PC-DMIS Planner Manual

For PC-DMIS Planner 2013 MR1

By Wilcox Associates, Inc.

Copyright © 1999-2001, 2002-2013 Hexagon Metrology and Wilcox Associates Incorporated. All rights reserved.

PC-DMIS, Direct CAD, Tutor for Windows, Remote Panel Application, DataPage, and Micro Measure IV are either registered trademarks or trademarks of Hexagon Metrology and Wilcox Associates, Incorporated.

SPC-Light is a trademark of Lighthouse.

HyperView is a trademark of Dundas Software Limited and HyperCube Incorporated.

Orbix 3 is a trademark of IONA Technologies.

I-DEAS and Unigraphics are either trademarks or registered trademarks of EDS.

Pro/ENGINEER is a registered trademark of PTC.

CATIA is either a trademark or registered trademark of Dassault Systemes and IBM Corporation.

ACIS is either a trademark or registered trademark of Spatial and Dassault Systemes.

3DxWare is either a trademark or registered trademark of 3Dconnexion.

lp_solve is a free software package licensed and used under the GNU LGPL.

PC-DMIS for Windows uses a free, open source package called lp_solve (or lpsolve) that is distributed under the GNU lesser general public license (LGPL).

```
lpsolve citation data
----------------------
Description: Open source (Mixed-Integer) Linear Programming system
Language: Multi-platform, pure ANSI C / POSIX source code, Lex/Yacc based parsing
Official name: lp_solve (alternatively lpsolve)
Release data: Version 5.1.0.0 dated 1 May 2004
Co-developers: Michel Berkelaar, Kjell Eikland, Peter Notebaert
License terms: GNU LGPL (Lesser General Public License)
Citation policy: General references as per LGPL
Module specific references as specified therein
You can get this package from:
http://groups.yahoo.com/group/lp_solve/
```

PC-DMIS for Windows uses this crash reporting tool:

“CrashRpt”

Copyright © 2003, Michael Carruth

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

Neither the name of the author nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Table of Contents

PC-DMIS Planner 1
- PC-DMIS Planner 2013 MR1: Introduction 1
- Using PC-DMIS Planner: An Overview of Essential Steps 2
- Understanding the User Interface 3
- Setting Preferences 4
  - Setting Up the View 4
  - Setting Up CAD and Graphics 7
  - Defining Rotation Options 29
  - Defining GD&T Colors 32
  - Setting the Measurement Units 32
  - Setting Fonts 32
  - Setting Planner Options 33
  - Modifying Default Parameters and Rules 34
  - Set as Priority 38
- Importing and Exporting Files 40
  - Importing CAD Data 40
  - Importing a DXF/DWG File 42
  - Importing a SolidWorks File 49
  - Importing an Inspection Plan 55
  - Importing an XYZIJK File into an Inspection Plan 56
  - Using Direct CAD Interfaces 56
  - Exporting an Inspection Plan 56
  - Using Command Line Options 57
  - Importing a CSV File into Planner 58
- Working with the Graphics Display Window 60
  - Changing the Graphic Modes 60
  - Changing the Graphic Views 61
  - Repainting the Screen 61
  - Changing Screen Modes 62
  - Text Box Mode 65
  - Switching Between Curve and Surface Modes 65
  - Highlighting Surfaces and Curves with the Mouse 66
  - Working with CATIA Captures 66
  - Working with Assemblies of Parts 67
  - Editing CAD Vectors 78
  - Viewing CAD Nominal Information 79
  - Editing Screen Colors 80
  - Working with CAD Levels 81
  - Capturing the Graphic Display window 86
  - Setting OpenGL Options 87
  - Modifying Display Symbols 87
  - Displaying Mouse Coordinates in the Screen Counters Window 88
  - Centering the Graphics Feature View 89
  - Viewing the Graphic Display in Demo Mode 89
- Working with Planner CAD Files 89

Working with Inspection Plans 90
Creating Auto Features 112
Creating a Freeform Scan Feature 119
Creating a "Perimeter Points" Feature 120
Constructing Features 125
Adding Dimensions to Features 131
Creating a Location Dimension 131
Creating a Distance Dimension 133
Creating an Angle Dimension 134
Creating Feature Control Frame Dimensions 135
Adding Features from GD&T Callouts 143
Creating Alignments 145
Creating an Iterative Alignment 147
Inserting Comments 149
Working with Section Cuts 150
Description of the Section Cut Dialog Box 150
Creating a Section Cut 154
Creating Slot Edge Points 156
Creating Tab Vector Points 159
ISO Dimension Tolerances 162
Adding GD&T CAD Callouts 163
Defining Material Thickness 167
Viewing Toolbars and the Status Bar 168
Constructed Features Toolbar 168
Dimensions (CAD GD&T) Toolbar 169
Dimensions (Plan) Toolbar 169
Graphic Modes Toolbar 170
Graphic Items Toