundary points directly
- Measure the points using the laser sensor
- Use the CAD data



Boundary Points and Vectors area

Boundary Points are not available or necessary for Free-form scans

You can change the column widths of the **Boundary Point** list if you click and drag the right or left edge of a column header to the desired size. The software saves this information to your PC-DMIS Settings Editor each time it changes.

Setting Boundary Points by Typing

To set the boundary of a scan by typing:

1. Double-click the desired boundary point in the '#' column to display 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.

Click **Next** to accept the changes and display the next boundary point for editing.

Setting Boundary Points Using the Measured Point Method

To set the boundary of the scan using measured points:

1. Place the laser sensor at the desired location.
2. On the jog box, press the **Probe Enable** button (only available on DEA and Brown and Sharpe machines).

- This automatically updates the value of the selected boundary point in the **Boundary Points and Vectors** list. The software then selects the next boundary point (if any) in the list.
- In the case of a PATCH scan, PC-DMIS adds an extra boundary point automatically if the selected 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 click the **OK** button.



The **Probe Enable** light on the jog box alternates between off and on every time you press the **Probe Enable** button. This is not important and has no influence on the probe itself.

Setting Boundary Points Using the CAD Data Method

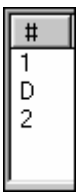
PC-DMIS lets you select the boundary points by using surface CAD data.

When using CAD surface data:

1. Make sure that you have imported solid CAD data.
2. Select the **Draw Surfaces** icon  .
3. Select a boundary point by clicking on the desired location in the Graphic Display window. PC-DMIS highlights the selected surface and automatically updates the value of the currently selected Boundary point. PC-DMIS then moves the focus to the next boundary point (if any are available). For PATCH scans, PC-DMIS automatically adds an extra boundary point if the current point is the last point in the list.

Editing Boundary Points

Boundary points can be edited by double clicking the number of the desired point in the '#' column.



column

This displays the **Edit Scan Item** dialog box allowing you to edit the X, Y, Z values.



Edit Scan Item dialog box

Clearing Boundary Points

You can easily clear the **Boundary Points** list of any of the scan types.

1. Right-click while the cursor is inside the **Boundary Points** list.
2. Click the **Reset Boundary Points** button that appears to reset all the boundary points to zero. The number of boundary points is set to the minimum for each scan type.

Generate

The **Generate** button is available only for DCC scans using CAD data.

After the boundary points for a scan have been defined, click the **Generate** button. PC-DMIS slices the CAD with the plane defined by the start point and cut vector and then generates the theoretical points from the curve defined by this slice. If the **Create** button is then clicked, PC-DMIS inserts a scan with nominal hit data into the measurement routine.

Undo

The **Undo** button allows you to remove the hits that have been generated using the **Generate** button as outlined in the Generate topic.

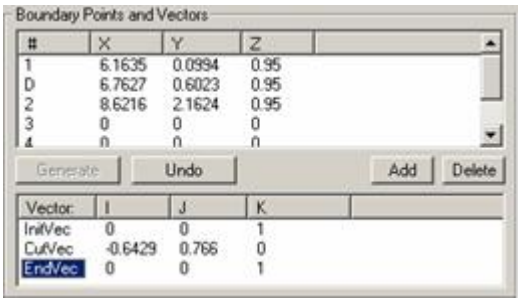
Adding and Deleting Boundary Points



Add / Delete buttons

The **Add** and **Delete** buttons allow you to add or delete boundary points to the list of boundary points. There are some restrictions regarding each type of scan. For example, a LINEAROPEN scan only takes a Start Point, a Direction point and an Ending Point. It will not allow you to add more points or delete these points. Refer to each scan for specific restrictions.

Vectors Area



Boundary Points and Vectors area

The bottom portion of the **Boundary Points and Vectors** area displays a list of vectors that PC-DMIS will use to start and stop a scan. Some of the vectors listed below may not be found in the list for a specific scan, indicating that they are not used for that scan. Please refer to each scan for more details. You can edit each of these vectors by double-clicking on the vector to edit in the vector column.



Vector column

This displays the **Edit Scan Item** dialog box:



Edit Scan Item dialog box

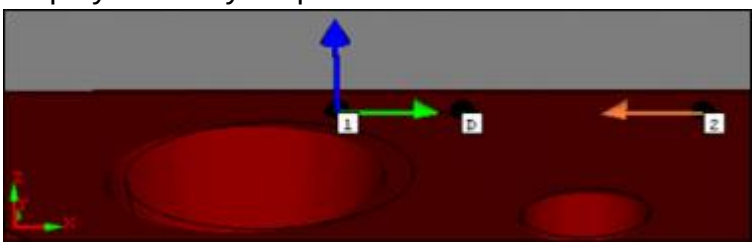
With the I, J, and K boxes, you can edit the I, J, and K values.

- **Next** - This button cycles through the available vectors in the **Initial Vectors** list. Some of the initial vectors can be flipped. If so, then the **Flip** button becomes available on the **Edit Scan Item** dialog box.
- **Flip** - This button reverses the direction of the selected vector.

Graphical Representation of 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 Graphical Display area of your part.



Colored arrows showing vectors

Vector	Graphical Representation
Initial Touch	Blue arrow
Direction	Green arrow
Boundary Plane	Orange arrow

Initial Touch Vector (InitVec)

The values that are displayed in the **Initial Touch Vector** row indicate the vector PC-DMIS will use to take the first touch in the scanning process.

To edit the I, J, K Initial Touch vector:

1. Double-click on **InitVec** in the vector column to open the **Edit Scan Item** dialog box.
2. Change the values.
3. Click the **OK** button to accept the changes and close the dialog box.

Cut Plane Vector (CutVec)

A cut plane is used internally for DCC scanning calculations. This cut plane is derived from the Initial Touch vector, and the vector between the first and last points for the Linear Open DCC scan. Please refer to the individual scans for details on how the Cut Plane vector is derived.

End Touch Vector (EndVec)

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).

Pointcloud Reference Feature

The **Pointcloud Reference Feature** displays a list of the available Pointcloud objects into which PC-DMIS places the scanned surface data. You must select a pointcloud from this list or PC-DMIS cannot create the scan.

Measure

If you select the **Measure** check box and click the **Create** button, PC-DMIS will start measuring the scan immediately. If you don't select the **Measure** check box when you click **Create**, PC-DMIS inserts a scan object into the Edit window that can be measured later. This allows you to set up a series of scans that can be inserted into the Edit window and measured at a later time.

Changing the Scan Speed

You can set the scan speed at theoretical scan points for defined laser scan paths. PC-DMIS then moves the laser probe at this speed after it crosses the theoretical point during the scan.

On a very large part, you can do the following:

- Collect more points in the area where there are features such as circles and slots. To collect more points, you can lower the scan speed where you need to extract the features.
- Collect fewer points in the area where there are no features. To collect fewer points, you can increase the scan speed where there are no features to extract.

These speed change point settings give you more control over the scan speed and can help reduce the scan time.

To change a scan speed, do the following:

1. From the **Theoretical Scan Points** area, select a theoretical scan point.
2. Right-click on the selected theoretical scan point row to display the menu.
3. From the menu, select **Change scan speed** to open the **Edit Scan Speed** dialog box. Type the value in the **Scan speed** box.
4. To calculate the scan speed from the laser stripe spacing distance, click the **Calculate Laser Stripe Spacing** icon  . This opens the **Calculate Scan Speed** dialog box.
5. Type the value in the box and click **Calculate**. PC-DMIS calculates the scan speed in millimeters per second. PC-DMIS then inserts the calculated value in the **Scan speed** box of the **Edit Scan Speed** dialog box.

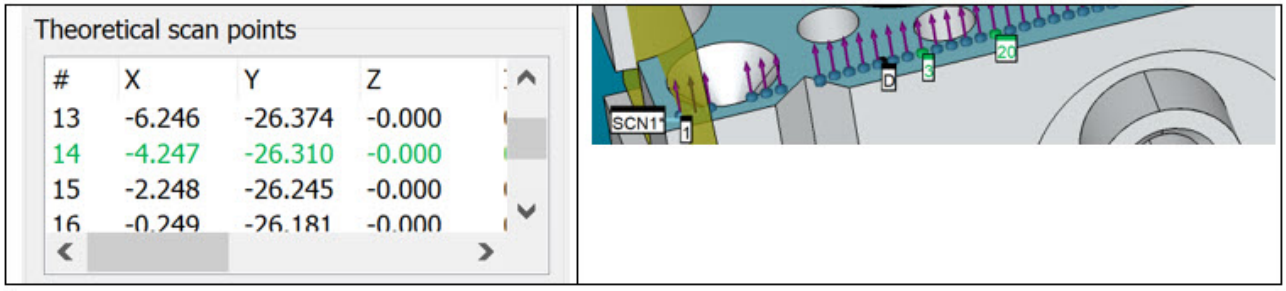
6. Click **OK** to set this scan speed value.

Setting Speed at Multiple Points

To set scan speed at multiple theoretical points, press Ctrl and click the points in the # column. PC-DMIS highlights in yellow all the selected items on the CAD. To deselect a point, press Ctrl and click on the selected point.

To select scan speed change points on the CAD, click the spheres on the CAD in the Graphic Display window. PC-DMIS then highlights the points in yellow. To deselect a point, click on a selected sphere.

To select multiple points on a CAD, click the points to select them. To deselect all points, click anywhere in the gray area of the dialog box.



Textual (green text) and Graphical (green sphere) representation view of theoretical scan points. In the Graphic view, a label with scan speed (in green) shows the speed set at that point.

Right-click Menu

Delete All Theoretical Points - This option deletes all theoretical points in order to recreate your scan.

Change scan speed - This option lets you change the speed at a single theoretical point. The scan now uses this speed.

Reset scan speed - This option resets the scan speed for the selected theoretical point back to the original speed.

Reset all scan speeds - This option resets the scan speed for all theoretical points back to the original speed.

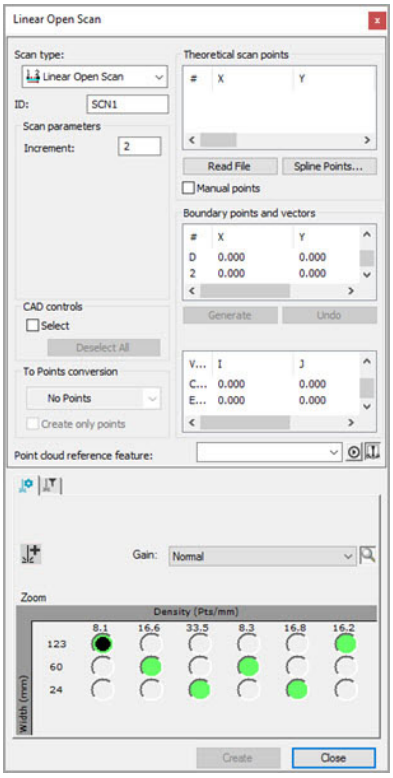
When you click the **Generate** or **Spline Points** button, PC-DMIS modifies the path points. If you assign the scan speed change to the points of the original path, PC-DMIS tries to find the closest point to the original scan speed change point within a small tolerance. If a close point is available, PC-DMIS assigns a new scan speed change point. If a close point is not available, PC-DMIS removes the scan speed change point.

After you click the **Generate** or **Spline Points** button, ensure that all the path points are defined.

The speed change points are available only on machines with the FDC controller. If the controller does not support control points, PC-DMIS executes scans without changing the scan speed.

In PC-DMIS 2020 R1 and earlier, PC-DMIS doesn't support speed change points. If you save a measurement routine to PC-DMIS 2020 R1 and earlier, PC-DMIS removes the speed change points. When you open the measurement routine in that older version, the speed change points are not available. This ensures that you execute your measurement routine at the constant scan speed.

Performing a Linear Open Advanced Scan



Linear Open Scan dialog box

The **Linear Open Scan** method scans the surface along a line. This procedure uses the starting and ending points for the line, and also includes a direction point for the calculation of the cut plane. The probe always remains within the cut plane while doing the scan.

To Create a Linear Open Scan

1. Ensure that you have a laser probe enabled.
2. Place PC-DMIS into DCC Mode.

3. Select the **Insert | Scan | Linear Open** menu item. The **Scan** dialog box opens with the **Linear Open Scan** option already selected in the **Scan type** list.

4. If you want your scan to traverse some of the surfaces between the start and end points but not all the surfaces, use the **Select** check box and select those surfaces as discussed in the "CAD Controls" topic. To access these controls, click the **Advanced >>** button in the upper-right corner of the dialog box if necessary, and then click the **Graphics** tab at the bottom.

5. If you are going to use the boundary points to help define the scan path, add the 1 point (starting point), the D point (direction to scan), and the 2 point (ending point) to the scan by following the appropriate procedure discussed in the "Boundary Points" topic.

6. If you want to scan *all* the surfaces between the start and end points, you do not need to select all of the surfaces. To have PC-DMIS automatically scan across all the surfaces:

    a. After you've selected the 1, D, and 2 points, access the **Advanced** section of the **Graphics** tab.

    b. Mark the **Select** check box. Then click **Deselect All** to deselect any selected surfaces.

    When you select the **Generate** button later in this procedure, PC-DMIS automatically generates the scan across all surfaces between the start and end points.

7. Make any changes to the vectors in the **Vectors** list. To do this, double-click on the vector, make any changes to the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.

8. Type the name of the scan in the **ID** box.

9. Select the **Measure** check box if you want to execute the scan and measure it at creation time.

10. Set the distance between the generated theoretical points in the **Increment** box.

11. Select the method for defining the scan path from the **Read File**, **Manual Hits**, **Generate**, and **Spline Points** options.

12. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.

13. If needed, make additional modifications to your scan.

14. In the **Pointcloud reference feature** box, type the ID of the pointcloud object that will receive the surface data.

15. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

> **WARNING:** Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

16. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is cleared.

Scan Parameters

The **Increment** box in the **Scan Parameters** area lets you set the increment distance between the theoretical points when you click the **Generate** button.

Vectors

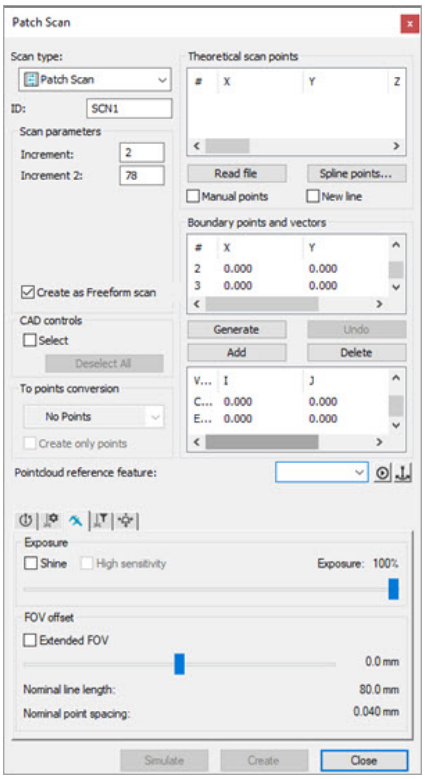
Vectors used:

- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

For additional information, see "Vectors" under "Common Functions of Scan Dialog Boxes".

> The Cut Plane vector (CutVec) is the cross product of the Initial Touch vector (InitVec) and the line between the start and end point.

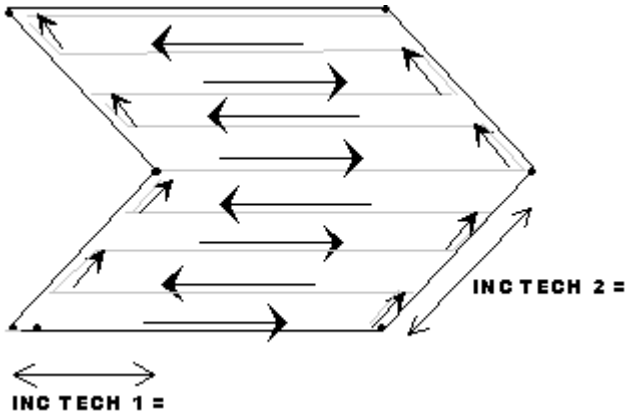
Performing a Patch Advanced Scan



Patch Scan dialog box

The Patch scan is like a series of Linear Open scans done parallel to each other.

The **Patch Scan** method scans the surface of the part based on the Scan Parameters. The probe always remains within the cut plane while doing each scan line. It uses the **Increment** value to determine the distance between points on each line. When the scan reaches the boundary at the end of a line, the scan moves to the next line by the **Increment 2** value and starts a new scan line that moves in the opposite direction. The following figure describes this process.



Patch scan increment example

To Create a Patch Scan

1. Ensure that you have a laser probe enabled.
2. Place PC-DMIS into DCC Mode.
3. Select the **Insert | Scan | Patch** menu item. The **Scan** dialog box appears with the **Patch Scan** option already selected in the **Scan type** list.
4. Set the values for **Increment** and **Increment 2**. These determine the spacing of the points if you select the **Generate** or **Spline** buttons or the **New Line** check box to define the scan. **Increment** defines the spacing between each point on a scan line and **Increment 2** defines the spacing between scan lines.
5. If your active sensor is an HPL-10.10 sensor, you may want to consider changing the scan type from a Patch scan to a Freeform scan. To do this, select the **Create as Freeform scan** check box.

   > The Freeform scan is found to be the best scan type when you use the HPL-10-10 laser sensor as it doesn't stop after each pass like the Patch scan does. Because of this, you may want to consider using the **Create as Freeform scan** check box option to convert Patch scans to Freeform scans when you use the HPL-10-10 sensor.

6. If your scan traverses multiple surfaces, consider selecting surfaces as discussed in the "CAD Controls" topic.
7. If you are going to use the boundary points to help define the scan path, add the 1 point (starting point), the D point (the direction to begin scanning), the 2 point (the end point of the first line), the 3 point (to generate a minimum area), and, if desired, the 4 point (to form a square or rectangular area). This will select an area that you wish to scan. Pick these points by following the appropriate procedure discussed in the "Boundary Points" topic.
8. Make any changes to the vectors in the **Vectors** list. To do this, double-click on the vector, make any changes to the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.
9. Type the name of the scan in the **ID** box.
10. Mark the **Measure** check box if you want to execute the scan and measure it at creation time.
11. Select the **Generate** button to generate a preview of the scan on the CAD model in the Graphic Display window. When you generate the scan, PC-DMIS starts the scan at the start point and follows the chosen direction until it reaches the boundary point.

The scan then moves back and forth scanning in rows along the chosen area and at the specified increment value until it finishes the process.

12. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
13. If needed, make additional modifications to your scan.
14. In the **Pointcloud reference feature** box, type the ID of the pointcloud object that will receive the surface data.
15. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.



WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

16. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is cleared.

Patch Scan Parameters

The **Increment** and **Increment 2** boxes described below are available when creating and measuring a **Patch** scan.

Increment

The **Increment** allows you to set the increment distance between each point when Generate or Spline/Line is used to define the scan path.

Increment 2

The **Increment 2** allows you to set the increment distance between scan lines when Generate or Spline/Line is used to define the scan path.

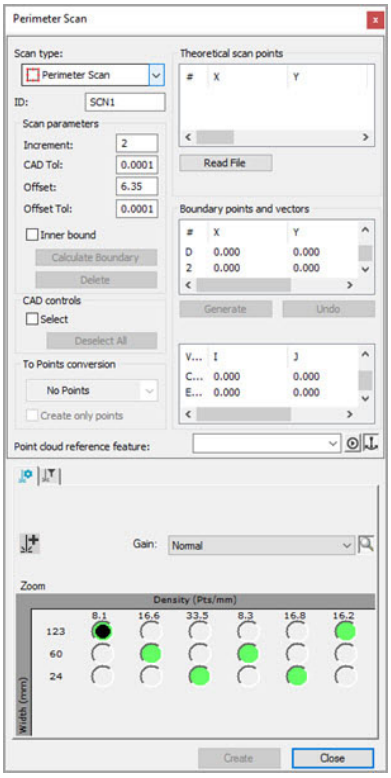
Initial Vectors

Vectors used:

- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

The cut plane vector is derived by crossing the Initial Touch vector (InitVec) 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.

Performing a Perimeter Advanced Scan



Perimeter Scan dialog box

The **Perimeter Scan** method scans the surface of the part based on the selected surfaces. This procedure traverses the selected surfaces within the created boundaries.

To Create a Perimeter Scan

To create a Perimeter scan:

1. Ensure that you have a laser probe enabled.
2. Place PC-DMIS into DCC Mode.
3. Select the **Insert | Scan | Perimeter** menu item. The **Scan** dialog box appears with **Perimeter Scan** already selected in the **Scan type** list.
4. Select the surface or surfaces to use to create the boundary. If you select multiple surfaces, you should select the surfaces in the same order that you want them to be scanned. To select the necessary surface or surfaces:

a. Verify that the **Select** check box is marked. Select each surface that you want to scan in the order that you want them scanned. PC-DMIS highlights the surfaces as you select them.

b. After you select the desired surfaces, clear the **Select** check box.

c. Click on the surface near the boundary where the scan is to begin. This is the Start Point.

d. Click on the same surface a second time in the direction that you want to execute the scan. This is the Direction Point.

e. Click the point where the scan is to end. This point is *optional*. If you do not provide an End Point, the scan ends at its Start point.

5. Type the appropriate values into the **Scan parameters** area which includes the following boxes:

   - **Increment** box
   - **CAD Tol** box
   - **Offset** box
   - **Offset Tol (+/-)** box

6. Select the **Calculate Boundary** button to calculate the boundary from which to create the scan. The red dots on the boundary indicate where the hits are taken on the perimeter scan.

   The boundary calculation should be a relatively quick process.

   If the boundary does not look correct, click the **Delete** button to delete the current boundary. You can then create another.

   If the boundary appears incorrect, it usually means that you need to increase the CAD tolerance.

   After you change the CAD tolerance, click the **Calculate Boundary** button to recalculate the boundary.

   Verify that the boundary is correct before you perform a perimeter scan because it takes much longer to calculate the scan path than it does to recalculate the boundary.

7. Verify that the **Offset** value is correct.

8. Click the **Generate** button. PC-DMIS calculates the theoretical values used to execute the scan. This process involves a very time intensive algorithm. Depending on the complexity of the selected surfaces and the number of points that are being calculated, it may take a while to compute the scan path. A five-minute wait is common. If the scan does not appear to be correct, you can use the **Undo** button to delete the proposed scan path. If needed, you can alter the offset tolerance and recalculate the scan.

9. If needed, you can delete individual points by selecting them one at a time in the **Theoretical Path** area and pressing the Delete key on your keyboard.

10. In the **Pointcloud reference feature** box, type the ID of the pointcloud object that will receive the surface data.

11. In the **Hit Type** list, you can select **Surface Point** or **Edge Point** in case you want to convert scan data into Surface Point or Edge Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

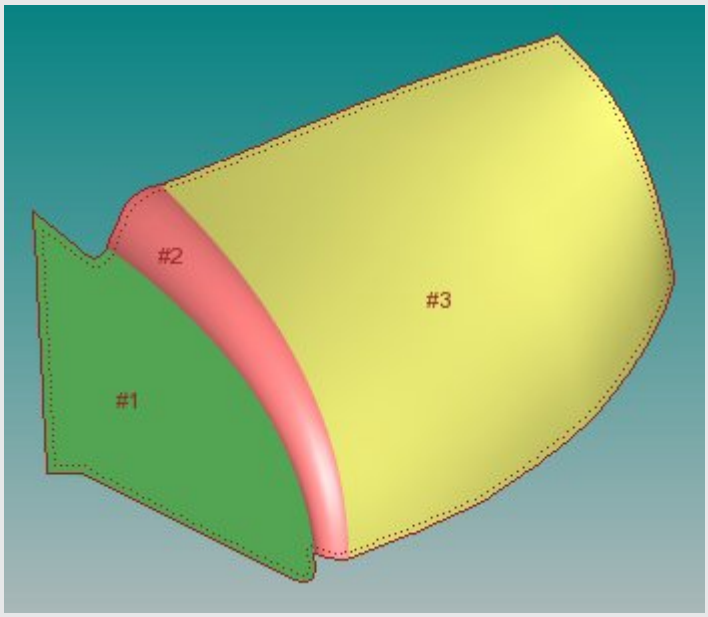
    WARNING: Be aware that if the **Measure** check box is marked, the machine begins to move as soon as you click **Create**. Make sure you are well clear of the machine to avoid injury.

12. Click the **Create** button to store the Perimeter scan in the Edit window if the **Create only points** check box is not selected. It is executed like any other scan.

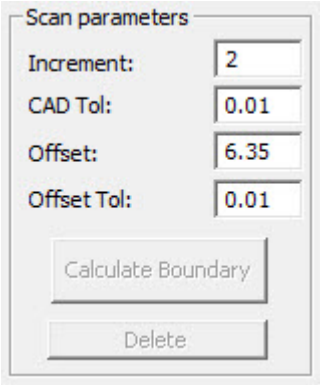
    You can click the **AutoWrist** button () from the scan's dialog box to enable the AutoWrist functionality. When you enable the AutoWrist option but you don't have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS asks you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Suppose you selected these three surfaces. Each surface borders another, but the outside of each surface makes up the composite boundary (indicated by the solid line). The offset distance is the amount that the scan is offset from the composite boundary (indicated by the dotted line)



Perimeter scan example

Perimeter Scan Parameters



Scan Parameters area

The **Scan Parameters** area of the dialog box allows various options for constructing a Perimeter Scan. These include:

Increment

The **Increment** box indicates the distance between each of the hit points on the scan.

CAD Tol

The **CAD Tol** box is useful in detecting 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 indicates the distance in from the perimeter where the scan will be created and executed.

Offset + / -

The **Offset Tol (+/-)** box indicates the amount of allowable deviation from the offset value. It is a user supplied value.

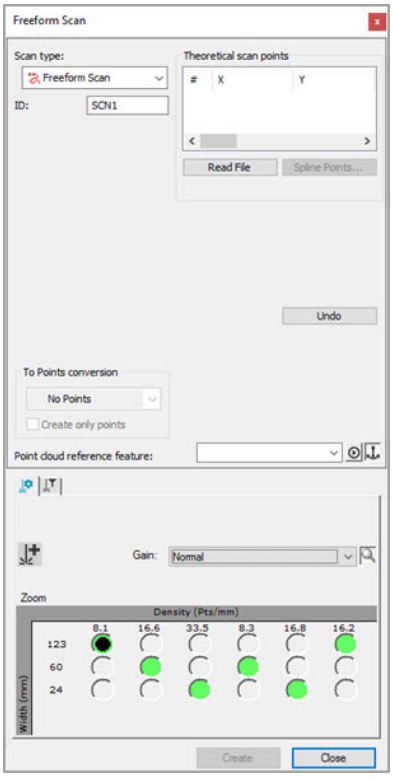
Calculate Boundary

The **Calculate Boundary** button determines the composite boundary of the input surfaces. The Calculated boundary appears as red dots in the Graphic Display window.

Delete

The **Delete** button deletes the previously created boundary.

Performing a Freeform Advanced Scan



Freeform Scan dialog box

The **Freeform Scan** method defines a scan path that is not restricted to following any particular set of rules. You can define the scan path to move in any direction, including crossing back over itself.

Creating a Freeform Scan

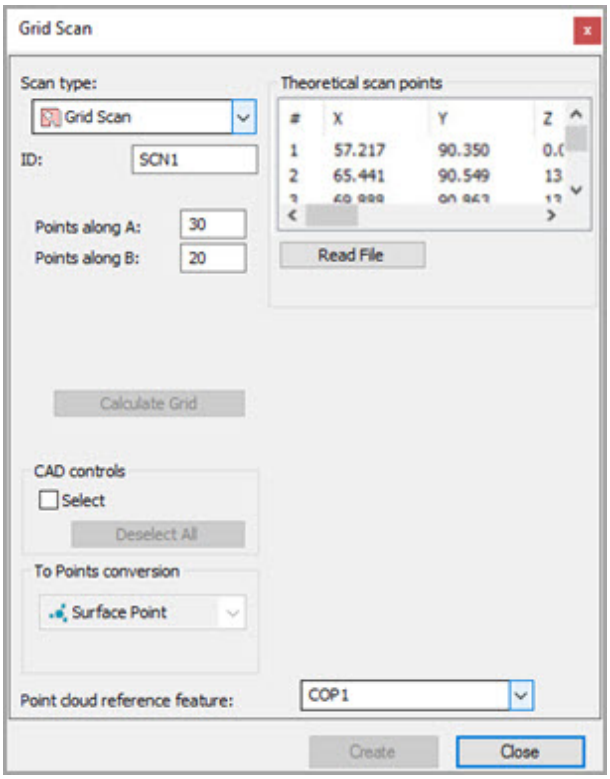
1. Place PC-DMIS into DCC Mode.
2. Select the **Insert | Scan | Freeform** menu item. The **Scan** dialog box opens with the **Freeform Scan** already selected in the **Scan type** list.
3. You need to define the scan path. You can use the **Read File** option or use the **Manual Points** method to do this.
4. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
5. After you select five or more **Theoretical Points**, use the **Spline Points** option to better define the path.
6. If needed, make additional modifications to your scan.
7. In the **Pointcloud reference feature** box, type the ID of the pointcloud object that will receive the surface data.
8. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

   WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

9. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is not selected.

   You can click the **AutoWrist** button () from the scan's dialog box to enable the AutoWrist functionality. When you enable the AutoWrist option but you don't have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS asks you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Performing a Grid Advanced Scan

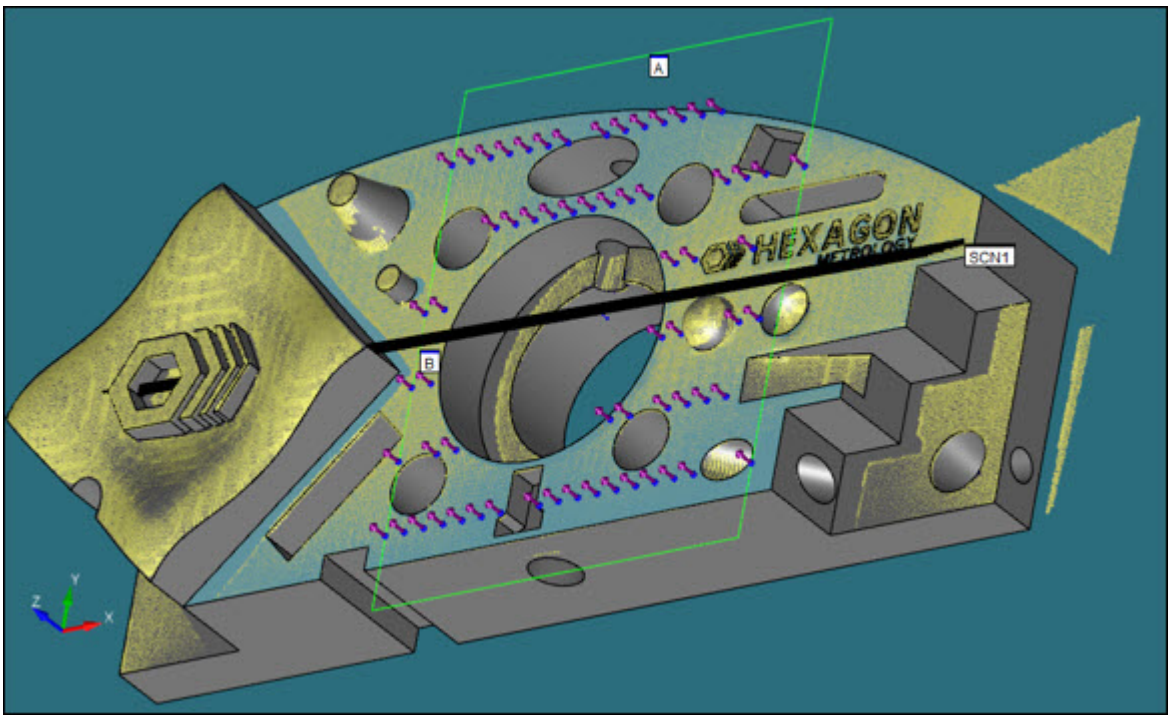


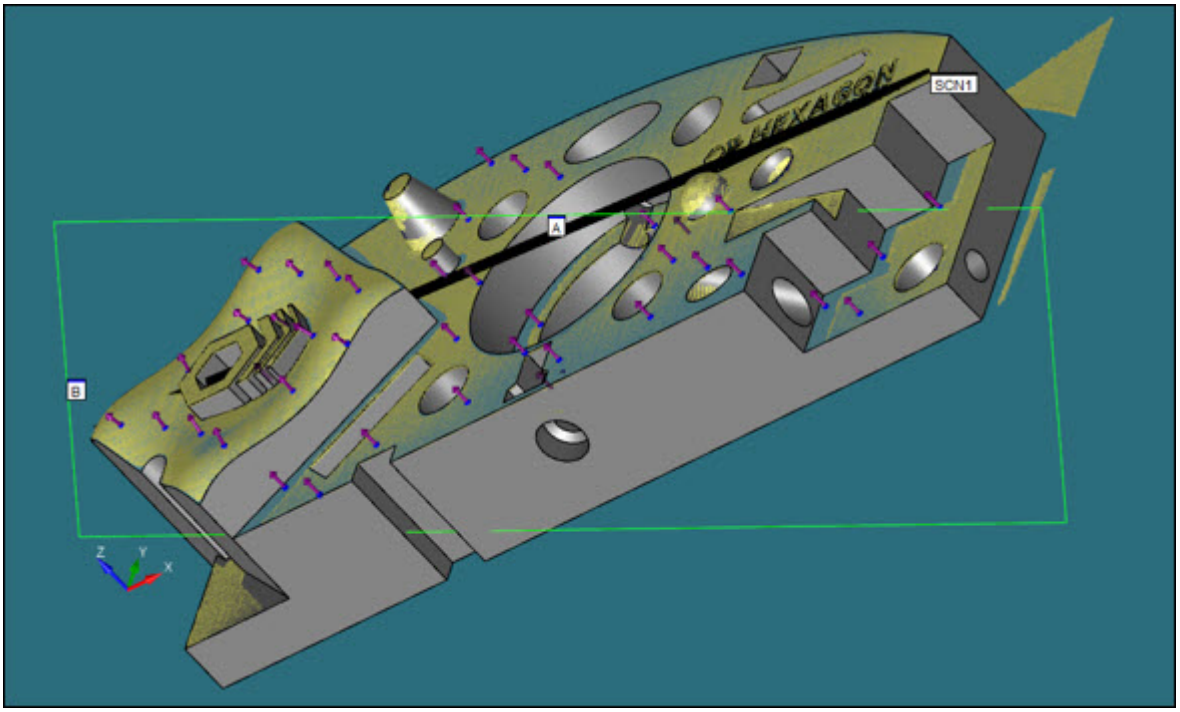
Grid Scan dialog box

The **Grid Scan** method creates a grid of points within a visible rectangle and then projects those points down on top of any selected surfaces. The rectangle and the grid of points in it, depend on the orientation of the CAD model in the **CAD** tab.

To define how many hits inside the boundary to space and drop onto the selected surface or surfaces, use the **Hits along A** and **Hits along B** boxes.

Consider the following figures that show grid Surface Points extracted from a COP:





Creating a Grid Scan

1. Ensure that you have a laser probe enabled.
2. Place your CAD model into Solid mode.
3. Place PC-DMIS into DCC Mode.

4. Select the **Insert | Scan | Grid** menu item. The **Scan** dialog box opens with the **Grid Scan** already selected in the **Scan type** list.
5. If you want to use a custom name for the grid, in the **ID** box, type the name of the grid.
6. In the **Hits along A** and **Hits along B** boxes, specify how many hits in the A and B directions to space and drop onto the selected surface or surfaces.
7. Click and drag a rectangle on the screen over the surface or surfaces you want to include in your scan. This rectangle defines the boundary of the grid. The boundary is projected onto the CAD surface or surfaces. PC-DMIS draws points on the CAD model on any surfaces that you selected when you drew the rectangle.
8. Mark the **Select** check box if you want to deselect some surfaces. PC-DMIS highlights the selected surfaces and draws points on only those surfaces. It does not draw points on any deselected surfaces, even if they are within the boundary of the rectangle.
9. If you mistakenly select a surface, press Ctrl and click on that surface a second time to deselect it. To deselect all of the highlighted surfaces at once, click the **Deselect All** button.
10. To recompute grid points at any time (that is, to apply different A and B values on the selected surfaces), click the **Calculate Grid** button.
11. In the **Pointcloud reference feature** box, type the ID of the COP object from which to extract the surface data.
12. In the **Hit Type** list, **Surface Point** is the only option available. This is because the scope of the dialog box is to convert grid data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.
13. Click the **Create** button. PC-DMIS inserts the Surface Point laser commands into the Edit window in a collapsed `Group` command.

Performing a Manual Laser Scan on DCC Machines



Manual laser scans on DCC machines only work on FDC controllers and therefore only on Bridge machines with indexable heads. Manual laser scanning functionality is not available on horizontal arms with CW43L wrists.

To create a manual laser scan on a DCC machine:

1. Start PC-DMIS online with a laser sensor.

2. From the main menu, select **File | New** to start the machine in Manual Mode.
3. Press the **Probe Enable** button on the jog box (you only need to press the button once, regardless of the button's state). The sensor initializes and the **Laser** tab appears in the Graphic Display window. The software automatically creates a COP command.



If the **Probe Toolbox** is open, you can still change the sensor **Zoom** settings as needed.

4. Use the **Laser** tab and position the probe over the part in range as needed.
5. On the jog box, change the **Probe Enable** option to the "Enable" state. If not, it does not collect data.
6. On the jog box, press the **Record** button to start scanning. The **Laser** tab closes immediately, and the scanned data populates the COP object and the Graphic Display window in real time.
7. Use the jog box and move the probe over the part to scan it until you are satisfied with the data coverage.
8. To stop scanning, press the **Record** button again.
9. If necessary, press the **Probe Enable** button again to scan more data. PC-DMIS prompts if you want to empty the existing COP command or if you want to add the new data to it.
10. Repeat from Step 6 above to continue scanning.

You can also create a manual scan on a DCC machine by:

1. Follow steps 1-4 above.
2. Use the jog box, change the **Probe Enable** button to the "Disabled" state.
3. On the jog box, press the **Record** button.
4. On the jog box, use the **Probe Enable** button to toggle data collection "On" and "Off".
5. On the jog box, press the **Record** button a second time to stop scanning and finalize the COP data.

Setting the Machine Speed for Scanning

To properly define your machine's speed for scanning with your laser, you need to do the following:

- Your controller must support VHSS. PC-DMIS uses this high speed mode by default when the CMM supports it.
- The `ScanSpeed` entry, found in the **Leitz** section of the PC-DMIS Settings Editor, limits the maximum scan speed value that you can send to the controller. By default, this is set to 50 mm/sec. Any value set by a SCANSPEED/ Edit window command is limited to the value of the `ScanSpeed` entry. This value can be increased according to the CMM limits.
- By default, PC-DMIS's **Acceleration** value, located in the **Opt. Probe** tab of the **Parameter Settings** dialog box, is set very low (10 mm/sec). To get higher scan speeds, you can increase this value to a desired value up to the limits allowed by your machine. To access this tab, select the **Edit | Preferences | Parameters** menu item and then click on the **| Opt. Probe** tab.

Simulate Scanning by Importing a Pointcloud

You can simulate the scan of an imported pointcloud with the **Simulate Scan** button  on the **Pointcloud** toolbar (**View | Toolbars | Pointcloud**). The button has two states, Play and Stop (default).

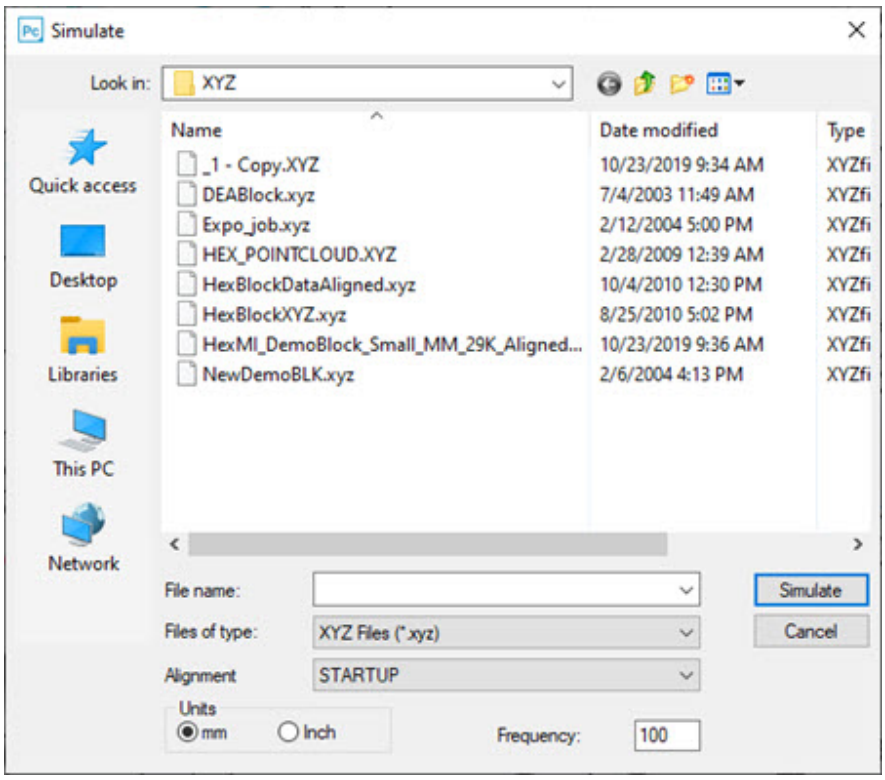
The simulation imports the scan passes in the order they were originally scanned. You can import XYZ and PSL files to simulate scans.

This allows you to do the following:

- You can visualize how the pointcloud was originally scanned.
- You can see the result of real-time filters because, during the simulation import, the software applies any active settings on the **Laser Data Collection Settings** dialog box to the imported pointcloud data. For details on the **Laser Data Collection Settings** dialog box, see the "Laser Data Collection Settings" topic in this documentation.

To simulate a scan:

1. Click the **Simulate Scan** button to open the **Simulate** dialog box.



2. Use the **Simulate** dialog box to navigate to the Pointcloud file that you want to import. Use the **Files of type** list to filter the file types to display.

   The pointcloud file must contain stripe information.

   - If the file you select does not contain stripe information, as soon as you click the **Simulate** button, PC-DMIS displays an error message that says PC-DMIS cannot simulate the file because the file doesn't have the stripe information. It then aborts the import process.
   - If the file contains stripe information, the import continues. PC-DMIS imports the pointcloud data into the first Pointcloud operator it finds below the current cursor position in the Edit window.

   If the existing Pointcloud operator already contains data, the new data is added to the existing data.

   If a Pointcloud operator does not exist, PC-DMIS creates a new Pointcloud operator.

3. From the **Simulate** dialog box, enter a value in the **Frequency** box to set the speed of the simulation during the import process. The default is 100 and the range is 1 to 500 inclusive from slowest to fastest speed.

4. Click the **Simulate Pointcloud** button  . PC-DMIS simulates the scan as it imports the pointcloud data and the button's icon changes to the **Simulate**

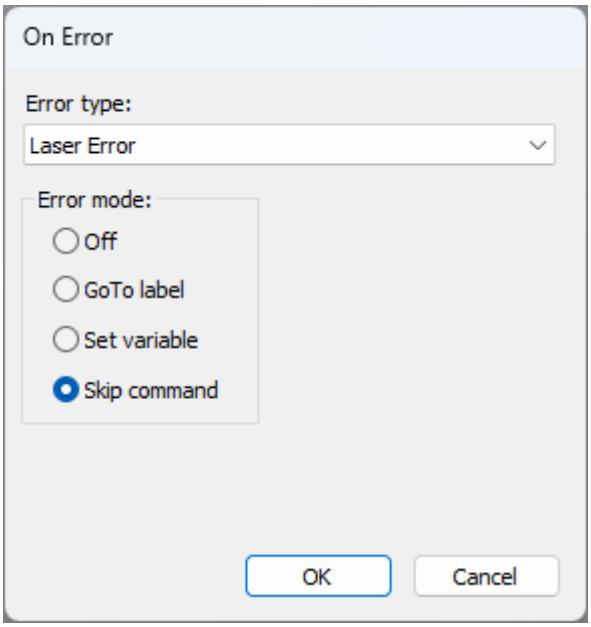
Pointcloud - Stop icon  . Click the button again to stop the simulation and change the button's icon to its original state.

Related Topics:

Scanning Your Part Using a Laser Sensor

Handling Laser Sensor Errors with On Error Command

The **Insert | Flow Control Command | On Error** option opens the **On Error** dialog box:



On Error Dialog Box

You can tell PC-DMIS to skip commands that generate certain laser sensor-related errors during execution by using the `OnError` command. The command only applies to the default Asynchronous Execution mode.

The information in this topic is specific to laser configurations. For more information on this dialog box and how it applies to tactile probes, see "Branching on an Error" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation.

Important Note for Robotic Automation and Execution Manager Users

In most cases, the `OnError` command will report failed features in the OnError/Laser output file, but it will not block the execution of the PC-DMIS measurement routine. However, if a Construction or Alignment command is dependent on a failed feature, PC-DMIS will block the execution for safety reasons.

When you execute a PC-DMIS measurement routine using Robotic Automation and Execution Manager, it is executed exclusively in Offline mode. You can configure PC-DMIS to skip errors that typically show a message that pauses the execution of the

measurement routine. This avoids blocking the inspection line due to the detection of defective parts and allows them to be "passed" from a PC-DMIS execution viewpoint.



Those parts will need to be checked and eventually rejected offline. It is the user's responsibility to define how to handle these "passed" parts in their production process.

In order to enable that option, it is necessary that the following conditions are met:

- The PC-DMIS license must include the "Robotic Automation" option.
- The selected Environment must be "Robotic Automation".
- In the Robotic Automation Environment, the **Machine Type** must be "Robotic Automation".
- In the Robotic Automation Environment, the **Machine Controller Type** must be "Offline".

Once the configuration is correctly defined with the appropriate license, environment, and machine settings, when a Laser feature that is used in a construction or an alignment fails, the execution of the PC-DMIS measurement routine will continue.

For example, the following snippet from an OnError/Laser output file shows these reported errors:

The following features failed to execute:

CIR3 : No laser stripes found for feature extraction.

CIR3 : causing error in Alignment (Translate Alignment)

Please note that:

- If a Construction or an Alignment fails, some or all the subsequent measurements may fail without notice during the execution cycle.
- Some Laser features may not even show as "failed" in the OnError/Laser output file, but their measurement results can still be wrong in the report due to the skipping of a local alignment for example.

Error type - PC-DMIS Laser tracks these error conditions:

- Laser error
- Temperature out of threshold - 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 upper threshold limit or below the lower threshold limit that the Temperature Compensation command defines. For

more information, see "Branching on an Error" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation.



The On Error command must be placed above the Temperature Compensation command in the measurement routine.

Error mode - PC-DMIS can take these possible actions depending on the error type:

- **Off** - If PC-DMIS encounters an error in this mode, execution stops completely, instead of skipping the command.
- **GoTo label** - The measurement routine flow moves to a defined label (see "Using Labels" in the "Branching by Using Flow Control" chapter in the PC-DMIS Core documentation). These options become available:
  - **Label ID** - In this box, 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 execution continues, and PC-DMIS skips commands if they generate any of the following errors:
  - No laser stripes found for feature execution
  - No scan data
  - Feature calculation error

The command has the following syntax in the Edit window in Command mode:

```
ONERROR/LASER_ERROR,TOG1
```

`TOG1` = This defines the **Error mode** setting, **Off**, **GoTo label**, **Set variable**, or **Skip command**.



You can control how PC-DMIS handles error messages when you perform feature extractions. This also prevents the display of error messages when you execute measurement routines under the Robotic Automation environment. For details, see the "LaserErrorsOutputToFile" topic in the PC-DMIS Settings Editor documentation.

Using the Mesh Commands

All mesh commands are available from the **Mesh** toolbar (**View | Toolbars | Mesh**).

The mesh commands are:

- **Mesh** - This option opens the **Mesh Command** dialog box to create a mesh feature from any number of pointclouds. You do not need to have any COPs defined to create a Mesh. If no COPs are defined, an empty Mesh object is created in the Edit window.

  This option is available from the main menu (**Insert | Mesh | Feature**). It is also accessed by clicking the **Mesh** button ( ) on the **Pointcloud**, **QuickCloud**, or **Mesh** toolbar. When you select the option or button, the **Mesh Command** dialog box appears.

  For details, see the "Creating a Mesh Feature" topic.

- **Mesh Operator** - This option is available from the main menu (**Insert | Mesh | Operator**) or from the **Mesh** toolbar. This opens the **Mesh Operator** dialog box that you can use to create a Mesh operator.

  For details, see the topic "Creating a Mesh Operator".

  The operators are:

  - Mesh CROSS SECTION Operator
  - Mesh EXPORT Operator
  - Mesh IMPORT Operator
  - Mesh COLORMAP Operator
  - Mesh EMPTY Operator
  - Mesh SELECT Operator

- **Import Mesh in STL Format** - This button opens the **Mesh Import Data** dialog box that you can use to import an STL mesh data file. If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created, and the software imports the STL data. If a Mesh object already exists in the PC-DMIS Edit window, then the software adds the STL data to the Mesh object.

  For details, see the "Mesh IMPORT Operator" topic.

  This option is available from the main menu (**File | Import | Mesh**). You can also access this option from the **Mesh** toolbar.

  For details, see the topic "Import Mesh in STL Format".

-  **Export Mesh in STL Format** - This button opens the **Export Mesh Data** dialog box that you can use to export a Mesh in an STL, ASCII, or STL Bin file format.

  For details, see the "Mesh EXPORT Operator" topic.

  This option is available from the main menu (**File | Export | Mesh**). You can also access this option from the **Mesh** toolbar.

  For details, see the "Export Mesh in STL Format" topic.

-  **Empty a Mesh** - This option empties a Mesh object. To use this function, position the cursor in the Edit window directly ON the Mesh object that you want to empty, then click the button. If your cursor is not on a Mesh object, the Mesh object immediately above your cursor position is emptied.

  For details on the Empty a Mesh command, see the topic "Empty a Mesh".

  >  Note that this is different from inserting the Empty command operator. In this case, the empty command is placed above the Mesh object to empty. For details on the Empty command operator, see the "Mesh EMPTY Operator" topic.

-  **Select Mesh** - Use this button to select and delete a subset of triangles contained in the mesh data object. When you use this button, the selection method uses a polygon to remove triangles in the 3D view.

  For details, see the "Mesh SELECT Operator" topic.

-  **Colormap a Mesh** - This button opens the **Mesh Operator** dialog box that you can use to create a Mesh COLORMAP operator.

  For details, see the "Mesh COLORMAP Operator" topic.

-  **Mesh Alignment** - This option displays the **Mesh/CAD Alignment** dialog box. Use the dialog box to align the mesh to a CAD model.

  For details, see the "Mesh Alignment" topic.

-  **Receive a Mesh from OptoCat** - When you click this option to ON, PC-DMIS is placed in a state where it is waiting and ready to receive a mesh from the OptoCat application.

  For details, see the "Receive a Mesh from OptoCat" topic.

Creating a Mesh Feature

You can create a **Grid Mesh** or a **3D Mesh** from the **Create Mesh** dialog box.

You must have the Mesh license enabled to use or view this option.

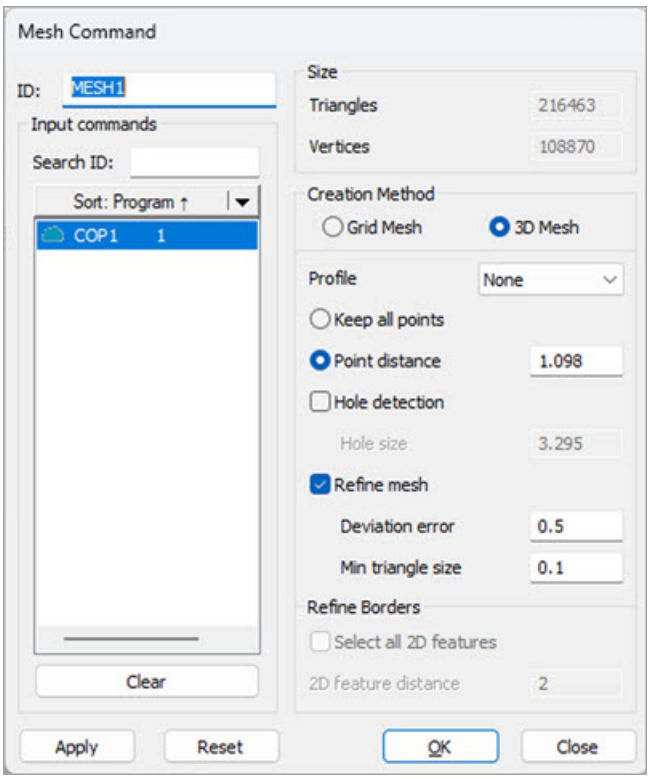
Follow one of these procedures to create the appropriate Mesh feature:

The **3D Mesh** is the preferred method and typically results in a better mesh. The design of the **Grid Mesh** method is for you to use it when the scan of the original COP used the Pointcloud **Mesh Display** option. When you use the **Grid Mesh** method with a COP, and you did not scan using the **Mesh Display** option, the result may be an incomplete or poorly-defined mesh. Creating a mesh is a time-consuming operation.

For details on the Pointcloud Display options, see "Pointcloud Display Area" in this documentation.

Creating a 3D Mesh Feature

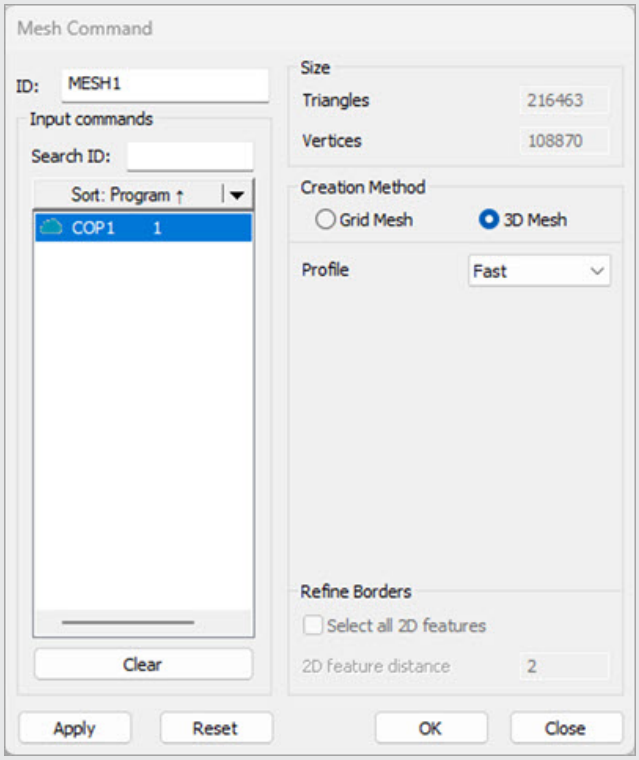
1. Select **Insert | Mesh | Feature** from the main menu to display the **Mesh Command** dialog box. You can also access this option from the **Mesh** button () on the **Mesh** toolbar (**View | Toolbars | Mesh**). PC-DMIS displays the **Mesh Command** dialog box.



Mesh Command dialog box - 3D Mesh Creation Method

2. Select the features and pointclouds to be meshed together from the list.
3. Select the **3D Mesh** option from the **Creation Method** section.
4. Update the options in the **Mesh parameters** section as needed:
    - **Profile** - This option allows you to select from a list of pre-configured meshing settings:
        - **None** - This is the default option that allows you to manually set all the meshing properties.
        - **Fast** - This pre-configured meshing option allows for the fastest mesh generation.
        - **Normal** - This pre-configured meshing option is slower and more detailed than the **Fast** option.
        - **Detailed** - This pre-configured meshing option is slower but more detailed than the **Normal** meshing option.

When you select **Fast**, **Normal**, or **Detailed** option from the **Profile** list, PC-DMIS hides all the manual settings in this area of the dialog box.



Mesh Command dialog box showing the Fast profile option selected.

Profile options described as being "more detailed" mean that PC-DMIS generates a larger number of triangles around holes and high curvature areas of your CAD model than the previous profile option.

- **Keep all points** - When you select this option, PC-DMIS uses all the points in the pointcloud to create the mesh.

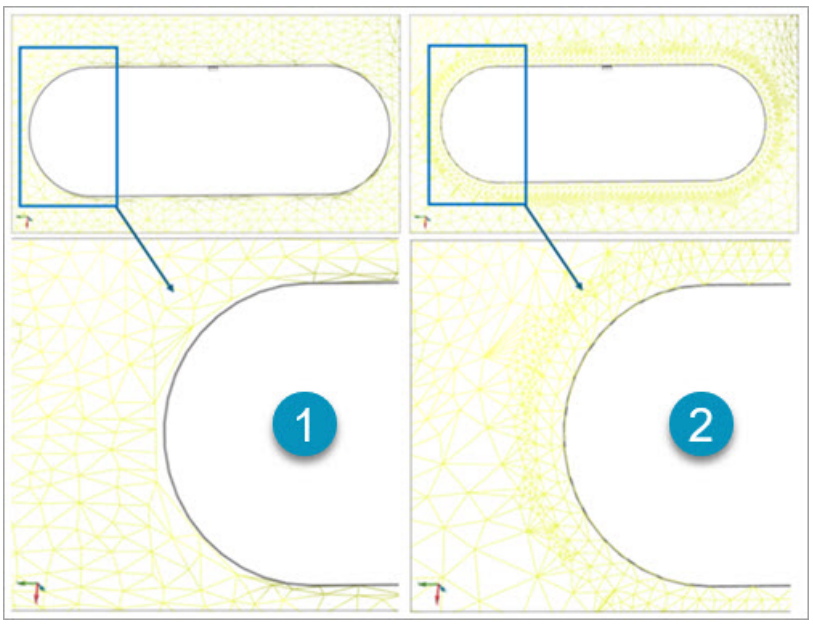
  When you select the **Keep all points** option, PC-DMIS requires more processing time to mesh the pointcloud.

- **Point distance** - This value defines the minimum distance between neighboring points that the software uses to create the vertices of each triangle in the mesh.

  The **Point distance** option is the default and preferred setting. When you select this option, PC-DMIS projects a "grid" of this

size onto the mesh and takes ONLY the best points in each grid element.

- **Hole detection** check box - When you select this check box, PC-DMIS determines when to exclude points based on the **Edge Size** value.
    - **Hole size** - This setting defines the minimum hole or gap size during a scan. For details on this option, see the "Data Filtering Area" topic in this documentation.
- **Refine mesh** check box - When this check box is selected, the following parameters refine the mesh being created:
    - **Deviation error** - The value entered determines how far points can deviate from the mesh construction and still be included in the mesh.
    - **Min triangle size** - The value entered determines the minimum size a triangle can be based on the points being evaluated.
- **Refine Borders** area - If previously measured or extracted 2D features are available (Circles, Polygons, Round Slots, or Square Slots), PC-DMIS can refine their mesh borders. Once you select an existing COP or Mesh from the feature list, all valid 2D features become available for you to select. You can then select one or more of these additional features to further refine the mesh borders.
    - **Select all 2D features** check box - Select this check box to automatically select all available 2D features.
    - **2D Feature Distance** - This value helps determine which vertices are included when refining the mesh borders. For most cases, the default value of 2 is sufficient. In the rare cases where borders are not sufficiently refined, increasing this value usually resolves the problem.



Example showing:

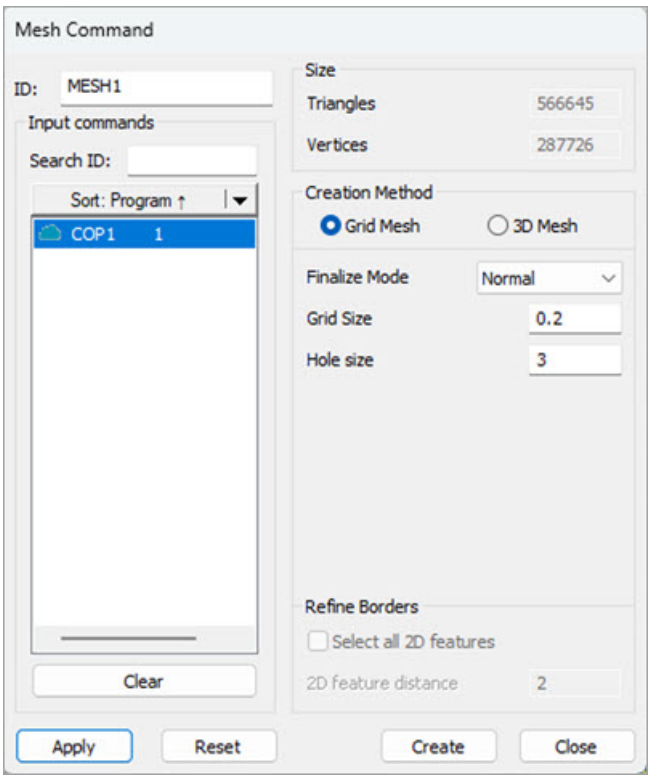
***1** - The original mesh with gaps between the mesh border and the slot geometry.*

***2** - The result after using the 2D slot geometry to refine the mesh border.*

5. Click **Apply** to apply any changes made in the **Mesh Command** dialog box. Click **Create** to generate the new Mesh command.

Creating a Grid Mesh Feature

1. Select **Insert | Mesh | Feature** from the main menu to display the **Mesh Command** dialog box. You can also access this option from the **Mesh** button () on the **Mesh** toolbar (**View | Toolbars | Mesh**). PC-DMIS displays the **Mesh Command** dialog box.



Mesh Command dialog box - Grid Mesh Creation Method

The **Size** section details the number of triangles and vertices defined in your Mesh feature.

The **Creation Method** section allows you to select between the **Grid Mesh** and **3D Mesh** creation methods.

2. Select the features and pointclouds to be meshed together from the list.
3. Select the **Grid Mesh** option from the **Creation Method** section.
4. Select the **Finalize Mode** option from the list. The option you select defines the amount PC-DMIS reduces and smooths the mesh display. The available options are:

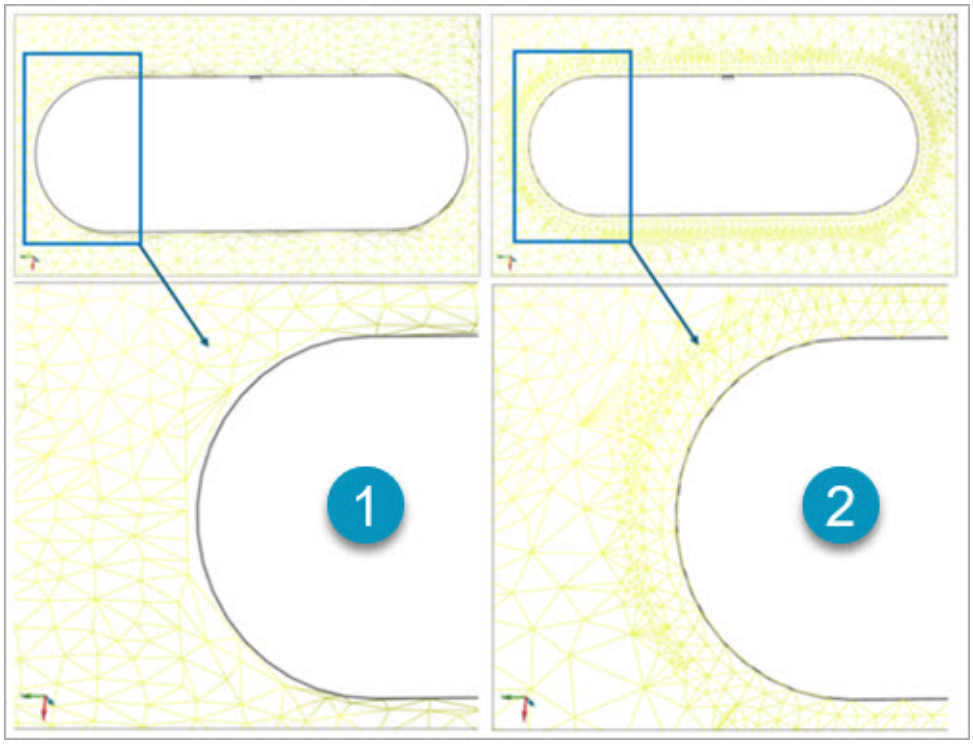
    - **Precise** (least amount of smoothing)
    - **Normal**
    - **Smooth** (most amount of smoothing).

    For details on this option, see the "Pointcloud Display Area" topic in this documentation.
5. Enter the **Grid Size** value. This defines the size of each triangle in the mesh display grid. For details on this option, see the "Pointcloud Display Area" topic in this documentation.

6. Enter the **Hole Size** value. This setting defines the minimum hole or gap size during a scan. For details on this option, see the "Data Filtering Area" topic in this documentation.

7. **Refine Borders** area - If previously measured or extracted 2D features are available (Circles, Polygons, Round Slots, or Square Slots), PC-DMIS can refine their mesh borders. Once you select an existing COP or Mesh from the feature list, all valid 2D features become available for you to select. You can then select one or more of these additional features to further refine the mesh borders.

    - **Select all 2D features** check box - Select this check box to automatically select all available 2D features.
    - **2D Feature Distance** - This value helps determine which vertices are included when refining the mesh borders. For most cases, the default value of 2 is sufficient. In the rare cases where borders are not sufficiently refined, increasing this value usually resolves the problem.



Example showing:

***1** - The original mesh with gaps between the mesh border and the slot geometry.*

***2** - The result after using the 2D slot geometry to refine the mesh border.*

8. Click **Apply** to apply any changes made in the **Mesh Command** dialog box. Click **Create** to generate the new Grid Mesh command.

When done, you can do any of the following:

- Click **Reset** to remove the created mesh from the Edit window and Graphic Display window.
- Click **Close** to close the dialog box and cancel the mesh operation if you did not click **Create** to create the mesh feature.

Creating a Mesh Operator

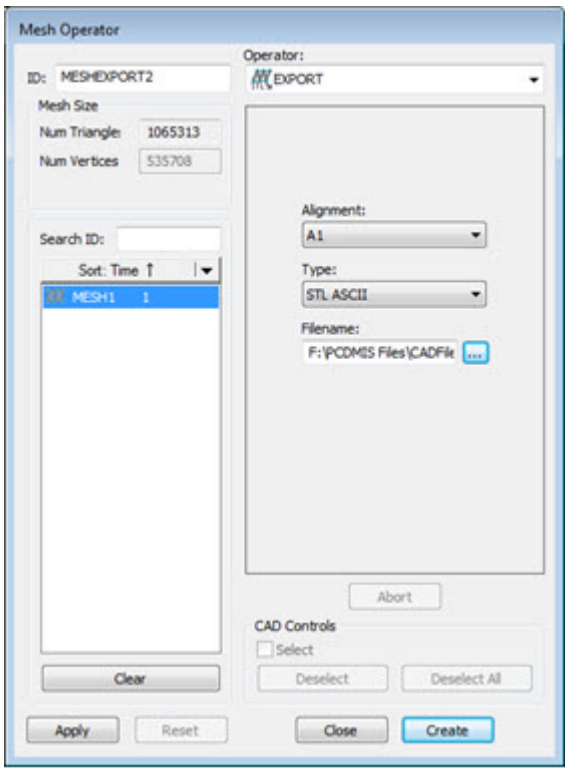
The Mesh operator commands listed below perform different operations on a Mesh object. The units for these commands are defined by the measurement routine.

The Mesh license must be enabled to use or view this option.

To create a Mesh operator:

1. Click the **Mesh Operator** button () from the **Mesh** toolbar (**View | Toolbars | Mesh**) to access the **Mesh Operator** dialog box. The dialog box is also accessible from the menu (**Insert | Mesh | Operator**).

   PC-DMIS disables the **Mesh Operator** button if no Mesh object exists. Click the **Mesh** button () to create an empty Mesh object.



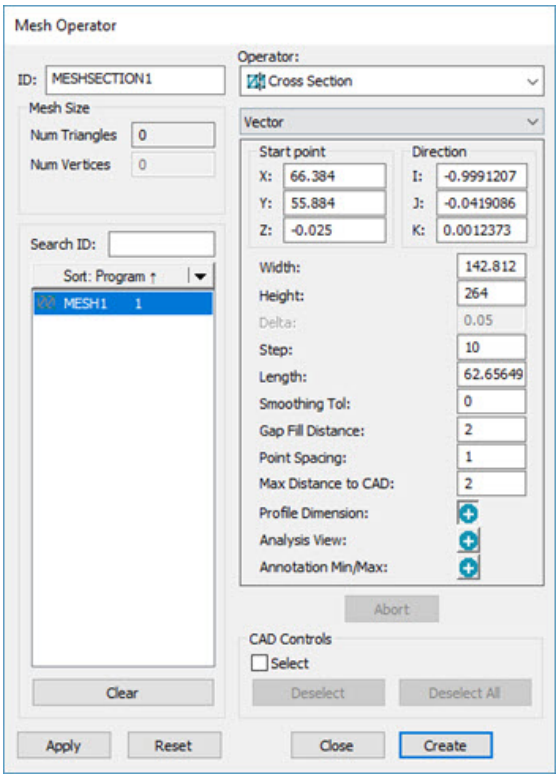
Mesh Operator dialog box

2. Select the type of operator to create from the **Operator** list.
3. Select the mesh from the **Feature List** box.
4. Select the options to use. The available options depend on the type of operator that you selected.
5. Click **Create**. PC-DMIS inserts the command into the Edit window. For example, the `EXPORT` operator command is `MESH/OPER,EXPORT` which you can see in the code below:

```
MESHEXPORT1=MESH/OPER,EXPORT,FORMAT=STL ASCII,FILENAME=F:
\TRAINING\TEST1_STL.STL,

    REF,MESH1,,
```

Mesh CROSS SECTION Operator



Mesh Operator dialog box - Cross Section operator

The Mesh CROSS SECTION operation generates a subset of polylines determined by the defined intersection of a set of parallel planes with the Mesh object. The software defines the set of planes by the start point, direction vector, step distance between the planes, and length. The software determines the number of planes by the **Step** distance divided into the **Length** plus one.

You can evaluate the Mesh CROSS SECTION operator by the profile dimension.

To apply the CROSS SECTION operation to a Mesh:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Cross Section a Mesh** button ( ) to open the **Mesh Operator** dialog box. You can also click the **Insert | Mesh | Operator** menu option.
2. From the **Mesh Operator** dialog box, select **Cross Section** from the **Operator** list.

From the **Mesh** toolbar, click the **2D Section Slide Show** button ( ) to display cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

The drop-down list under the **Operator** list contains these options: **Vector**, **Axis**, **Curve**, and **2 Points**. For details on how the **Curve** function works, see the "Creating a Cross Section along a Curve" topic. For details on the **2 Points** option, see the "Creating a Cross Section between 2 Points" topic.

The Mesh CROSS SECTION operator uses the following options:

- **Start point**: This option indicates the coordinates of a point that belongs to the first plane cutting the mesh.

  The software displays the start point as a blue ball in the Graphic Display window. You can use the ball as a handle to drag to a new location. You define the start point by your first click in the Graphic Display window. In the actual Edit window command, the start point value is held in the START PT parameter.

- **Direction** (applies only to the **Vector** and **2 Points** option): This value indicates the direction of the normal vector. You define the direction by your second click in the Graphic Display window. In the actual Edit window command, the **Direction** value is held in the NORMAL parameter.

- **Axis** (applies only to the **Axis** option): Use this option to create a cross section along the X, Y, or Z axis. Select the desired axis (the default is X), set a start point in the Graphic Display window, and set an end point. The section plane cuts the part at a given step value over the length of the cross section.

- **Width**: This value indicates the width of the section under consideration. If the value is 0, the system calculates the value as the CAD bounding box value.

- **Height**: This value indicates the height of the section under consideration. If the value is 0, the system calculates the value as the CAD bounding box value.

- **Delta**: The software does not use this value for Mesh cross sections.

- **Step**: This value indicates the distance between the planes. In the actual Edit window command, the step value is held in the INCREMENT parameter.

  If the value in the **Step** box is greater than the value in the **Length** box, then only one section cut is created at the start point.

- **Length**: This value indicates the maximum distance between the first and last plane. The software displays the length value in the **Length** parameter of the dialog box. PC-DMIS displays it as a purple line in the Graphic Display window.

- **Smoothing Tol**: You can use the **Smoothing Tol** to remove small steps in the cross section and create a smoother measured polyline. This setting filters out the points within the smoothing tolerance value and then fits a polyline to the data using the

Point Spacing value. A value of 0 (zero) turns off smoothing. This is the default value.



You can also define **Point Spacing** with the `CrossSectionCopCadCrossSectionStep` entry. For details on this entry, see "`CrossSectionCopCadCrossSectionStep`" in the PC-DMIS Settings Editor documentation.



You should set the **Smoothing Tol** very small so that the measured cross section does not deviate greatly from the actual data. Except for extreme situations (for example, a very large CAD model and/or a very poor density of points), you should set this parameter between a few tenths of a mm (maximum) and a few thousandths of a mm (minimum).

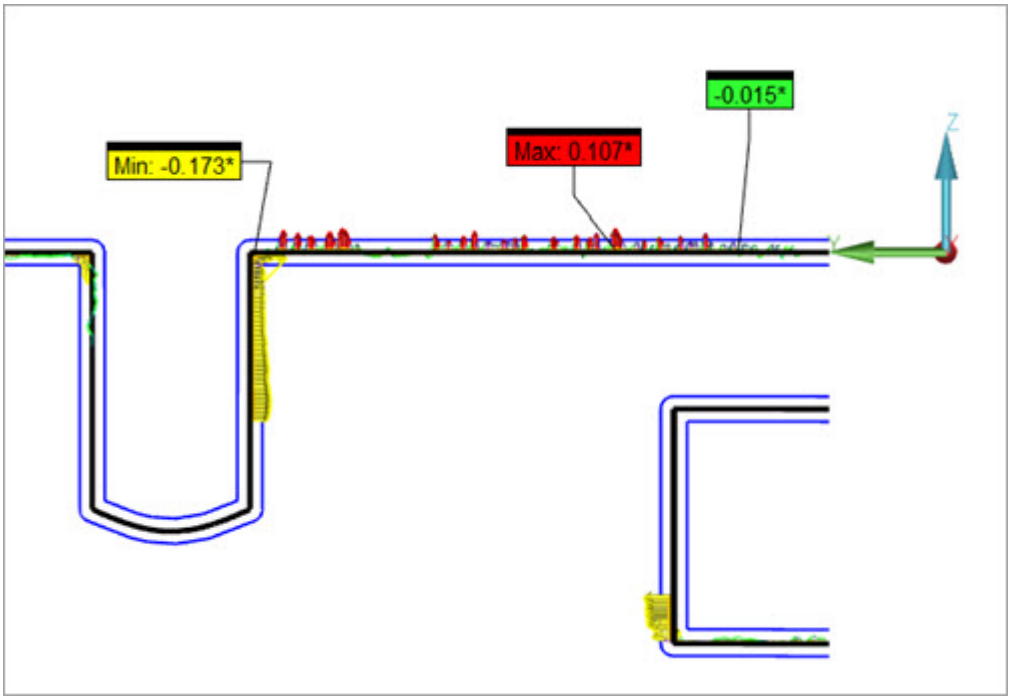
- **Gap Fill Distance**: This value defines the maximum gap distance along the yellow measured polylines of a cross section. If gaps equal to or smaller than this value appear, PC-DMIS fills in the gaps with calculated points. You can also set this value in the PC-DMIS Settings Editor. For details, see the "`CrossSectionMaximumEmptyLength`" topic in the PC-DMIS Settings Editor documentation.
- **Point Spacing**: Use this entry only when the `CrossSectionCopCadCrossSectionDrivenByCad` entry is set to 1 (True). This value is the step used along the CAD polylines to look for the best interpolated Mesh point. For greater accuracy, or if the CAD model is very small, you can set this value to a smaller value.



You can also set this value in the PC-DMIS Settings Editor. For details, see the "`CrossSectionCopCadCrossSectionStep`" topic in the PC-DMIS Settings Editor documentation.

- **Max Distance to CAD**: The value defines the maximum distance of the Mesh data relative to the nominal CAD model. The default value is 2 mm. If the Mesh data object deviates more than the **Max Distance to CAD** value from the CAD model, the software may not compute the yellow measured cross section. You can adjust this value to account for large deviations of the Mesh data relative to the CAD model.
- **Profile Dimension**: Click the **Add** button  to create a new profile dimension for each cross section. For details on Profile Dimension, see "Dimensioning Profile - Line or Surface" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

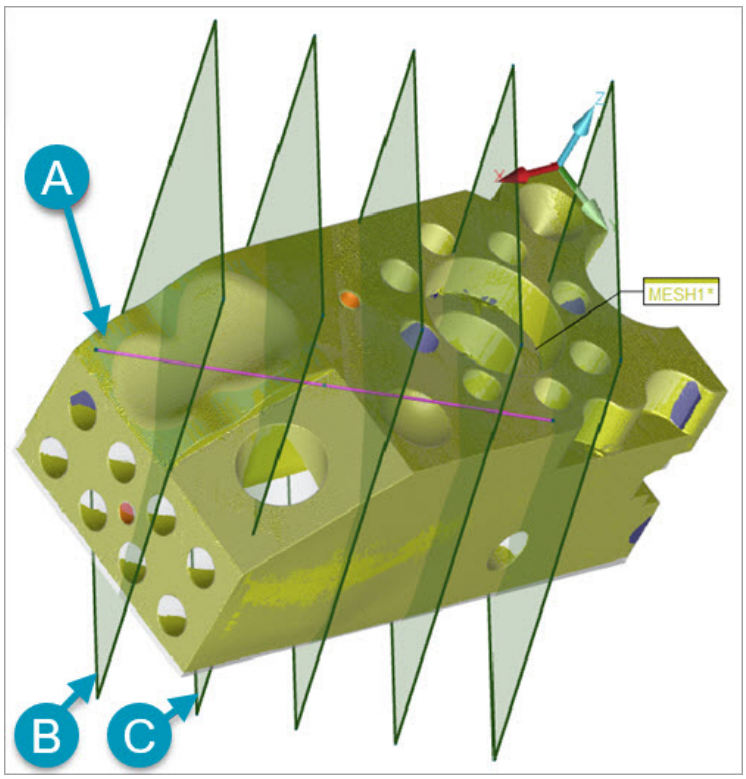
- **Analysis View**: Click the **Add** button to create the `ANALYSISVIEW` command in the Edit window. For details on the `ANALYSISVIEW` command, see "Create Analysis View Command" in the "Inserting Report Commands" chapter in the PC-DMIS Core documentation.
- **Annotation Min/Max**: Click the **Add** button to create the minimum and maximum values in the form of annotation labels for the active cross section.



  PC-DMIS calculates the minimum and maximum points each time you execute the measurement routine.

- **CAD Controls**: Mark the **Select** check box to select CAD surfaces in the Graphic Display window. PC-DMIS filters out any cross sections that do not pass through the selected surfaces when you click **Create**.

  For example, if you selected surface A after the start and end points are defined, PC-DMIS only generates cross sections at B and C:



Example of a selected surface (A) limiting the cross sections to only (B) and (C)

Selected surfaces do not affect what you see when you click the **View** button.

When the cutting planes are visible in the Graphic Display window, you can manipulate them as follows:

- To resize the height and width of the cutting planes, drag a plane's edge handle to a new position.
- To rotate the set of planes around their axis, drag a plane's corner handle to a new position.
- To redefine the purple line's `START` or `END` definition, drag the first or last purple length line's blue point handle to a new position. While the direction is changing, the values in the dialog box and the number of planes in the Graphic Display window are updated. In Axis mode, the direction of the planes does not change.
- To move the set of planes, drag the purple length line's middle blue point handle.

> When a cross section is created or edited, the cutting planes appear in a transparent view as shown above.

Click **Create** to:

- Insert a `MESH/OPER,CROSS SECTION` command for each plane into the Edit window as shown in the code below.

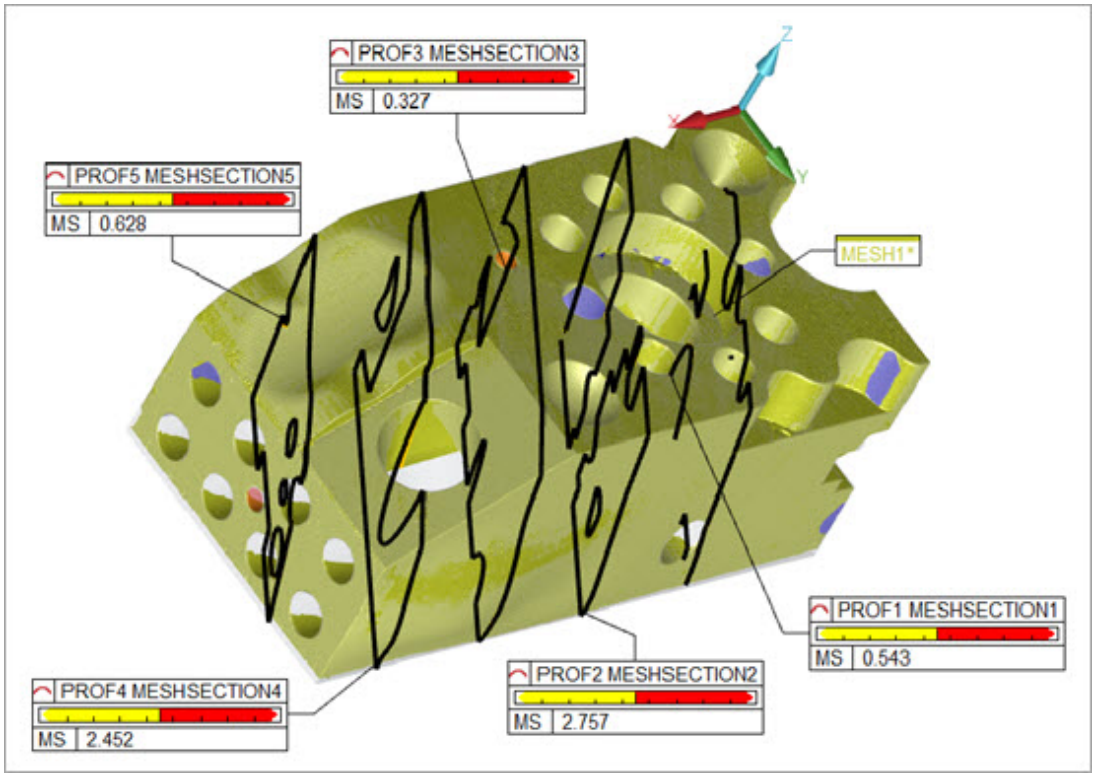
```
 MESHSECTION3=MESH/OPER,Cross
Section,TOLERANCE=0.05,WIDTH=117.715,HEIGHT=227.086,

     START PT = -6.439,60.097,6.276,NORMAL =
    0.9684394,-0.2221293,-0.1130655,SIZE=76

     REF,MESH1,,
```

The black polylines represent the nominal CAD, the yellow polylines represent the measured polyline.

- Insert a label for each plane into the Graphic Display window as shown below:



Finished cross sections showing five planes

Defining the Cross Section by Typing Values

Use the **Mesh Operator** dialog box to enter any of these values:

- **START PT**: This value specifies the cross section's starting point using the **Start Point X**, **Y** and **Z** boxes.
- **NORMAL**: This value specifies the cross section's vector using the **Direction I**, **J** and **K** boxes.
- **WIDTH**: This specifies the value of the cross section's width property in the **Width** box.
- **HEIGHT**: This specifies the value of the cross section's height property in the **Height** box.

- **TOLERANCE**: This specifies the value PC-DMIS uses to determine the maximum distance from the plane a point must be in order for PC-DMIS to consider it as part of the cross section in the **Delta** box.
- **INCREMENT**: This specifies the value between cutting planes in the **Step** box.
- **LENGTH**: This specifies the value between the first and last cutting planes in the **Length** box.
- **SMOOTHING TOLERANCE**: This specifies the tolerance value to refine the points associated with the generated cross section in the **Smoothing Tol** box.

Defining the Cross Section by Using the Graphic Display Window

To define some of the cross section parameters, click the CAD model in the Graphic Display window to select the **Start Point**. A pink line appears. Click a second point on the CAD model to determine the **Direction** vector and the **Length**.

Creating a Profile Dimension from the Graphic Display Window

When you double-click a cross section label, PC-DMIS creates a new profile dimension that evaluates the selected cross section.

Exporting Mesh Cross Sections

Once you create the Mesh cross sections, you can export them in IGES, STL ASCII, STL BIN, and XYZ formats from the **Mesh Operator** dialog box.

- If you select the IGES or the XYZ format, PC-DMIS only lists Mesh Cross Section commands in the feature list.
- If you select one of the STL formats, PC-DMIS only lists Mesh commands in the feature list.

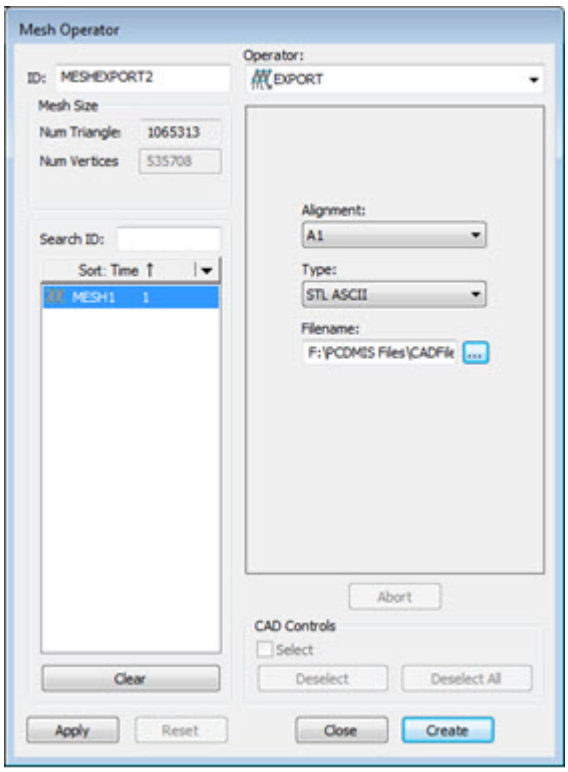
For details on the Mesh Export operator, see "Mesh EXPORT Operator" in this documentation.

Mesh EXPORT Operator

Exporting Mesh Data

To export mesh data with the Mesh EXPORT operator:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Mesh Operator** button () to access the **Mesh Operator** dialog box.



Mesh Operator dialog box - Export operator

2. From the **Operator** list, select the **EXPORT**.
3. From the **Feature List** box, select the Mesh.
4. Select the options to use. The Mesh EXPORT operator uses these options:

   Alignment: This option indicates the type of alignment to include when exporting the data.

   Type: This list provides the options to determine the file type for the EXPORT operator. The file type options for the EXPORT operator are STL ASCII and STL Bin to export the Mesh data object.

   If you have Mesh Cross Sections defined, the list contains a third option called IGES. You can also click the **Export Mesh Cross Section to IGES** button from the **Mesh** toolbar (**View | Toolbars | Mesh**) to perform this operation. For details on the Mesh Cross Section operator, see "Mesh CROSS SECTION Operator" in this documentation.

   Filename: Indicates the name of the export file. Enter the path and file name or use the **Browse** button to navigate to it.

5. Click **Create** and PC-DMIS inserts the EXPORT command into the Edit window. The command is `MESH/OPER,EXPORT` . The mesh data exports to the file location defined in the **Filename** box.

   For example:

```
 MESHEXPORT1=MESH/OPER,EXPORT,FORMAT=STL ASCII,FILENAME=F:
\PCDMIS FILES\STL\TEST1_STL.STL,

    REF,MESH1,,
```

Exporting Mesh Cross Section Data in IGES Format

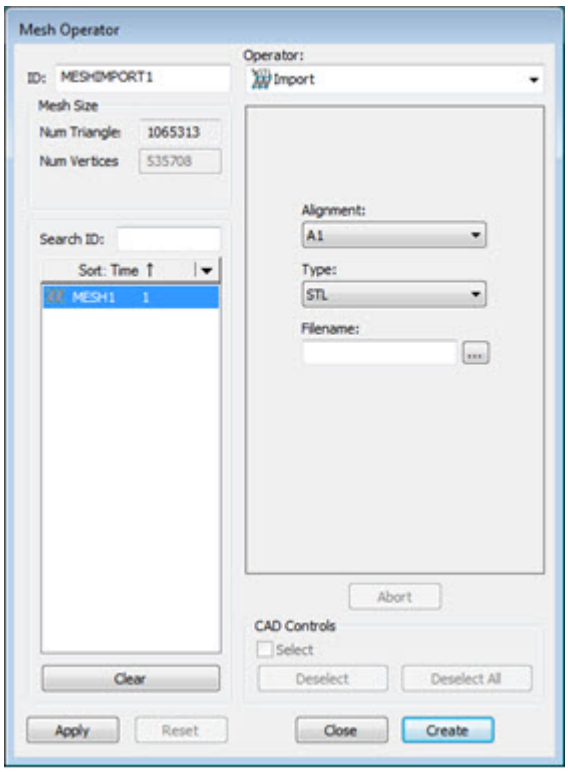
To export Mesh cross section data in IGES format:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Mesh Operator** button, or select it from the menu (**Insert | Mesh | Operator**) to open the **Mesh Operator** dialog box.
2. From the **Operator** list, select the **Mesh Export** operator.
3. From the **Type** list, select the **IGES** option. The software displays all Mesh cross sections in the **Feature List** box.
4. Select the cross sections in the **Feature List** box that you want to export.
5. Click **Apply** and then click **Create** to complete the export of Mesh Cross Sections in IGES format.
6. Click **Close** to return to the main PC-DMIS screen.

Mesh IMPORT Operator

To create a Mesh IMPORT operator:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Mesh Operator** button () to access the **Mesh Operator** dialog box.



Mesh Operator dialog box - Import operator

2. From the **Operator** list, select **IMPORT**.
3. From the **Feature List** box, select the Mesh.
4. Select the options to use. The Mesh IMPORT operator uses these options:

   Alignment: Indicates the type of alignment to include when importing the data.

   Type: The option for the IMPORT operator is **STL**.

   Filename: Indicates the name of the import file. Enter the path and file name, or use the **Browse** button to navigate to it.

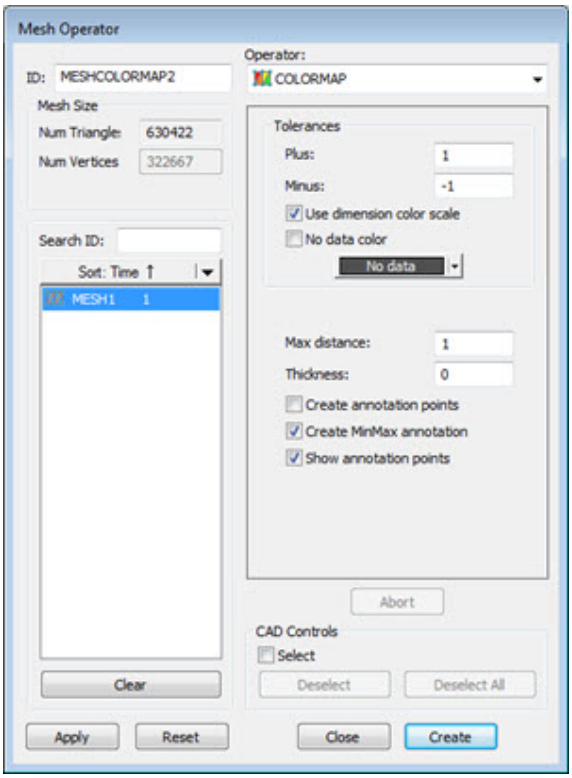
5. Click **Create**. PC-DMIS inserts the IMPORT command into the Edit window. The command is `MESH/OPER,IMPORT` . The mesh data imports from the file location defined in the **Filename** box.

   For example:

```
MESHIMPORT1=MESH/OPER,IMPORT,FORMAT=STL,FILENAME=F:\PCDMIS
FILES\STL\Test2_STL.STL,

     REF,MESH1,,
```

Mesh COLORMAP Operator



Mesh Operator dialog box - Colormap operator

The Mesh COLORMAP operation applies a colored shading to the selected mesh. The software shades the colormap according to the deviations of the mesh compared to CAD. PC-DMIS uses the colors defined in the **Edit Dimension Colors** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes discussed below.

Because the Mesh colormap shows the color deviations on the Mesh object, when you apply the colormap, the software hides the CAD model. By comparison, the Pointcloud colormap colors the deviations on the CAD model, and PC-DMIS does not hide the CAD model. To show or hide the CAD model, click the **Show CAD** button () on the **Graphic Items** toolbar. For details, see "Graphic Items Toolbar" in the "Using Toolbars" chapter in the PC-DMIS Core documentation.

The colors that PC-DMIS uses for the colormap come from the **Edit Dimension Colors** dialog box (**Edit | Graphic Display Window | Dimension Colors**).

Select **View | Other Windows | Dimension Colors** to view the color scale from the **Dimension Colors Bar**.

Show / Hide Colormaps

You can show or hide colormaps in the Graphic Display window in different ways. When hidden, PC-DMIS does not show the colormaps in the Graphic Display window as you move through the Edit window.

The **Activate Colormap** button has two states: Enabled and Disabled. From the **Graphics Items** toolbar or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**), click the **Activate Colormaps** button () so it's in the enabled state (). PC-DMIS now actively shows colormaps in the Graphic Display window.

To hide the colormaps in the Graphic Display window, click the **Activate Colormaps** button again so it's in the disabled state (). You can also select **None** from the **Colormaps** list to disable colormaps.

To show the colormaps:

- Click the **Activate Colormaps** button so it's in the enabled state. When you enable this button, PC-DMIS displays the colormaps in the Graphic Display window based on the cursor position in the Edit window.
- Select a colormap from the **Colormaps** list.
- When you apply or execute a colormap, PC-DMIS automatically sets the **Activate Colormaps** button to the enabled state.



When the cursor is on a Mesh, Point, Surface or Thickness colormap in the Edit window, the active colormap appears in the Graphic Display window. PC-DMIS also displays the **Colormap ID** in the **Colormap** combo box.

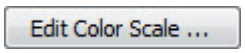
If your cursor is above all the colormaps in the Edit window, PC-DMIS does not show any colormap in the Graphic Display window, and sets the **Colormap** combo box to **None**.

To apply the Mesh COLORMAP operation to a Mesh:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Color a Mesh** button (), or from the menu, select **Insert | Mesh | Colormap**.
2. Update these options based on your needs:

   Tolerances - Use this option to set the upper (**Plus**) and lower (**Minus**) tolerance values.

   Use dimension color scale check box - Select this check box to use the **Dimension Colors Bar** to define the color bar used for the Mesh Colormap color properties. For details, see "Using the Dimension Colors Window (Dimension Colors Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



Edit Color Scale - If you do not select the **Use dimension color scale** check box, the software enables the **Edit Color Scale** button. When you click this button, the functionality to dynamically change the color, scale and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. For details, see the "Edit the Color Scale" topic.

No data color check box - When you select this check box, the software maps the selected color to areas on the selected surfaces where no data is found.

Max distance - The software only includes points that fall within the **Max distance** value as part of the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10%, for example) than the largest deviation.

Thickness - This adds a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to the Mesh surface model.

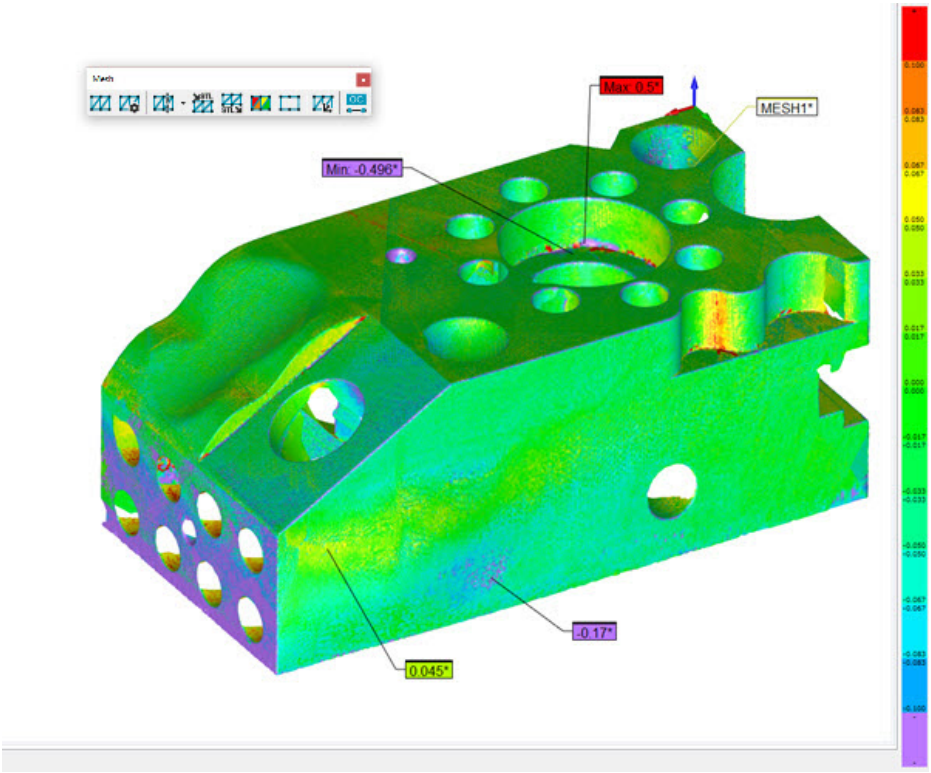
Create annotation points check box - Annotations are a way to display the deviation for a specific location on a surface colormap with its associated color. To create an annotation:

a. Select the **Create annotation points** check box. This clears the **Select** check box in the **CAD Controls** area and disables most of the options on the right side of the dialog box.

b. Select a point on the colormapped Mesh in the Graphic Display window. PC-DMIS evaluates and creates an annotation label in the same background color as the Mesh deviation point with the deviation value. You can move the label around in the Graphic Display window as any other label.

> Once you create annotation labels, they remain in the same position and have the same characteristics when you restart the measurement routine, or if you restart PC-DMIS and reload the same measurement routine.

Create MinMax annotations check box - When you select this check box, PC-DMIS creates the minimum and maximum values in the form of annotation labels for the active Mesh Surface Colormap.



Example of mesh colormap with min, max, and various point annotation labels displayed

The software calculates the minimum and maximum points each time you execute the measurement routine.

To show, hide, or delete annotation labels, right-click on a label to display the shortcut menu, and then select the appropriate option.

Delete Annotation - The software deletes the selected annotation label.

Show All Annotations - The software displays all annotation labels.

Hide All Annotations - The software hides all annotation labels.

Delete All Annotations - The software deletes all annotation labels.

Show annotation points check box - When you select this check box, the software displays all annotation points.

3. Click **Create** to insert a `MESH/OPER,COLORMAP` command into the Edit window as shown in the code below.

```
 MESHCOLORMAP1=MESH/OPER,COLORMAP,PLUS TOLERANCE=0.5,MINUS
TOLERANCE=-0.5,THICKNESS=0,MAX DISTANCE=1,

     REFINE FACTOR=0.1,TRIANGLES=401063,VERTICES=206625,

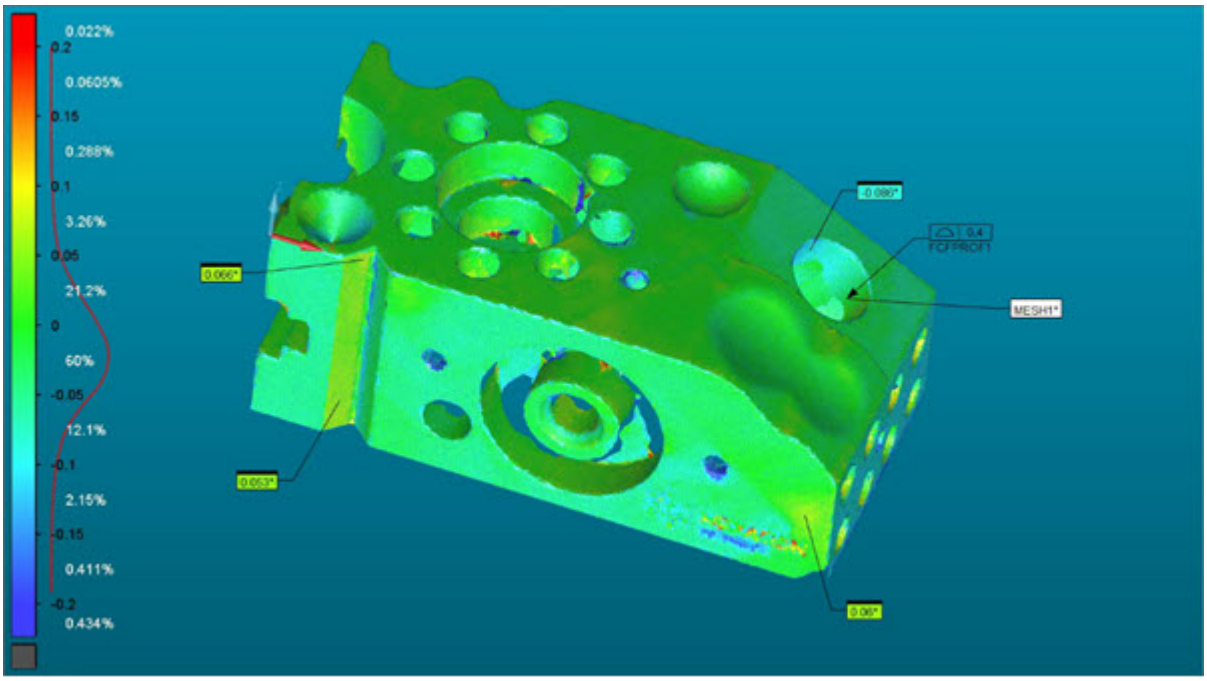
     REF,MESH1,,
```

Colormaps in the Report

For information on how the software shows colormaps in the report, see "Colormaps and the CadReportObject" in the "Reporting Measurement Results" chapter of the PC-DMIS Core documentation.

Dimensioning Surface Profile Using the Mesh COLORMAP

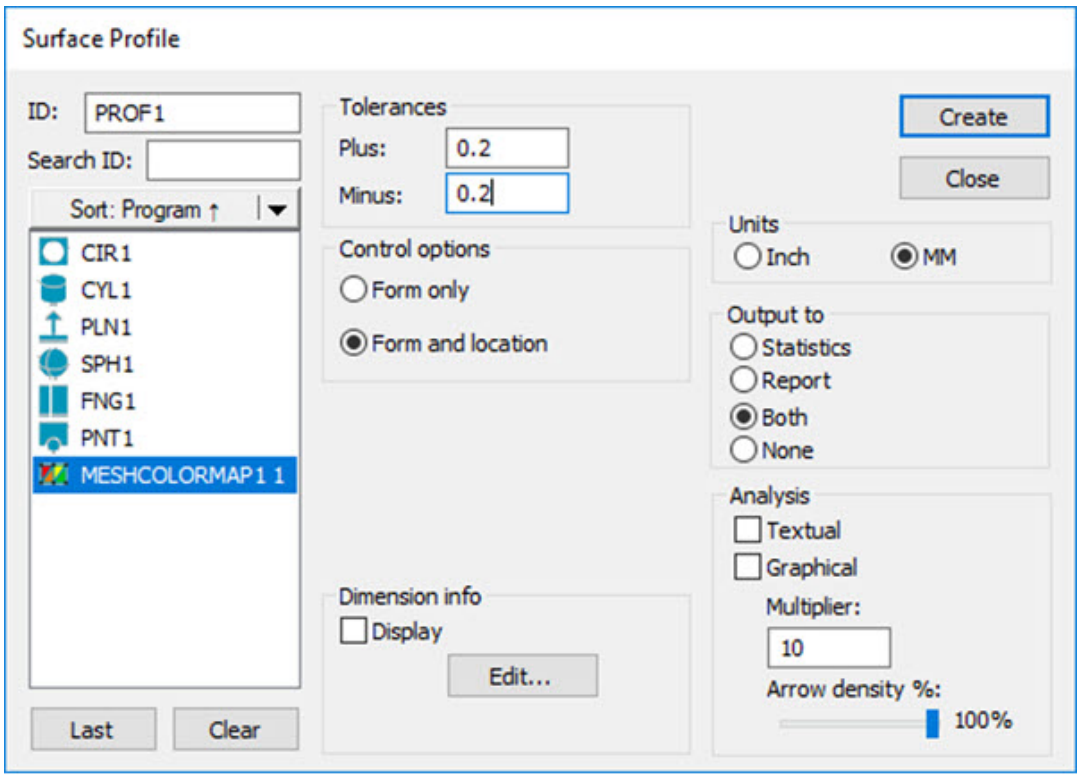
You can use a Mesh COLORMAP to create a Dimension Surface Profile.



Example of a dimension surface profile created using a mesh colormap

To create a Dimension Surface Profile from a Mesh COLORMAP, follow these steps:

1. Create a Mesh COLORMAP. For details, see "Mesh COLORMAP Operator".
2. Make sure you have the **Use Legacy Dimensions** option selected (**Insert | Dimension | Use Legacy Dimensions**).
3. Click the **Profile Surface Dimension** option from the **Dimension** toolbar (**View | Toolbars | Dimension**) or select it from the menu (**Insert | Dimension | Profile | Surface**). The **Surface Profile** dialog box opens.

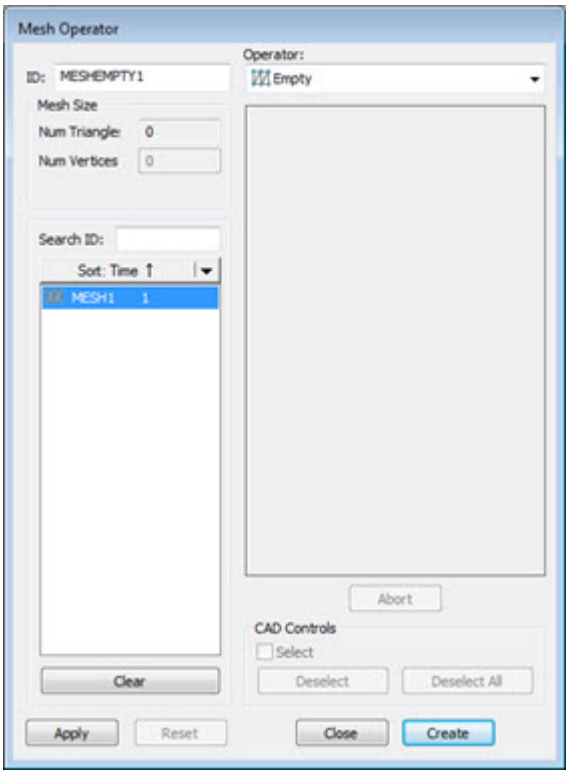


Surface Profile legacy dialog box for mesh colormap

For details on creating a legacy Surface Profile, see "To Dimension a Feature Using the Surface Profile Option" in the "Using Legacy Dimensions" chapter of the PC-DMIS Core documentation.

4. From the **Features** list box, select the desired Mesh COLORMAP.
5. Set the other options as needed and then click **Create**.

Mesh EMPTY Operator



Mesh Operator dialog box - Empty operator

When this command is executed, PC-DMIS removes all the data from the Mesh.

To apply the Mesh EMPTY operation to a Mesh:

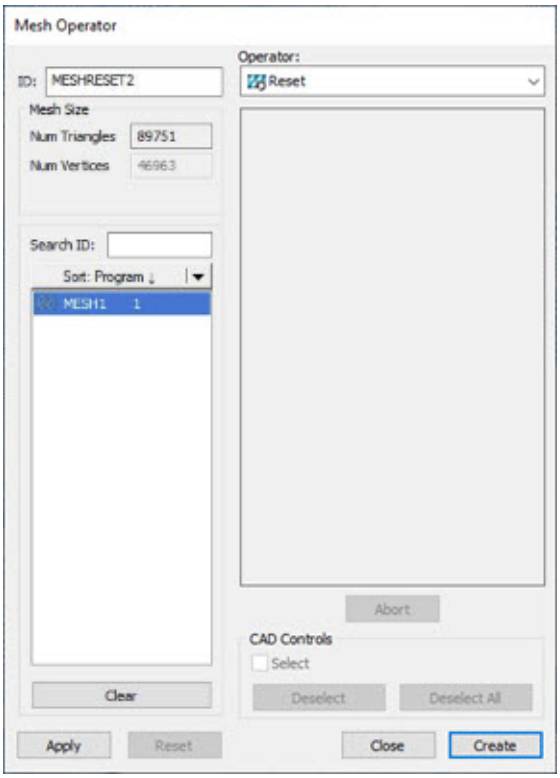
1. In the Edit window, position your cursor just above the Mesh you want to empty.
2. Click **Empty a Mesh** () on the **Mesh** toolbar, or select the menu option **Operation | Mesh | Empty**. The **Mesh Operator** dialog box appears.
3. Click **Create** to insert a `MESH/OPER,EMPTY` command into the Edit window. The software inserts it just above the Mesh you want to empty. This is the Mesh that the Empty command acts on.

   For example:

   ```
   MESHEMPTY1 =MESH/OPER,EMPTY,
       REF,MESH1,,
   ```

   > Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.

Mesh RESET Operator



Mesh Operator dialog box - Reset operator

The Mesh RESET operation reverts all Mesh SELECT operations and returns the original Mesh object.

To immediately apply the Mesh RESET operation, from the **Mesh** toolbar, click the **Mesh RESET** ( ) button.

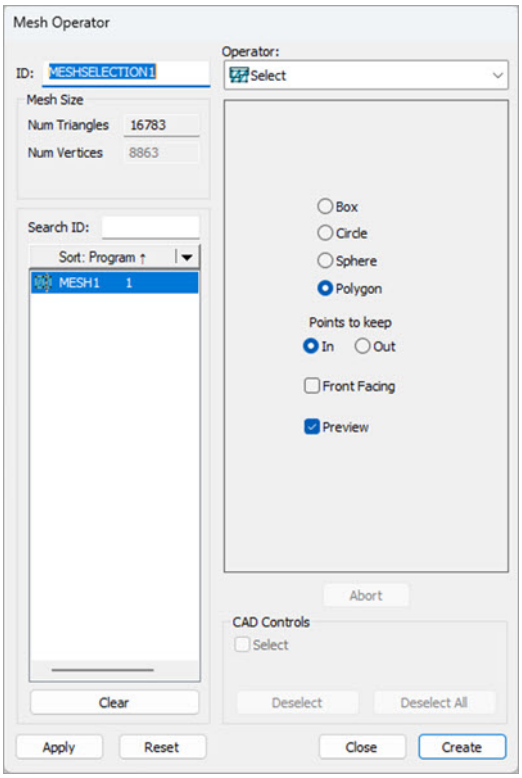
To create the Mesh Reset command:

1. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Mesh Operator** button to open the **Mesh Operator** dialog box.
2. From the **Operator** list, select the **Reset** operator.
3. Select the Mesh from the ID list and then click **Apply**.
4. Click **Create** to insert a `MESH/OPER,RESET` command into the Edit window as shown below.

```
MESHRESET1 =MESH/OPER,RESET,

    REF=MESH1,,
```

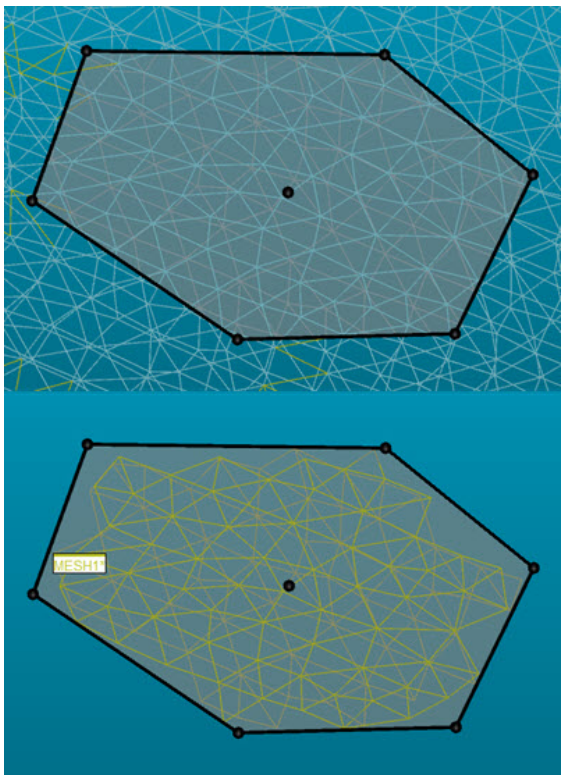
Mesh SELECT Operator



Mesh Operator Dialog Box - Select Operator

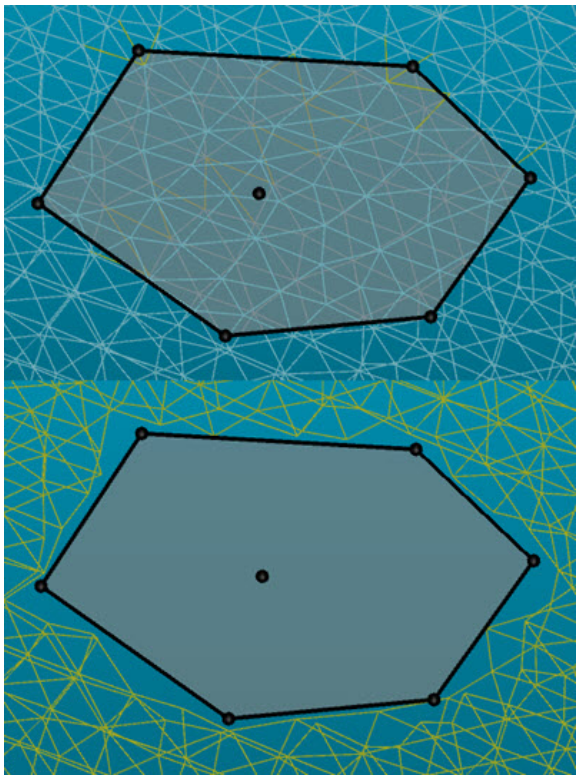
The **Mesh SELECT** operation selects a subset of triangles contained in a Mesh command.

- PC-DMIS keeps only the subset of triangles that are strictly inside the polygon selection when you select the **In** option.



The top Image is the initial selection.
The bottom image is the result after you click the Apply button.

- PC-DMIS keeps only the subset of triangles that are strictly outside the polygon selection when you select the **Out** option.



The top Image is the initial selection.
The bottom image is the result after you click the Apply button.

You can choose the **Box**, **Circle**, **Sphere**, or **Polygon** option. By default, the software uses the **Polygon** option when you first click the **Select Method** button from the **Mesh** toolbar.

To select a region of triangles:

1. With the **Mesh Operator** dialog box open, use the **ID** list to select the Mesh ID that you want to apply the selection to.
2. From the **Mesh Operator** dialog box, choose a selection method. The options are described below.

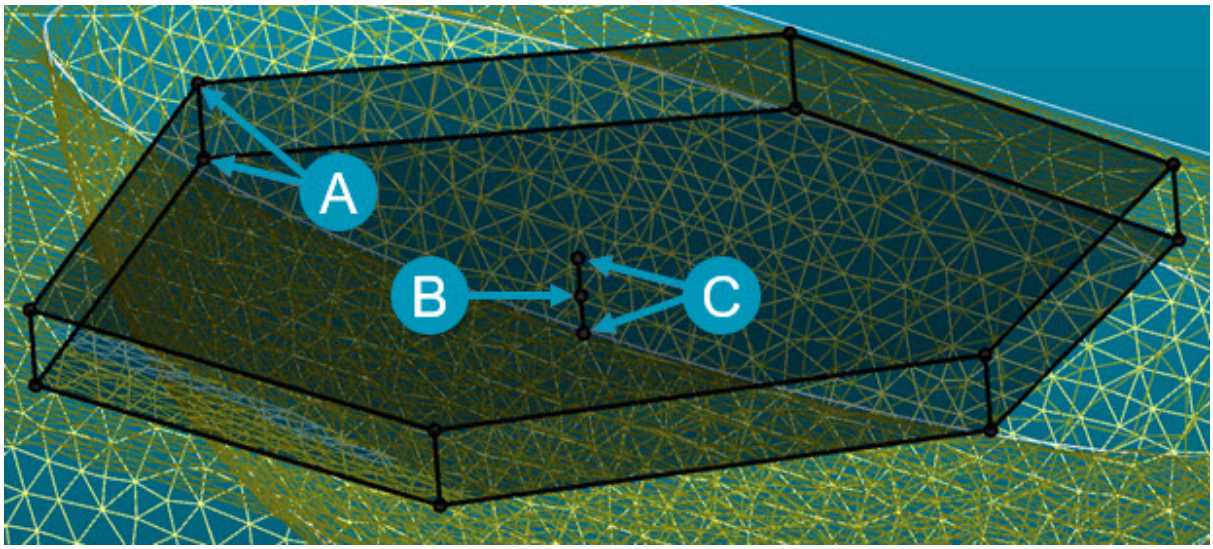
Use Ctrl + click when you make the selection. This snaps the center of the selection onto the Mesh.

Also, it may be helpful to turn off the **Show Graphic Window as Solid** option from the **Graphic View** toolbar (**View | Toolbars | Graphic View**) so you can see the individual triangles.

Selection Method	Points Needed
Box	Click and drag to define a rectangular shape.
Circle	Click to set the center of the circle and then drag to define the circle size.
Sphere	Click to set the center of the selection sphere and then drag to define the spherical shape.
Polygon	Click to define the vertices of the polygon. To close the polygon selection, double-click your left mouse button or press the End key.

3. Once you define the area of the Mesh that you want to apply the selection method, double-click or press the End key to close the selection.

4. Once you create the selection, you can drag the boundary control points to refine the region. You can also drag the center control point to re-position the shape.



Polygon Region Showing:
***A** - Region Boundary Points*
***B** - Region Center Control Point*
***C** - Region Center Depth Points*

5. By default, the selection is 3D with an infinite depth in the current view. You can drag the center depth control points to adjust the selection's depth.

6. If you want to keep only the subset of triangles that are strictly inside the selection, click the **In** option on the **Mesh Operator** dialog box. If you want to keep only the subset of triangles that are strictly outside the selection, click the **Out** option.

7. If you want to keep only the subset of triangles that are front facing in the selection view, click the **Front Facing** check box. For details, see the "Mesh Select - Front Facing" topic.

8. The software selects the **Preview** check box by default. This allows you to see the Mesh triangles when you select a region. You may want to clear the check box so the selected Mesh triangles are not highlighted. This may be helpful if you have a very large Mesh object and showing the selected triangles affects performance.

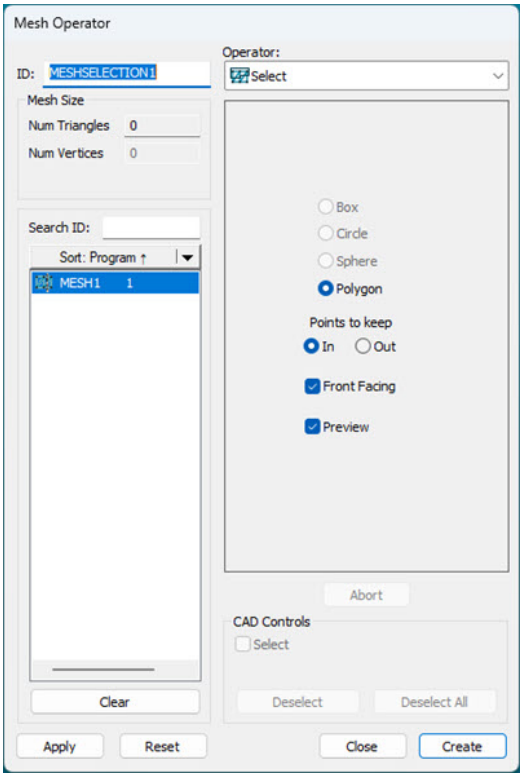
9. Click the **Apply** button to perform the selection.

10. Click the **Create** button to finish. PC-DMIS creates the `MESH/OPER,SELECT` command in the Edit window as shown below.

```
MESHSELECTION1=MESH/
OPER,SELECT,POLYGON,INSIDE=YES,FRONT_FACING=YES,0,
            <55.382,56.316,-0.003>,
   <56.806,57.098,0.087>,
            <57.689,57.124,-0.357>,
   <57.345,56.348,-1.019>,
            <56.224,55.748,-1.072>,
   <55.18,55.659,-0.611>,
         SIZE=0,REF=MESH1,,
```

A 0 (zero) depth value means infinite depth, cut through the entire mesh in the selection view.

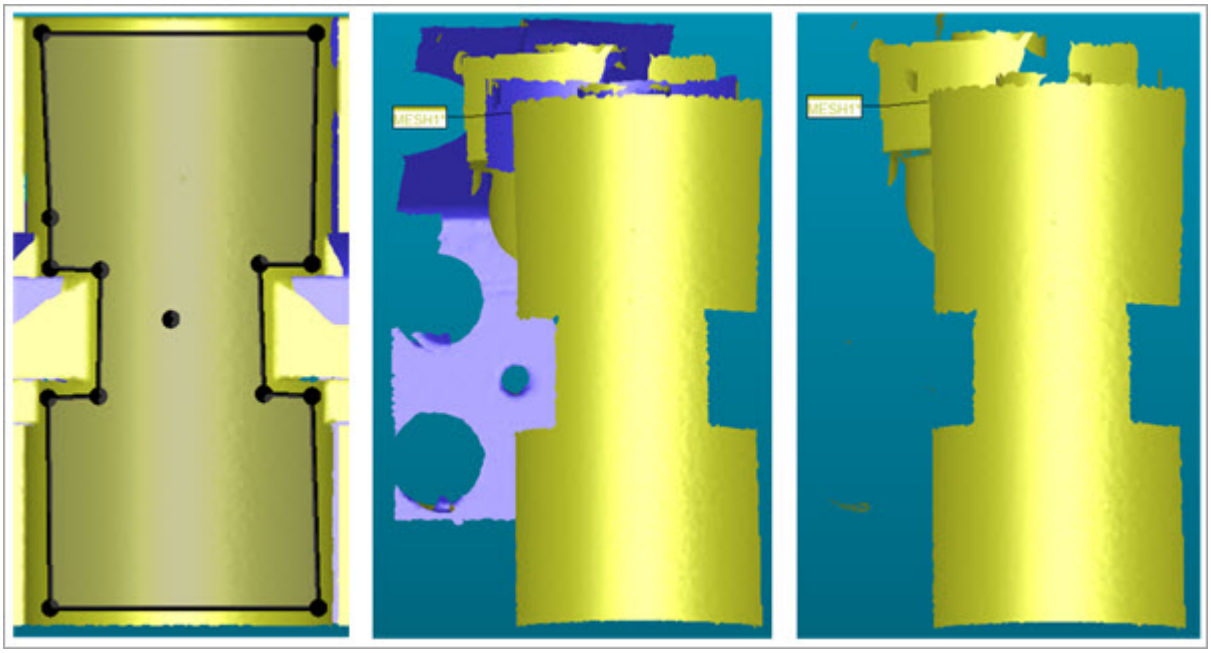
Mesh Select - Front Facing



Mesh Operator dialog box - Select operator with Front Facing option selected

If you only want to select the triangles that are front facing in the selection view, click the **Front Facing** check box on the **Mesh Operator** dialog box. When you disable this check box, PC-DMIS selects the triangles that are in both the front and rear facing views.

To see the front and back faces as contrasting colors, enable the **Show Graphic Window as Solid** option on the **Graphic Items** toolbar (**View | Toolbars | Graphic Items**).



Examples of Selection View (left), Selection with Front Facing disabled (middle), Selection with Front Facing enabled (right)

Mesh Thickness Colormap

The Thickness Colormap allows you to show and measure the part thickness as a colormap using only the Mesh or Pointcloud (COP) data object. You can also compare the measured thickness to the nominal CAD model thickness.

> The measured data object must have data on two opposing sides which have the opposite normal orientations to use this function.

To create a Mesh Thickness Colormap, from the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Mesh Thickness Colormap** button  to open the **Mesh Operator** dialog box. You can also do this from the **Thickness Colormap** menu option (**Insert | Mesh | Thickness Colormap**).

When you perform a thickness colormap of the data object (Pointcloud or Mesh), PC-DMIS calculates the measured thickness up to a **Max Thickness** value. The software does not evaluate any data value greater than the **Max Thickness**.

When the measured data is aligned to a CAD model, you can choose to create a thickness colormap which shows the deviation of the measured thickness in comparison to the nominal CAD model thickness.

> If you use a large **Max Thickness** value on a large data object, it can result in a longer processing time.

You can create the following thickness colormaps:

- Measure part thickness using either a Pointcloud or Mesh data object
- Create Thickness Colormap on specific CAD surfaces when you mark the **Select** check box in the **CAD Control** area of the dialog box and then select the CAD surfaces in the Graphic Display window.
- Compare to CAD thickness colormap, which shows the deviation of the Pointcloud or Mesh data object thickness in comparison to a CAD model.

Show / Hide Colormaps

You can show or hide colormaps in the Graphic Display window in different ways. When hidden, PC-DMIS does not show the colormaps in the Graphic Display window as you move through the Edit window.

The **Activate Colormap** button has two states: Enabled and Disabled. From the **Graphics Items** toolbar or from the menu (**Operation | Graphic Display Window | Graphic Items | Activate Colormaps**), click the **Activate Colormaps** button () so it's in the enabled state (). PC-DMIS now actively shows colormaps in the Graphic Display window.

To hide the colormaps in the Graphic Display window, click the **Activate Colormaps** button again so it's in the disabled state (). You can also select **None** from the **Colormaps** list to disable colormaps.

To show the colormaps:

- Click the **Activate Colormaps** button so it's in the enabled state. When you enable this button, PC-DMIS displays the colormaps in the Graphic Display window based on the cursor position in the Edit window.
- Select a colormap from the **Colormaps** list.
- When you apply or execute a colormap, PC-DMIS automatically sets the **Activate Colormaps** button to the enabled state.



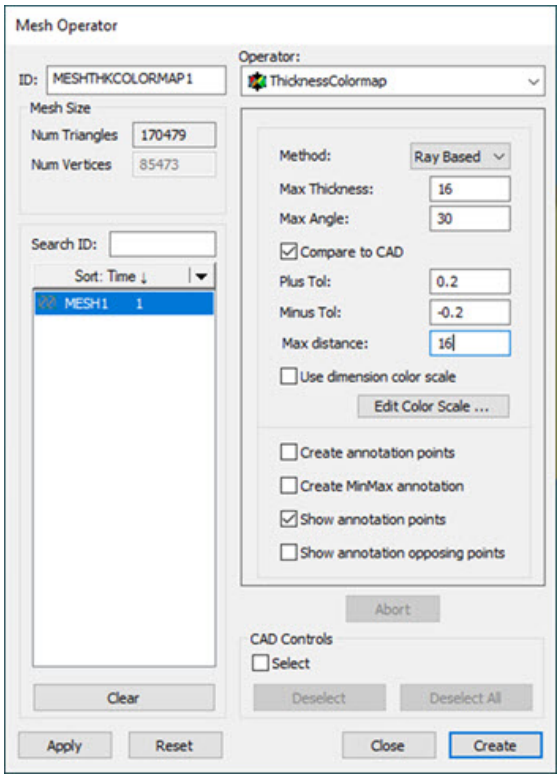
When the cursor is on a Mesh, Point, Surface or Thickness colormap in the Edit window, the active colormap appears in the Graphic Display window. PC-DMIS also displays the **Colormap ID** in the **Colormap** combo box.

If your cursor is above all the colormaps in the Edit window, PC-DMIS does not show any colormap in the Graphic Display window, and sets the **Colormap** combo box to **None**.

Measured Mesh Thickness Colormap

To measure a Mesh Thickness Colormap:

1. Click the **Thickness Colormap a Mesh** button () from the **Mesh** toolbar (**View | Toolbars | Mesh**) or click the menu option (**Insert | Mesh | Thickness Colormap**) to open the **Mesh Operator** dialog box for the Mesh Thickness Colormap operator.



2. Select a Mesh data object from the list.

3. From the **Method** list, select the measurement method type **Ray based** or **Sphere**. For details on these measurement methods, see "Thickness Colormap Method".

4. Type the value for **Max Thickness**. The software does not evaluate any data value greater than the **Max Thickness** value.

5. Type a **Max Angle** value. The software returns the relevant points whose estimated normals are within the specified angle of the CAD normals.

6. To create a thickness colormap of a Mesh data object in comparison to a CAD model, select the **Compare to CAD** check box. For details, see the "Compare to CAD Mesh Thickness Colormap" topic in this documentation.

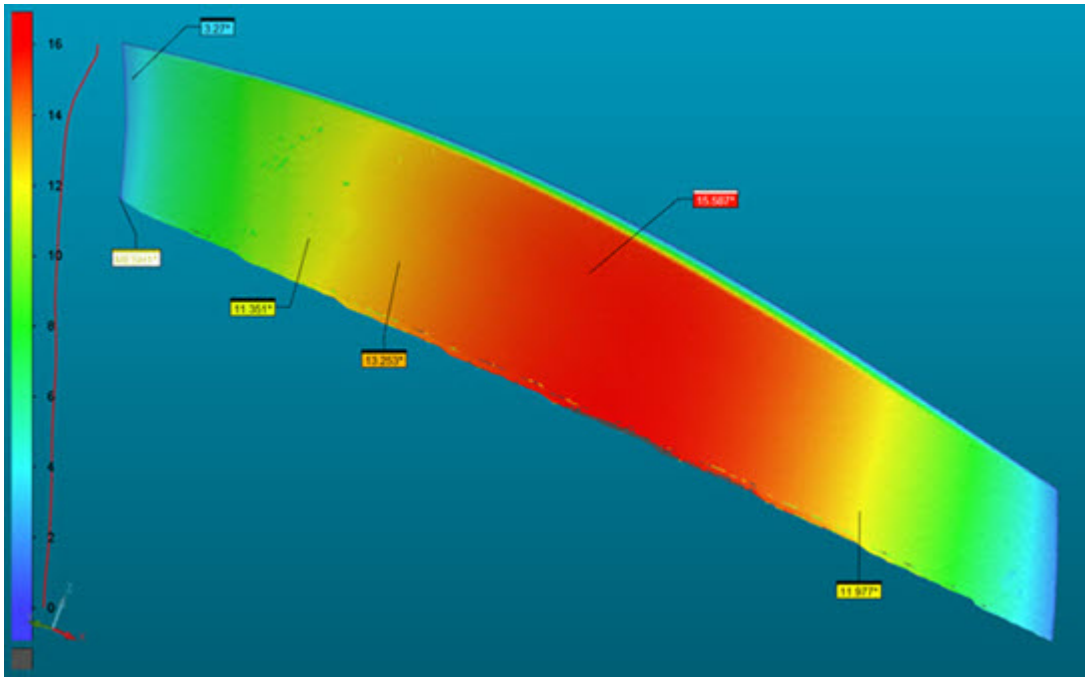
7. If you want to determine the Mesh Thickness Colormap on specific CAD surfaces, in the **CAD Control** area of the dialog box, mark the **Select** check box. You can then click the surfaces you want to perform the Mesh Thickness Colormap on.

8. Click **Apply** to perform the analysis necessary to create the Mesh Thickness Colormap operator.

9. Create annotations. For details on how to create annotation points for the Mesh Thickness Colormap operator, see "Thickness Colormap Annotations".

10. Click **Create** to create the command in the Edit window.

    You can see in the image below the result of applying a Thickness Colormap using a Mesh data object:



Example of a thickness colormap using a mesh data object

The command in the Edit window might look something like this:

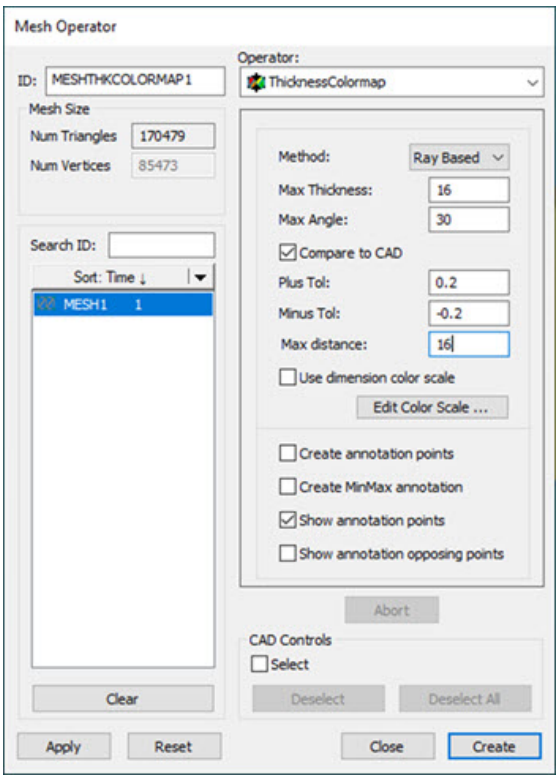
```
 MESHTHKCOLORMAP1=MESH/OPER,THICKNESSCOLORMAP,,SHOW
PARAMETERS=NO
TRIANGLES=170479,VERTICES=85473,REF=MESH1,,,,
```

Compare to CAD Mesh Thickness Colormap

> The **Compare to CAD** thickness colormap processing can be very time-consuming when working with a large mesh.

You can create a thickness colormap of a Mesh data object in comparison to a CAD model. In this case, from the **Mesh Operator** dialog box (**Insert | Mesh | Thickness Colormap**), select the **Compare to CAD** check box.

> When you select CAD faces for a Thickness Colormap compared to CAD, PC-DMIS uses both sides of the material to compute the thickness; however, PC-DMIS only colors the data for the selected faces.



PC-DMIS computes the deviation of the data object thickness in comparison to a CAD model.

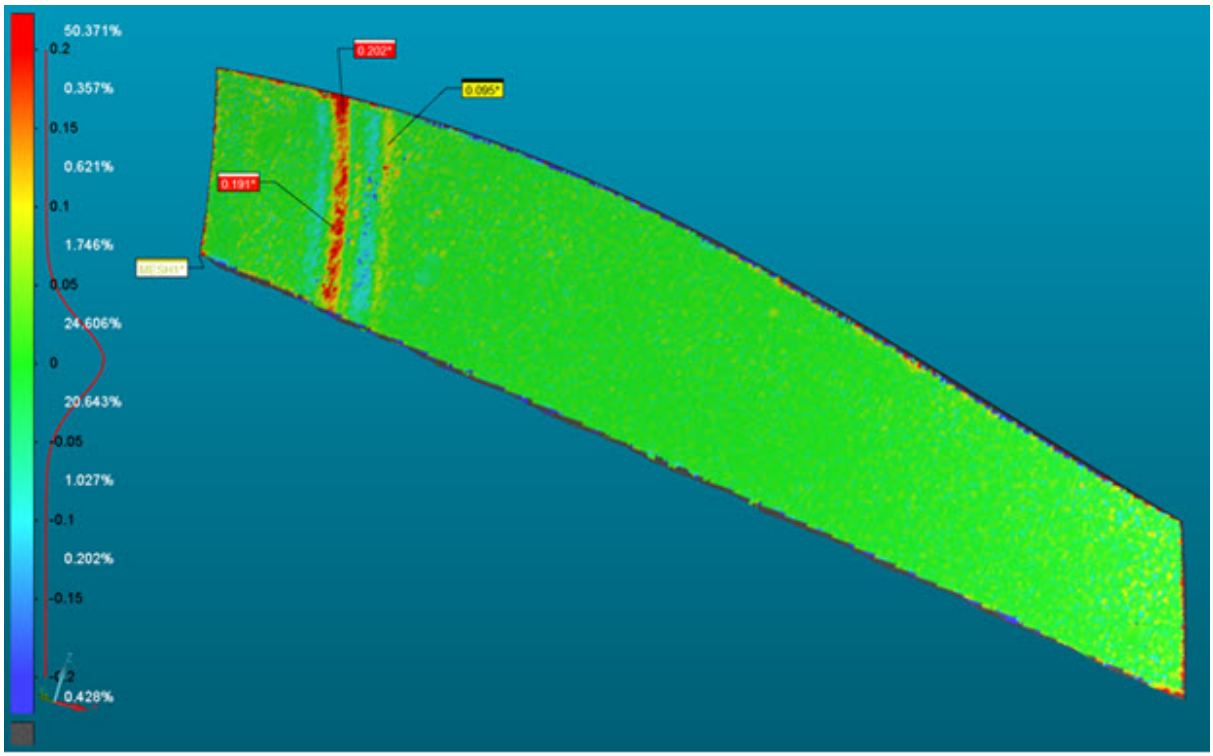
The software colormaps the Mesh data object used for the comparison to CAD to show the deviations.

To do this:

1. From the **Mesh Operator** dialog box, select **ThicknessColormap** from the **Operator** list.
2. Choose the corresponding Mesh data object.
3. From the **Method** list, select the method: **Ray Based** or **Sphere**. For details on these methods, see "Mesh Thickness Colormap Method".
4. Type the **Max Thickness**. The software does not evaluate any data value greater than this value.
5. Type the **Max Angle** value. This is the maximum angle between the ray that pierces the mesh and the normal at the pierced point.
6. Select the **Compare to CAD** check box and enter the **Max distance**. PC-DMIS uses the data within this distance from the CAD model for the colormap.
7. Type the appropriate tolerance values in the **Plus Tol** and **Minus Tol** boxes. You must use a minus sign when you type a negative number.
8. Click **Apply**.

9. Create annotations. For details on how to create annotation points for the Mesh Thickness Colormap operator, see "Thickness Colormap Annotations".

10. Click **Create**.



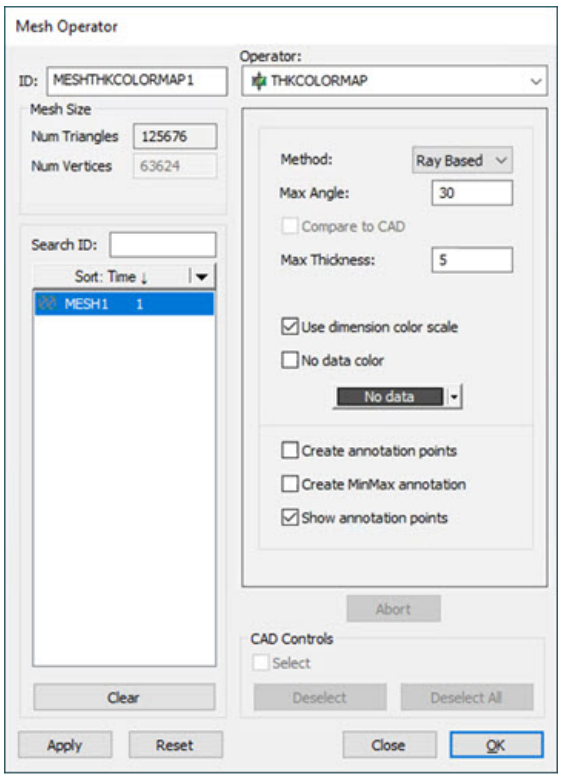
Example of a thickness colormap, mesh data object compared to CAD model

Mesh Thickness Colormap Method

> When you measure the thickness of a Pointcloud data object, you cannot select a method type. PC-DMIS automatically uses the Sphere method.

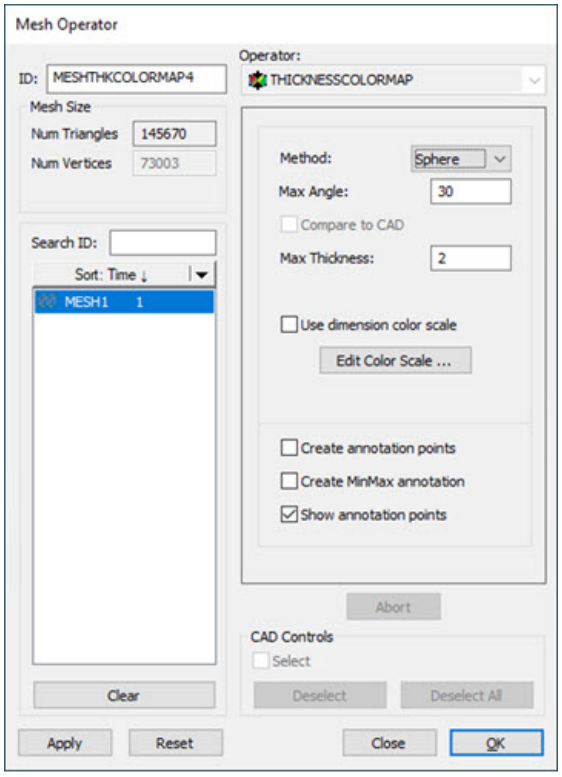
You can choose between two mathematical methods for computing the Mesh Thickness Colormap:

- **Ray Based** Method



When you select the **Ray Based** method for a Mesh, PC-DMIS pierces the Mesh at each vertex along its normal to the opposing side. The software then uses the data within the **Max Angle** to calculate the thickness.

- **Sphere** Method



When you select the **Sphere** method, PC-DMIS uses a maximum inscribed sphere to calculate the thickness between the two opposing sides.

For either method, if you select the **Use the dimension color scale** option and no data exists on the opposing side, PC-DMIS uses the No data color in that area. If you do not select the **Use dimension color scale**, you can define the No data color with the **Edit Color Scale**.

For details on how to edit the dimension color scale, see "Using the Dimension Colors Window (Dimension Colors Bar)" in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

For details on how to use the **Edit Color Scale** option, see "Edit the Color Scale" in this documentation.

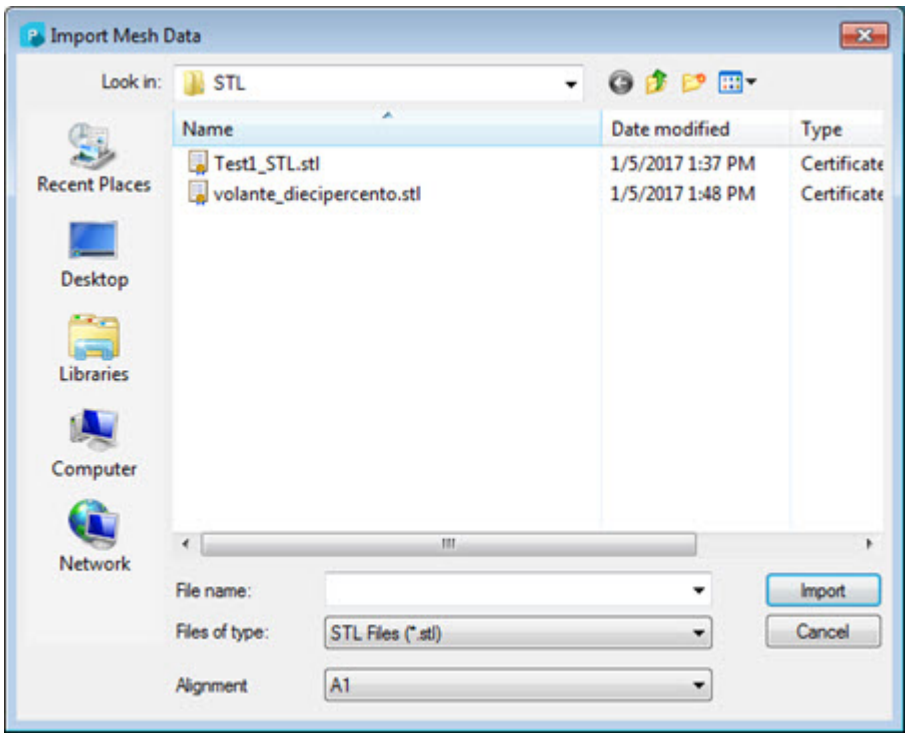
Import Mesh in STL Format

If a Mesh object does not exist in the PC-DMIS Edit window, then a new Mesh object is created and the STL data is imported. If a Mesh object already exists in the PC-DMIS Edit window, then the STL data is added to the Mesh object. If the data needs to be separated, you must create an empty Mesh object and then import the mesh STL data into that one.

The Mesh license must be enabled to use or view this option.

To import mesh data from an STL file:

1. Click the **Import Mesh in STL format** button () found on the **Mesh** toolbar (**View | Toolbars | Mesh**) to open the **Import Mesh Data** dialog box. You can also import a Mesh STL file from the menu (**File | Import | Mesh**).



Import Mesh Data dialog box

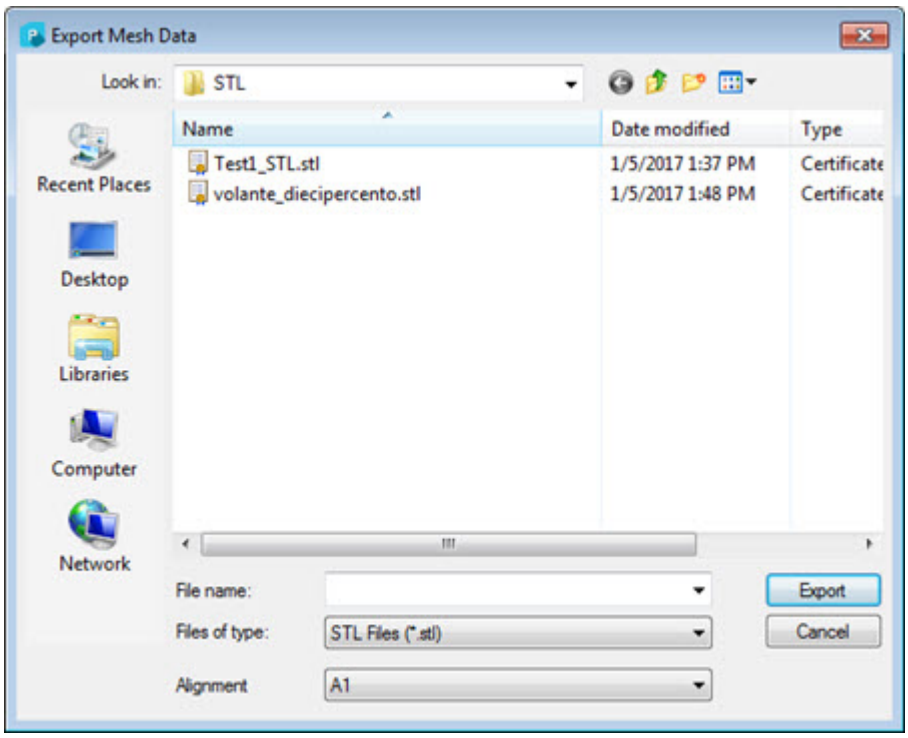
2. Use the dialog box to navigate to the location of the file containing the mesh data. Select the file type from the **Files of type** list to filter the list of files displayed in the dialog box. Left-click the file you want to import the mesh data from.
3. Select the alignment type from the **Alignment** list.
4. Click the **Import** button to import the mesh data. Click **Cancel** to exit the dialog box without importing any data.

Export Mesh in STL Format

You must have the Mesh license enabled to use or view this option.

To export mesh data to an STL file:

1. Click the **Export Mesh in STL format** button ( ) on the **Mesh** toolbar (**View | Toolbars | Mesh**) to open the **Export Mesh Data** dialog box. You can also export a Mesh in STL format from the menu (**File | Export | Mesh**).



Export Mesh Data dialog box

2. Use the dialog box to navigate to the location where you want to export the mesh data.
3. Enter a unique name for the file in the **File name** box.
4. From the **Alignment** list, select the alignment you want to apply to the mesh data.
5. Click the **Export** button to export the mesh data. Click **Cancel** to exit the dialog box without exporting the data.

Empty a Mesh

You must have the Mesh license enabled to use or view this option.

To empty a mesh:

1. In the Edit window, position your cursor on or just below the Mesh you want to empty. If you have two consecutive meshes defined in the Edit window, you must be on the mesh you want to empty.
2. Click the **Empty a Mesh** button  from the **Mesh** toolbar, or select **Operation | Mesh | Empty** from the menu.

   The Mesh is emptied of all its data.

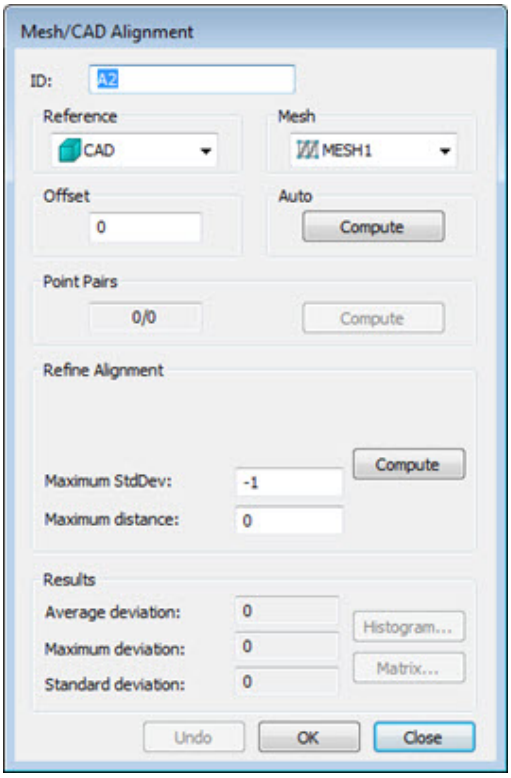
> Once you apply this command to a Mesh, there is no way to restore the Mesh data. You cannot click **Undo** to restore the lost data.

Mesh Alignment

In order to use the data you've collected in your mesh properly, you need to create an alignment between the mesh and the CAD data of your part model or between meshes. This is done using the **Mesh/CAD Alignment** dialog box.

You can access this dialog box from the **Mesh Alignment** button ( ) on the **Mesh** toolbar (**View | Toolbars | Mesh**).

Mesh/CAD Alignment Dialog Box Description



Default view of Mesh/CAD Alignment dialog box

The **Mesh/CAD Alignment** dialog box contains these options:

ID - This option displays the identification label for the alignment.

Reference - Select the reference object for your alignment, usually either the CAD itself or a defined Mesh. The mesh is aligned to the selected reference.

Mesh - This list lets you choose the mesh to use in the alignment.

Offset - This option defines an offset value for a surface CAD model and is typically used with sheet metal parts. Applying an offset value essentially gives the surface CAD model a thickness so that you can align the mesh data to a different face that isn't represented in the surface CAD model. For example, if you have a surface CAD model for the top of a part, but you want to align to a corresponding bottom surface, you could apply an offset value of the part's thickness to align the scanned data to the bottom side. Use a positive value if you want to apply a thickness in the same direction as the surface normal vector; use a negative value if you want to apply a thickness opposite the surface normal. It is only available for Mesh to CAD alignments.

Auto - This area lets you automatically align the CAD with the mesh with the **Compute** button. It is only available for Mesh to CAD alignments.

Point Pairs - This area lets you create a rough alignment based on selected points from the CAD that correspond to selected points from the Mesh. Once you have the needed pairs selected, click **Compute** to perform the rough alignment.

Refine Alignment - This area allows for a more refined alignment. Only the **Maximum Distance** option is available for Mesh-to-Mesh alignments.

Depending on the alignment being made, the **Refine Alignment** area of the dialog box may consist of the following items:

Total points - This box defines the number of random sampled points used to refine the alignment. This number must be a value of at least 3. A good number is around 200 points.

Maximum iterations - This box defines the number of repetitions the process makes in order to refine the alignment.

Compute - This button begins the refined alignment process. A progress bar on the status bar shows the progress as the process moves through the alignment iterations.

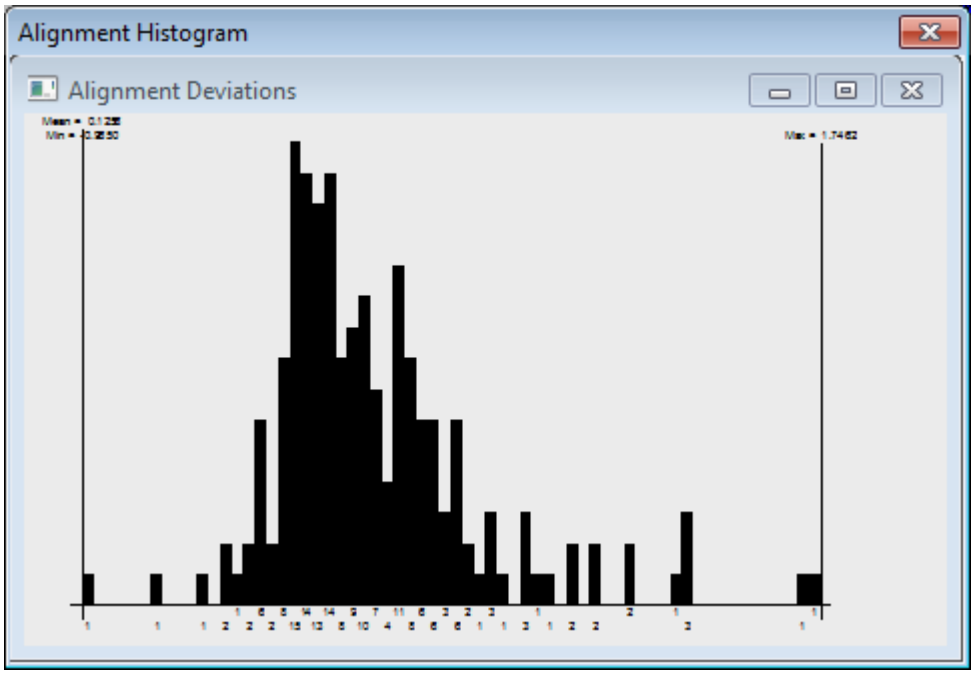
Maximum StdDev - This box defines the maximum standard deviation used during the execution of an auto alignment. If the entered value is exceeded during the command execution, you are prompted to optionally pick point pairs on the CAD/Pointcloud. A value of -1 disables the Maximum StdDev functionality.

Maximum Distance - This box defines the maximum distance PC-DMIS looks from the CAD for valid Mesh points. If there is no value, PC-DMIS uses the default value of 0 (zero), and the maximum distance becomes half the distance of the CAD bounding box.

Results - This area contains the following items:

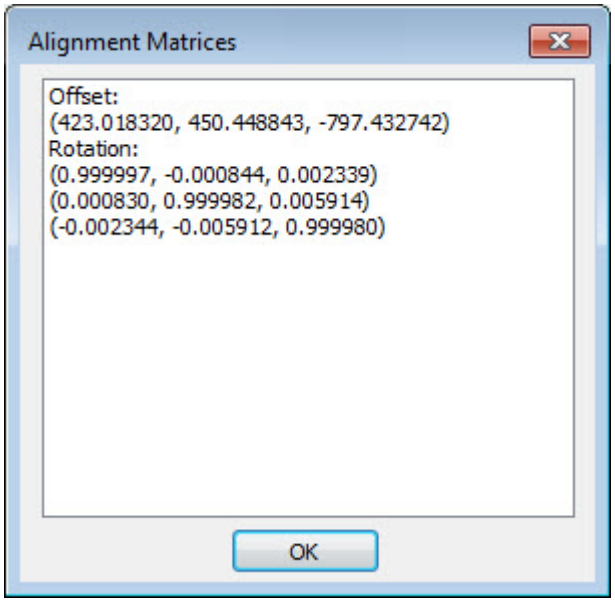
Information boxes showing the **Average Deviation**, **Maximum Deviation**, and **Standard Deviations** of the mesh data compared to the CAD model's data.

Histogram - This button takes a random sample of points from the mesh and projects them onto the CAD. The **Alignment Histogram** dialog box shows the deviations for that sample.



Sample Alignment Histogram dialog box for the selected mesh

Matrix - This button displays the **Alignment Matrices** dialog box for the mesh alignment. The numerical values of the mesh alignment in offset and rotation matrices are listed.



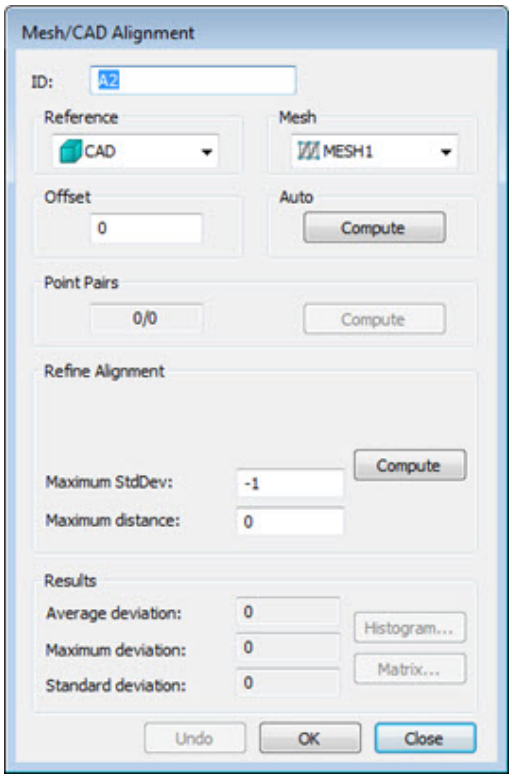
Sample Alignment Matrices dialog box for the alignment

Creating a Mesh/CAD Alignment

To create a Mesh to CAD alignment, do the following:

1. Ensure that you have an imported CAD model in the Graphic Display window and a `MESH` command in the measurement routine. These elements are required to align a mesh to the CAD.

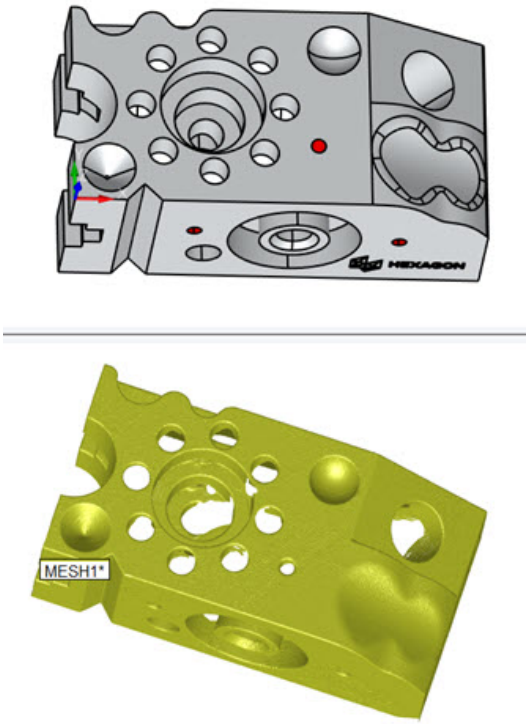
2. Select the **Insert | Mesh | Alignment** menu option or select the **Mesh Alignment** button ( ) on the **Mesh** toolbar. You can also access this dialog box by typing the `MESHCADBF` command in the Edit window's Command Mode between the `ALIGNMENT/START` and the `ALIGNMENT/END` commands. The **Mesh/CAD alignment** dialog box opens:



Mesh/CAD Alignment dialog box

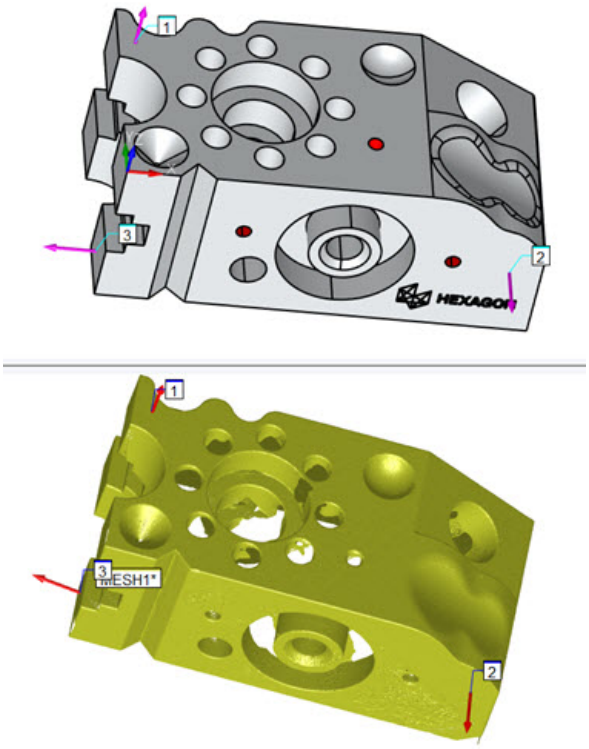
> For a complete description of the **Mesh/CAD Alignment** dialog box, see the topic "Mesh/CAD Alignment Dialog Box Description" in the PC-DMIS Laser documentation.

3. A temporary and split-screen view of the CAD model and the mesh appears in the Graphic Display window. You can use this split-screen view to visually see the alignment taking place. From the **Reference** drop-down list, select your point of reference; usually, either the CAD model itself or a defined Mesh is available. The Mesh is aligned to the selected reference.



Split-screen view showing the CAD model on the top view, and the mesh on the bottom view

4. If you have more than one mesh in your measurement routine, from the **Mesh** list, choose the mesh.
5. Perform the alignment:
    a. In the **Auto** section, click the **Compute** button. You should use this only when you have a full scan of the external faces of the part. This automatically performs an alignment of the Mesh to the CAD and also a refinement on the alignment as it's generating.
    b. If the auto compute does not compute a good alignment, use the **Point Pairs** area to perform a rough alignment. This brings the mesh close enough to the CAD if it's not already close. You can then refine the alignment further if needed. You should use this type of alignment if the mesh is not complete or if it contains scanned data that belongs to a fixture, table, or other similar feature.
        i. Click a desired number of points on the mesh.
        ii. Click corresponding locations on the CAD model.

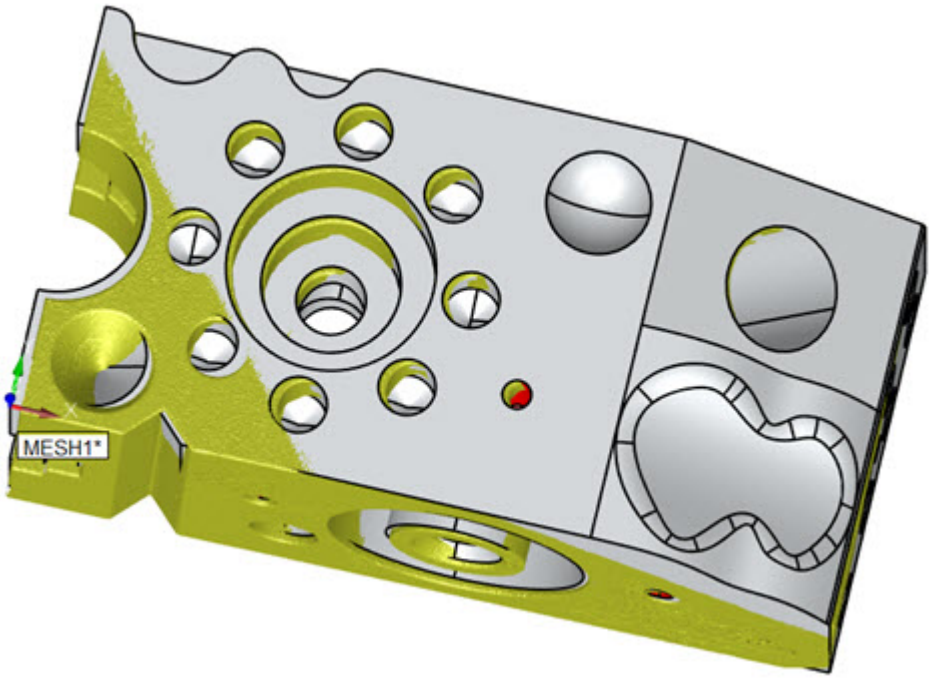


Split-screen view showing selected CAD (top) and corresponding mesh (bottom) points

iii. The more points you take around the different areas of the model and mesh, the better the rough alignment.

iv. Click **Compute** to create the rough alignment.

c. Next, use the **Refine Alignment** area whenever you want to refine your alignment. This brings the mesh closer to your CAD model. To obtain a good refined alignment, the mesh points should be close enough to the CAD points through an initial rough alignment.



A sample rough mesh-to-CAD alignment that needs a refinement

i. In the **Total Points** box, define the total number of random sample points to use in each iteration.

ii. In the **Maximum Iterations** box, define the number of iterations.

iii. In the **Maximum StdDev** box, define the maximum standard deviation for the auto alignment execution between the points in the mesh and the CAD model. When the auto alignment command is executed, if the standard deviation of the Mesh/CAD deviations is greater than the maximum value defined, you can select point pairs to get a better alignment. The default value of -1 is equivalent to an infinite allowed standard deviation.

iv. Define the maximum distance of the points from the CAD for use in the best fit routines. The default value is 0. In this case, an internal max distance based on the size of the mesh is used.

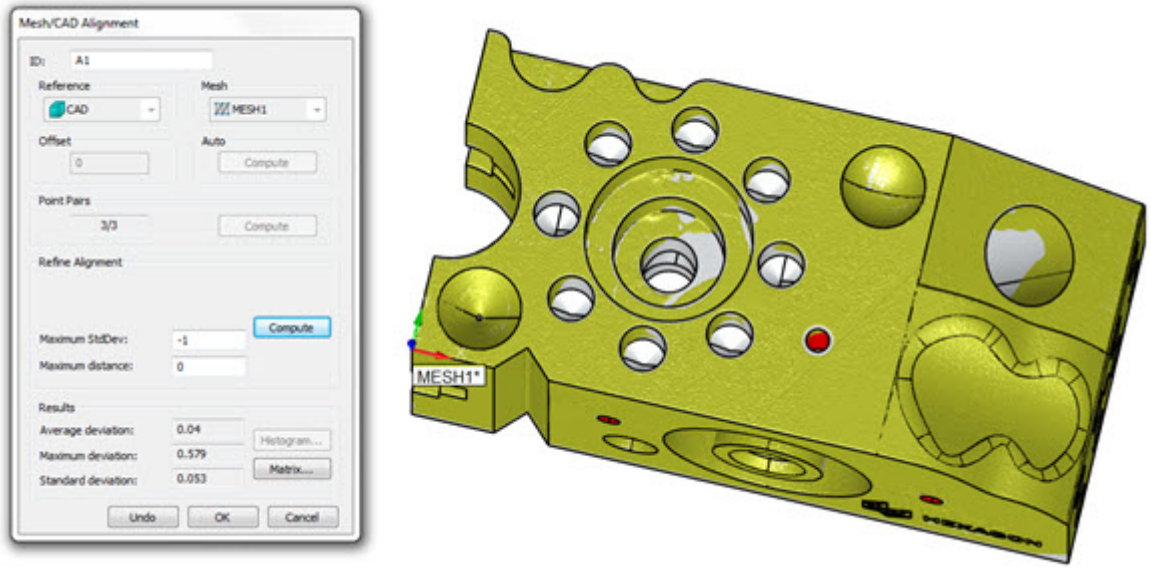
v. Click **Compute** to refine the alignment.

6. If a portion of the mesh doesn't align nicely with the CAD, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters; or you can try a different alignment.

7. If you have a surface model that represents a sheet metal part, and you want to align to the offset faces, define an **Offset** value that represents the constant thickness of the sheet metal part.

8. Use the **Results** area to see how well the mesh aligned with the CAD. Make any changes to the **Offset** or **Refine Alignment** values to improve the alignment if necessary. If any changes are made, be sure to click the **Compute** button to regenerate the alignment with the new values.

9. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the `MESHCADBF` command in the Edit window. For information on the Edit window command, see the "MESHCADBF Command Mode Text" topic.



Example of a completed mesh to CAD alignment

MESHCADBF Command Mode Text

The MESHCADBF command allows you to perform a best fit alignment of the mesh data with the CAD data.

Below is an example code snippet for a MESHCADBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
   MESHCADBF/REFINE=n1,n2,n3,SHOWALLPARAMS=TOG1,
   ROUGH ALIGNPAIR/
      THEO/<x,y,z>,<i,j,k>,
      MEAS/<x1,y1,z1>
   REF,TOG2,,
ALIGNMENT/END
```

n1 represents the offset value for applying a thickness.

n2 represents the maximum standard deviation value.

n3 represents the maximum distance value.

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/
   THEO/x,y,z,i,j,k,
   MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to `THEO/` represent the point on the CAD. The values next to `MEAS/` represent the corresponding point on the Mesh. These pairs are used to determine a rough transformation between the CAD and the Mesh which allows the Mesh to come close enough to the CAD to allow further refinements of the alignment.

TOG2 lets you choose the mesh to use for the alignment.

Creating a Mesh to Mesh Alignment

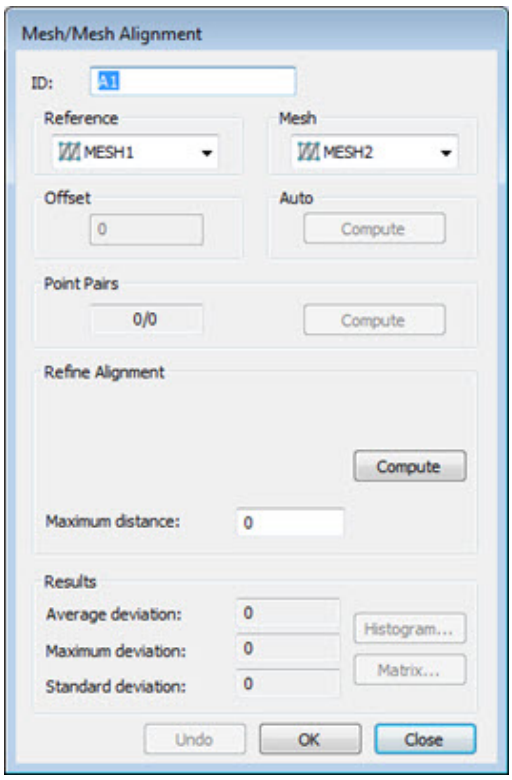
The Mesh to Mesh alignment functionality allows you to best fit align one mesh to another mesh which have been collected in two different reference frames that have some overlap. A typical example is two scans in two mesh commands, representing areas of a part that cannot be scanned in the same part orientation.

The alignment is done in two steps:

- A rough alignment, where pairs of points in the overlapping area of the two meshes are selected.
- A refined bestfit, which tries to bring the second mesh as close as possible to the reference mesh.

To create a Mesh to Mesh alignment, do the following:

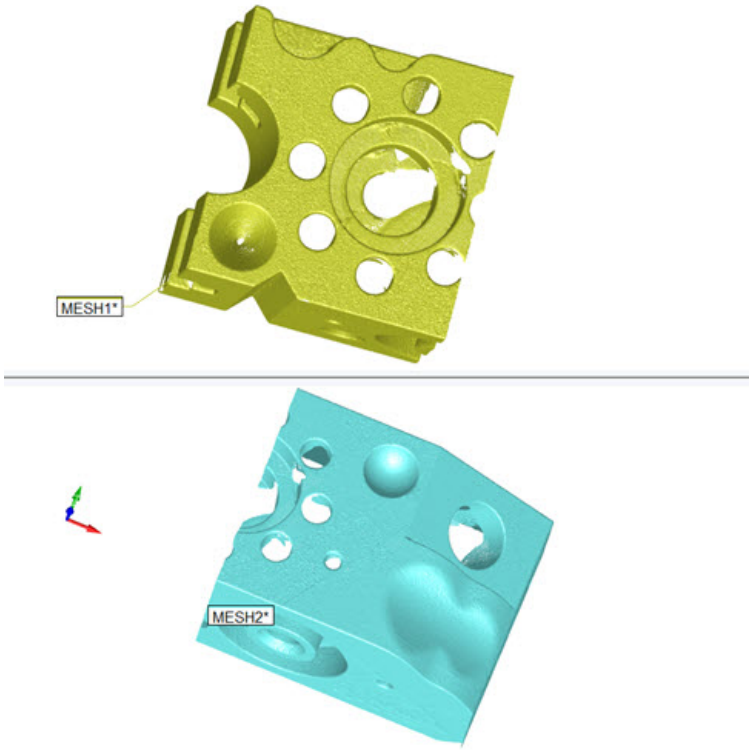
1. Ensure that you have two or more Mesh commands in the measurement routine that you are using to align. These elements are required to align the two meshes.
2. Select the **Insert | Mesh | Alignment** menu option. You can also access this dialog box by typing the `MESHMESHBF` command in the Edit window's Command Mode between the `ALIGNMENT/START` and the `ALIGNMENT/END` commands. The dialog box appears:



Mesh/Mesh Alignment dialog box

For a complete description of the dialog box, see the topic "Mesh/CAD Alignment Dialog Box Description".

3. A temporary split-screen view of the two meshes appears in the Graphic Display window. You can use this view to visually see the alignment taking place. Select the first Mesh used as a point of reference from the **Reference** drop down list.



Split-screen view showing a mesh to mesh alignment

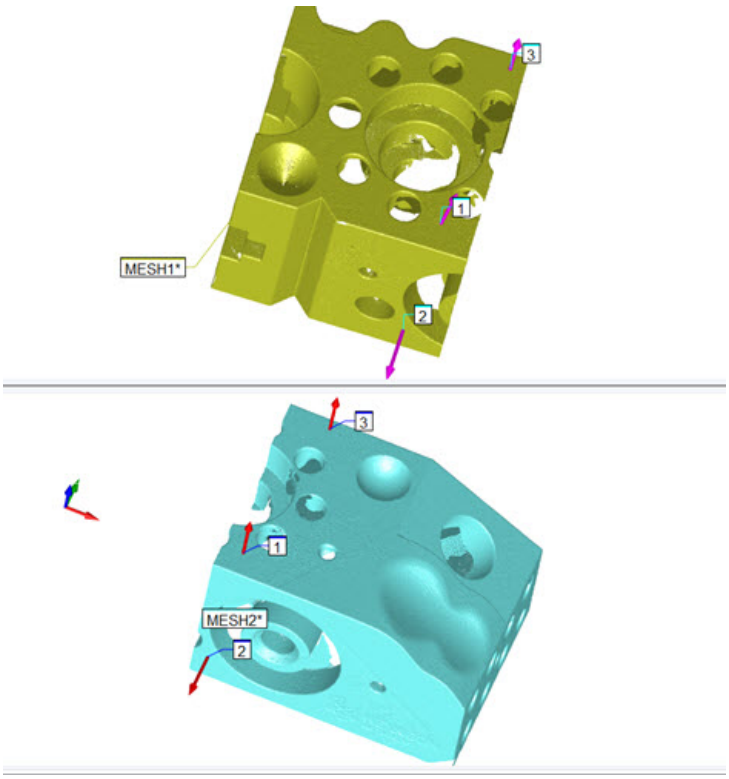
4. Use your mouse to manipulate and orient each view as needed to create the Point Pairs.

5. Perform the alignment:

    a. Click the **Compute** button in the **Auto** section. You should only use this when you have a full scan of the external faces of the part. This automatically performs an alignment of the Mesh to the reference Mesh and also a refinement on the alignment as it's generating.

    b. If the auto compute does not compute a good alignment, use the Point Pairs area to perform a rough alignment that brings the meshes close enough to each other. You can then refine the alignment further if needed. You should use this type of alignment if the mesh is not complete or it contains scanned data belonging to a fixture, table, or other similar feature.

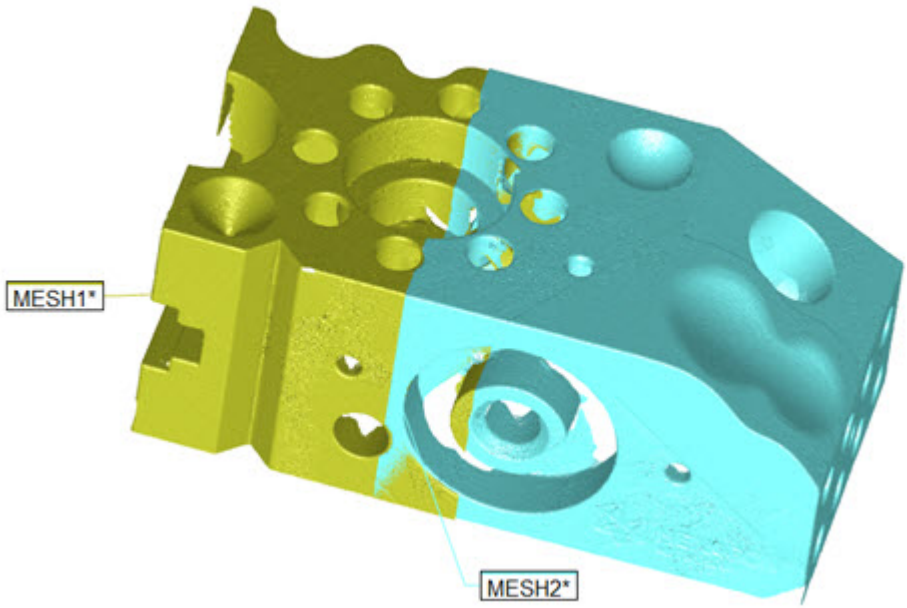
        - Click a desired number of points (at least three pairs) on each of the meshes on the overlap area. ONLY click points on the overlap area of the two meshes.



Split-screen view showing selected MESH1 and MESH2 meshes

- The more points you take around the overlap area of the meshes results in an improved alignment. Click **Compute** to create the rough alignment.

c. Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the two meshes closer to each other. In order to get a good refined alignment, the two mesh points should be close enough to each other through the initial rough alignment.



A sample rough mesh to mesh alignment that needs a refinement

i. Define the maximum distance between the points in the two meshes using the **Maximum Distance** box. The default value is 0 (zero). If the default value is used, PC-DMIS uses an internal default value related to the dimensions of the meshes.

ii. Click **Compute** to refine the alignment.

6. If a portion of one mesh doesn't align nicely with the other, you can click the **Undo** button and recompute the alignment using the same type of alignment with additional parameters, or you can try a different alignment.

7. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split-screen view and places the `MESHMESHBF` command in the Edit window. For details on the `MESHMESHBF` command, see the "MESHMESHBF Command Mode Text" topic in the PC-DMIS Laser documentation.

MESHMESHBF Command Mode Text

The `MESHMESHBF` command allows you to perform a best fit alignment of the reference mesh with a second mesh.

Below is an example code snippet for a `MESHMESHBF` alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
   MESHMESHBF/REFINE,SHOWALLPARAMS=TOG1,
   ROUGH ALIGNPAIR/
      THEO/<x,y,z>,<i,j,k>,
      MEAS/<x1,y1,z1>
   REF,TOG2,TOG3,,
ALIGNMENT/END
```

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
   ROUGH ALIGNPAIR/
      THEO/x,y,z,i,j,k,
      MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to `THEO/` represent the point for the reference Mesh. The values next to `MEAS/` represent the corresponding point on the second Mesh. These pairs are used to determine a rough transformation

between the reference Mesh and the second Mesh which allows the two meshes to come close to allow further refinements of the alignment.

TOG2 determines the reference Mesh used for aligning to the second Mesh.

TOG3 determines the second Mesh used for the alignment back to the reference Mesh.

Receive a Mesh from OptoCat

Use the **Receive a mesh from OptoCat** button ( ) on the **Mesh** toolbar to place PC-DMIS in a state where it is waiting and ready to receive a mesh from the OptoCat application.

When mesh data is received:

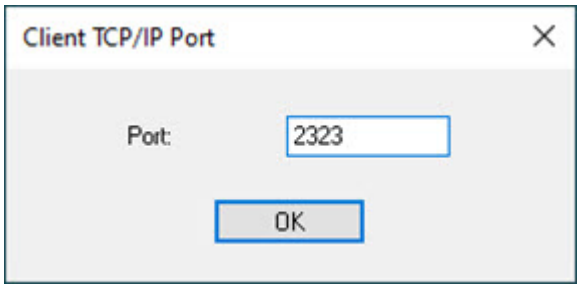
- If the PC-DMIS measurement routine already contains a mesh command, the existing mesh data is replaced with the new mesh data.
- If the PC-DMIS Inspection plan does not contain a mesh command, a mesh command containing the new mesh data is inserted into the measurement routine.
- After the received mesh data is inserted into the measurement routine, the measurement routine is automatically executed.

When clicked ON, the **Receive a mesh from OptoCat** button has a darker background color: .

Click the button to toggle this function ON and OFF.

To use this function:

1. Open the measurement routine you are importing the OptoCat Mesh data into.
2. From the **Mesh** toolbar (**View | Toolbars | Mesh**), click the **Receive a mesh from OptoCat** button. The **Client TCP/IP Port** dialog box appears.



3. Update the **Port** field if necessary. The port assignment on the computer must match the port assignment of the OptoCat application.

4. Click **OK**. PC-DMIS is ready to receive mesh data from the OptoCat application.

Receive a Pointcloud or Mesh from the HHScan Application

PC-DMIS can communicate with the HHScan application when you use the Hexagon AtlaScan Max or MarvelScan hand-held scanning devices shown below.



AtlaScan Max sensor (left) and MarvelScan sensor (right)

To place PC-DMIS in a state where it is waiting and ready to receive pointcloud or mesh data from the HHScan application, click the **Receive data from HHScan** button on the **Pointcloud** or **Mesh** toolbar.

To use this functionality to receive pointcloud data, your PC-DMIS license must include the **BIGCOP** option. To receive mesh data, your license must include the **MESH** option.

When PC-DMIS receives data:

- If the currently running measurement routine already contains a Pointcloud or Mesh command, PC-DMIS replaces the existing data with the new data.

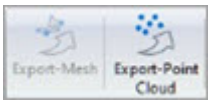
- If the currently running measurement routine does not contain a Pointcloud or Mesh command, PC-DMIS inserts a command into the measurement routine and populates it with the new data.
- After PC-DMIS inserts the received data into the measurement routine, PC-DMIS automatically executes the measurement routine.

When the **Receive data from HHScan** button is in the On state, it has a darker color  . Also, the status bar in the lower-left of the PC-DMIS screen shows the text "Receive data from HHScan". Once the button is in the On state, PC-DMIS monitors the "*C:\Users\Public\Documents\Hexagon\ZGSensor*" folder for any new files received from the HHScan application.

You can click the **Receive data from HHScan** button to toggle this function On and Off.

To use this function:

1. Open the measurement routine into which you want to import the pointcloud or mesh data from the HHScan application.
2. Click the **Receive data from HHScan** button to place PC-DMIS in the state to be ready to receive data from the HHScan application.
3. Use the HHScan application to generate the data.
4. Export the data from the HHScan application. You can perform the data export from the HHScan application in any of these ways:
    - From the **Export** section on the **Home** toolbar tab, click the **Export-Mesh** or **Export-Point Cloud** button to assign the destination folder and manually export the data.

      Export-Mesh and Export-Point Cloud buttons from the HHScan application

      > You must set the destination folder to "*C:\Users\Public\Documents\Hexagon\ZGSensor*" for PC-DMIS to detect and import the data.

    - You can have the HHScan application automatically export the data to the previously assigned folder location when the scan operation is completed. To do this, from the **Scan** section of the **Home** toolbar tab, click the **Scan-Surface** or **Scan-Point Cloud** button a second time to end the current data-collection session.



Scan-Surface and Scan-Point Cloud buttons from the HHScan application

- From the **Export** section of the **Advanced Features** toolbar tab, click the **Hexagon** button. This button is pre-configured to send data to the "*C:\Users\Public\Documents\Hexagon\ZGSensor*" folder.



Hexagon Export button from the HHScan application

While PC-DMIS is in the state to be ready to receive data from the HHScan application, it detects any new data that arrives in the "*C:\Users\Public\Documents\Hexagon\ZGSensor*" folder. Whenever data arrives, PC-DMIS imports the data into the open measurement routine as described.

HxGN Robotic Automation

HxGN Robotic Automation is a Hexagon application that communicates with PC-DMIS. When you connect PC-DMIS to a HxGN Robotic Automation system, you can use the HxGN Robotic Automation toolbar to export measurement routines and inspection plans to HxGN Robotic Automation.

The topics in this section are:

- Introduction to HxGN Robotic Automation
- PC-DMIS Workflow Using HxGN Robotic Automation
- Configuring HxGN Robotic Automation with the Connector Application

Introduction to HxGN Robotic Automation

HxGN Robotic Automation is a software tool for automated part scanning and inspection that you can use to:

- Model and create a virtual measurement cell
- Generate optimal robot scanning paths for measurements through robot simulation
- Execute robotic measurement routines
- Acquire scanned data

You can send scanned data to PC-DMIS to:

- Obtain measurement results and perform data analysis
- Generate reports

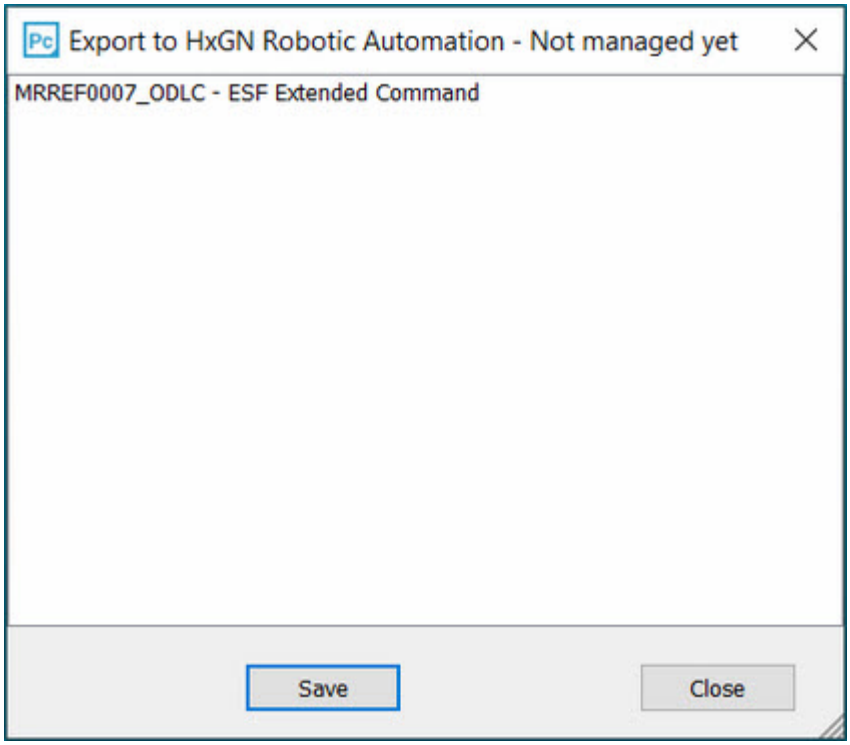
PC-DMIS Workflow Using HxGN Robotic Automation

> This workflow assumes you have the ROY-AUTOMATE license enabled.

To use PC-DMIS with the HxGN Robotic Automation application:

1. Create a PC-DMIS measurement routine using a laser probe. You can either manually create the measurement routine or import a CSV version with the CSV-Converter application.
2. On the **HxGN Robotic Automation** toolbar (**View | Toolbars | HxGN Robotic Automation**), click the **Export to HxGN Robotic Automation** button ( ) or select the **File |Export | HxGN Robotic Automation** menu option.

   For features not yet supported, HxGN Robotic Automation displays the **Export to HxGN Robotic Automation - Not managed yet** dialog box to list them:



   Click **Save** to save the report of unsupported features as a text file.

3. From the **HxGN Robotic Automation** toolbar, click the **TCP/IP Pointcloud Server receive data** button () to open the communication channel between PC-DMIS and HxGN Robotic Automation.

4. From the HxGN Robotic Automation application, create and execute the scanning program. For details on how to create and execute scanning programs in the HxGN Robotic Automation application, see the HxGN Robotic Automation documentation.

   PC-DMIS receives the scanned data back from the HxGN Robotic Automation application, and automatically executes the measurement routine.

How Native CAD Files are Handled in Robotic Automation

When you transfer a PC-DMIS measurement routine to Robotic Automation the very first time, PC-DMIS also transfers the original CAD model. Robotic Automation uses the CAD model for part planning.

PC-DMIS always stores CAD files in a default location, typically "C:\Users\Public\Documents\Hexagon\PC-DMIS\2026.2\CAD\".

If, for some reason, Robotic Automation does not find the CAD file in the default location, PC-DMIS displays a message similar to this:



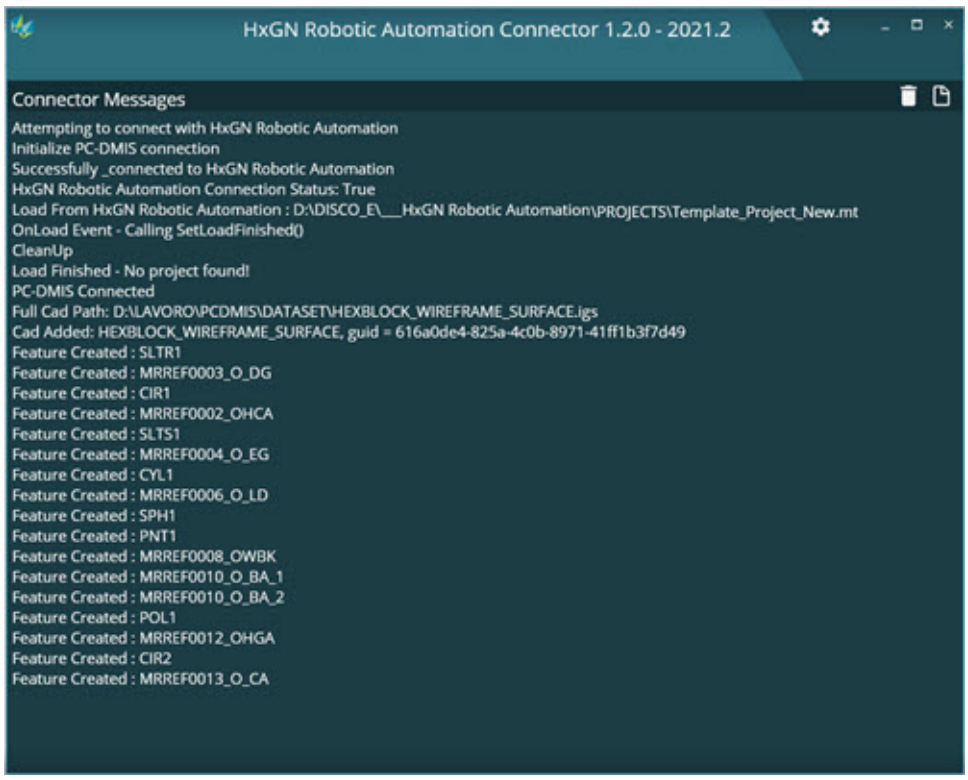
PC-DMIS displays this message to inform you that the native CAD model file associated with the current measurement routine was not found in the default location and needs to be copied into that folder.

Configuring HxGN Robotic Automation with the Connector Application

You configure the communication between PC-DMIS and HxGN Robotic Automation with the Connector application.

To do this:

1. With the HxGN Robotic Automation application running, click the left-most green **Connector Application** button () in the bottom-right of the screen. This opens the **HxGN Robotic Automation Connector** application's dialog box.

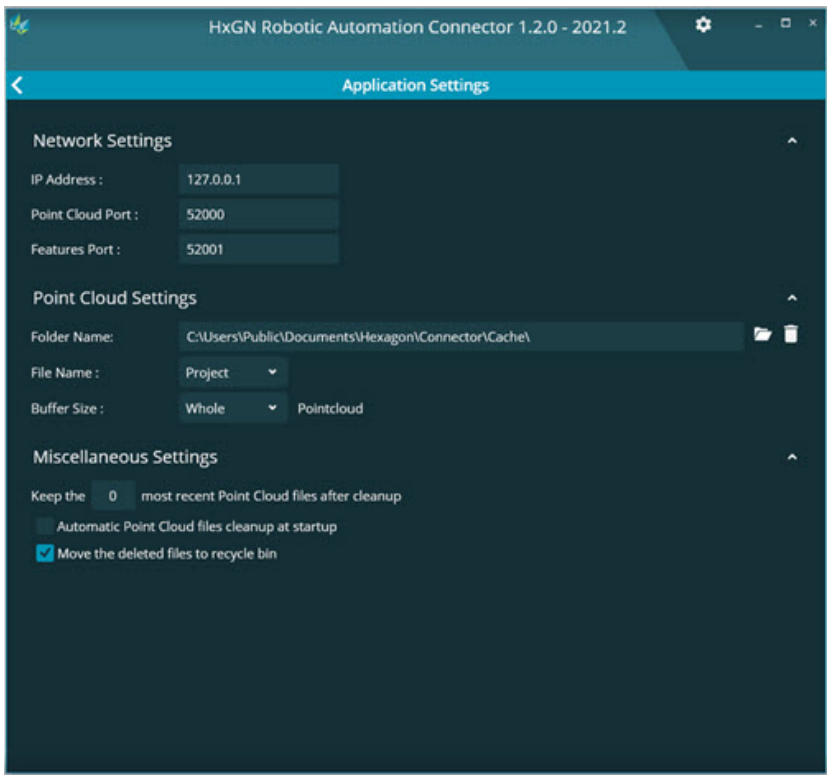


HxGN Robotic Automation Connector application dialog box

The Connector application displays the log of communication event messages.

- You can click the **Clear Log** button to permanently delete all log messages.
- You can click the **Export Log** button to copy the list of messages into a log file.
- You can click the **Clean Pointcloud Folder** button to delete the pointcloud data files in the folder where HxGN Robotic Automation stores the pointcloud data from PC-DMIS. You can define how the Connector application does this in the **Application Settings** dialog box, available from the **Setup** button ( ).

2. To configure or review the Connector application settings, click the **Setup** button ( ) in the upper-right of the dialog box.



HxGN Robotic Automation Connector dialog box - Application Settings

The **Application Settings** dialog box has these areas:

Network Settings - This area defines these settings:

IP Address - This box defines the communication address of the computer that runs the HxGN Robotic Automation application.

Pointcloud Port - This box defines the port number that PC-DMIS uses to read in the pointcloud data from the HxGN Robotic Automation application.

Features Port - This box defines the port number that the HxGN Robotic Automation application uses to read in the feature data from the PC-DMIS application.

Pointcloud Settings - This area defines these settings:

Folder Name - This box defines the location where HxGN Robotic Automation saves all pointcloud files. You can type the location directly into the box or use the **Folder** button ( ) to open a dialog box and navigate to the folder location of your choice.

File Name - This box defines the prefix of the HxGN Robotic Automation pointcloud file name. The default is the project name, but you can select the **Custom** option from the **File Name** list and type an alternate prefix for your file names.

Buffer Size - This option allows you to select how the Connector application sends the pointcloud to PC-DMIS for processing.

- If you select **Whole** from the list, the Connector application sends the entire pointcloud to the PC-DMIS application.
- If you select **Packets** and then type the number of stripes in the **Stripes** box, the Connector application sends the pointcloud in small chunks to the PC-DMIS application. Each chunk (or packet) contains the entered number of stripes. This continues until the entire pointcloud is sent.

Run Time Drawing check box - When you select this check box, PC-DMIS shows each pointcloud packet in the Graphic Display window as soon as it is sent to the PC-DMIS application.

Miscellaneous Settings - You can use this area to set the number of pointcloud files you want HxGN Robotic Automation to keep when you click the **Clean Pointcloud Folder** button.



Suppose you set this value to 1 and then you click the **Clean Pointcloud Folder** button. HxGN Robotic Automation deletes all but the most recent pointcloud files from the folder. If you set this value to 2 and you click the **Clean Pointcloud Folder** button, HxGN Robotic Automation deletes all but the two most recent pointcloud files from the folder, and so on.

- If you select the **Automatic Pointcloud files cleanup at startup** check box, HxGN Robotic Automation cleans the pointcloud folder based on the value you set each time you restart the application. This is permanent unless you select the **Move the deleted files to recycle bin** check box.
- If you do not want to permanently delete the pointcloud files when you click the **Clean Pointcloud Folder** button, you can select the **Move the deleted files to recycle bin** check box. This saves all the deleted pointcloud files into the Recycle Bin instead of permanently deleting them from your system.

3. Click the **Back** button () to return to the log screen or click the close button (**X**) in the upper-right of the window to close the **HxGN Robotic Automation Connector** application.

Glossary

Annotation
An annotation is a way to display the deviation for a specific location on a surface colormap with its associated color.

CCD
Charge Coupled Device - This is one of the two main types of image sensors used in digital cameras.

CI
Condition Indicator - This number indicates the quality of the points distribution for surface points.

Cloud of Points
A term used to describe a Pointcloud. A Pointcloud is a collection of data points used to define a feature on a CAD model.

COP
The COP (Cloud of Points) command is a container for XYZ coordinate data. The data can be input from an external file, or it can come directly from a laser sensor through the referring scan command(s). See Pointcloud.

Exposure
This parameter controls the exposure of the laser sensor.

Gauge side point
In a Flush and Gap Auto feature, this is the point on the gauge surface side indicating where the flush should be measured. (also called the gauge point)

LWM
Laser Wrist Map

Master side point
In a Flush and Gap Auto feature, this is the point on the master side surface where the flush is to be measured.

Mesh
A mesh is a set of vertices and triangles that are combined using a best fit algorithm to represent a 3D part shape.

milli-pixel
1 milli-pixel = 0.001 pixel

Overscan
This parameter controls how far beyond the nominal feature’s dimensions the probe will scan along both the major and minor axes of the feature.

Pointcloud
A Pointcloud is a collection of data points used to define a feature on a CAD model.

Row Overlap
This parameter controls how far each pass will overlap with the previous pass.

Sensor Frequency
This parameter controls the internal sensor frequency of the probe. The value that appears is the sensor pulses per second.

Surface CAD Model

A surface CAD model only has surfaces, and it does not create a solid. Some examples of this would be a plane feature, or a cylinder surface where there is no closed volume.

Index

2

2D Features 100
  Max Incidence Angle 100
2D Radius Gage 270**,** 269
  Dialog Box 270

3

3D Features 94
  Max Incidence Angle 94

A

Aligning Pointclouds 276**,** 277
Alignment 277
  Pointcloud to CAD 277
Angle Filter 100**,** 94
Angle Option 100**,** 94
Animation Parameters 161
Annotation Opposing Points 172
Annotation
  Opposing Points 172
  Show Opposing Points 172
Annotations 172
  Thickness Colormap 172
Auto Circle Feature 296
Auto Feature (Laser) 100**,** 94**,** 308**,** 306**,** 318**,** 307**,** 306**,** 316**,** 317
  Advanced Measurement Options 310
  Best Fit Math Type 310
  Command Buttons 311
  Feature Properties 308
  High Point 328, 328
  Measurement Properties 309
  Relative To 310
  Scan 297
Auto Feature Extraction 100**,** 94**,** 299**,** 293**,** 307
  with CAD Data 307
Auto Features 12
  M&H LS-R Laser Sensor 12

B

Big COP 132
Boundary Points 392
  Adding and Deleting 395
  Clearing 395
  Editing 394
  Generate 395

Setting by Typing 393
Setting Using the CAD Data Method 394

C

Calculation Methods 318**,** 315**,** 316**,** 317
Calibrate 10
Calibration Sphere
Manually Bisecting 56
Caliper 255**,** 266**,** 255
End Point 266
Mid Point 266
Start Point 266
Caliper End Point 266
Caliper Mid Point 266
Caliper Start Point 266
Circle 335**,** 335**,** 334**,** 333
Circle Feature 296
Color Bar Levels Area 175
Color Bar Profiles Area 177
Color Scale Area 176
Colormap 170**,** 167
Pointcloud Thickness 167
Thickness 170, 173, 167
Command Mode Text 328
High Point 328
Compare to CAD Pointcloud Thickness Colormap 170
Compare to CAD Thickness Colormap 170
COP 307**,** 135**,** 68**,** 132**,** 161
Big 132
Graphical Representation 135
Small 132
COP Command 139**,** 161
COP/OPER Command 161
BOOLEAN 254
CLEAN 235
EXPORT 245
IMPORT 251
Pointcloud EXPORT 245
Pointcloud IMPORT 251
PURGE 241
RESET 249
SELECT 186
COPALIGN Command 283**,** 288**,** 277
COPCADBF Command 283**,** 277
COPCOPBF Command 288**,** 277
Creating a Pointcloud to Pointcloud Alignment 277
Cut Plane Vector 397
CWS Parameter 102
Cylinder 370**,** 369**,** 366

D

Data Filtering 145
Depth Option 296
    Circle Feature 296
Dialog Box 160
Dialog Box/Simulate COP from CAD 160
Dialog Box/Simulate Pointcloud from CAD 160
Display in Scene Area 178

E

Edge Point 324
End Touch Vector 397
Example File Format
    Thickness Colormap 173
Exclusion Plane Section 155
Execution Mode 112
Export COP/OPER 245
Extended Surface Point Calculation Method 318
Extracting a Surface Point from a Mesh 299
Extracting with CAD data 307

F

Feature Extraction 299
Feature Extraction Angle Filter 100**,** 94
Feature Extraction Angle Option 100**,** 94
Filters 142
Flush and Gap 353**,** 351
Freeform Advanced Scan 410
Front Facing 454**,** 186

G

Gage 270**,** 269**,** 255
    Caliper 255
Gages 270**,** 269**,** 255
    Caliper 255
Getting Started 10
Graphical Overlays 117
Grid Advanced Scan 412

H

High Point 328
    Command Mode Text 328
High Point Path 328
HP-L Sensor 68
HP-L-10.10 Sensor 31
    Live View 31
HxGN Robotic Automation 481**,** 131**,** 481**,** 482
    Introduction 481
    Toolbar 131
    Workflow 482

HxGN Robotic Automation Workflow 482

I

Initial Touch Vector 397
Initial Vectors 405

L

Laser Attributes 9
Laser Auto 369**,** 318**,** 317
Circle 306
Command Mode Text 334, 369, 324, 353, 330, 343, 379, 314
Cylinder 306
Fractions 335
High Point 328, 328
High Point Path 328
Parameters 333, 366, 351, 342, 379
Paths 335, 370, 331, 343, 380, 344, 315
Round Slot 306
Sphere 306
Square Slot 306
Laser Data Collection Settings 155**,** 142**,** 143
Exclusion Plane Section 155
Profiles Section 143
Laser Filtering Properties 73
Point Reduction Filter 73
Laser High Point Auto Feature 328**,** 328
Path 328
Laser probe Auto feature 315
Laser Probe Toolbox 100**,** 68**,** 77**,** 61
Acceptance Angle Filter tab 77
Laser Clipping Region Properties 78
Laser Filtering Properties 100, 94, 71
Laser Scan Properties 68, 64
Laser Scan Properties:Exposure 68
Laser Scan Properties:Gain 68
Laser Scan Properties:Sensor Frequency 66
Position Probe tab 62
Position Probe tab:Controls 63
Position Probe tab:Positioning Your Laser Sensor 63
Laser Scan Properties 68
Gain 68
Laser Surface Point 318**,** 315**,** 316**,** 317
Calculation Methods 318, 315, 317
Laser View 114
Levels Area 175
Linear Open Advanced Scan 400
Parameters 402
Live View 31
HP-L-10.10 Sensor 31

M

M&H LS-R Laser Sensor 12**,** 11**,** 10**,** 12
Auto Features 12
Operation Requirements 11
Scanning 11
Max Incidence Angle 100**,** 94
Measure Laser Probe Options 54
Mesh 255**,** 299**,** 465**,** 455
Alignment 465, 465
Auto Feature Extraction 299
Extracting a Surface Point 299
Thickness Colormap 455
Mesh Alignment 465**,** 465
Mesh Alignment Dialog Box 465
Mesh Operator 449**,** 454**,** 455
RESET 449
Mesh Reset 449
Mesh Select Operator 454
Front Facing 454
Mesh Thickness Colormap 455
Mesh Toolbar 465**,** 128
Alignment 465, 465
MESH/OPER Command 454**,** 455
SELECT 454
MESHALIGN Command 472
MESHCADBF Command 472
MESHMESHBF Align Command 477
MESHMESHBF Command 477

P

Patch Advanced Scan 403
New Line 391
Parameters 405
PC-DMIS Integration Requirements 11
M&H LS-R Laser Sensor 11
M&H LS-R Laser Sensor:Operation 11
Operation 11
Scanning 11
Perimeter Advanced Scan 406
Parameters 409
Planar calculation method 315**,** 316
Plane 331**,** 330
Point Reduction Filter 73
Pointcloud 307
Colormap 167
Pointcloud Alignment 276**,** 277**,** 132
Pointcloud Alignment Dialog Box 277
Pointcloud Colormap 170
Thickness 170
Pointcloud Colors 120
Pointcloud Export COP/OPER 245

Pointcloud Import COP/OPER 251
Pointcloud Mesh 255
Pointcloud Operator
Animation Parameters for Pointcloud Simulation 161
Boolean 254
Clean 235
Export 245
Import 251
Manipulating 165
Pointcloud Export 245
Pointcloud Import 251
Purge 241
Reset 249
Select 186
Surface Colormap 175
Toolbar 121
Pointcloud Select/Front Facing 186
Pointcloud Server 121
Pointcloud Thickness Colormap 170**,** 167
Compare to CAD 170
Pointcloud to Pointcloud Alignment 277
Pointcloud Toolbar 277**,** 121
Pointclouds 255**,** 178**,** 307**,** 142**,** 135**,** 121**,** 68**,** 132**,** 161
Graphical Representation 135
Simulate 161
Simulate Function 161
Simulate Function:Animation Parameters 161
Simulate:Animation Parameters 161
Probe Calibration 12
M&H LS-R Laser Sensor 12
Profiles Area 177
Profiles Section 145**,** 143
Data Filtering 145

Q

QuickCloud Toolbar 255**,** 128**,** 121
QuickFeature 307
Extracting 307
QuickFeature Extraction 307
QuickFeature Implementation 306
QuickMeasure Toolbar 121

R

Ring Band 87
Round Slot 343**,** 343**,** 342

S

Scan Line Indicator 115
Scanning 382**,** 67**,** 381
Auto Features 297
Boundary Points 392

CAD Controls 383
Colors 120
Common Functions 382
Freeform 410
Graphical Representation of Vectors 396
Grid 412
ID 383
Initial Vectors 405
Linear Open 400
Measure 398
Patch 403
Perimeter 406
Point Cloud Reference Feature 398
Row Overlap 67
Scan Parameters 383
Scan Type 383
Setting the Machine Speed 415
Speeds 415
To Points conversion 385
Vectors Area 396
Scanning with the M&H LS-R Laser Sensor 11
Select COP/OPER 186
Sensor 10
HP-L-10.10 31
M&H LS-R 11, 10
M&H LS-R:Operation Requirements 11
M&H LS-R:Scanning 11
Sequential Execution Mode 112
Show Annotation Opposing Points 172
Simulate 161
Animation Parameters 161
Simulate COP from CAD 160
Simulate Pointcloud 161
Animation Parameters 161
Function 161
Simulate Pointcloud from CAD 160
Simulate Pointcloud Function 161
Animation Parameters 161
Small COP 132
Sound Events 113
Sphere 380**,** 379**,** 379
Spherical Calculation Method 315**,** 317
Spherical Surface Point Calculation Method 317
Spline Points 391
Calculation Type 392
Curve Type 391
Increment 392
Point Spacing Type 392
Weight 392
Square Slot 344**,** 343**,** 342
Surface Colormap 176**,** 175

Surface Point 315**,** 318**,** 317**,** 314
SurfacePointType settings entry 315

T

THEO COP 160
THEO COP from CAD 160
Theoretical Points 389
Deleting 390
Editing 390
Manual Points 391
Read File 390
Thickness Colormap 170**,** 173**,** 167**,** 172**,** 171
Annotation 172
Annotations 172
Compare to CAD 170
Example File Format 173
Mesh 455
Show in Report 171
Thickness Colormap Annotations 172
Toolbar 131**,** 128**,** 121**,** 121
HxGN Robotic Automation 131
Mesh 465, 128
Mesh Alignment 465, 465
QuickCloud 255, 128, 121
QuickMeasure 255, 121

U

Using the Depth Option 296
Auto Circle Feature 296
Using the Simulate Pointcloud Function 161
Animation Parameters 161

V

Vectors 402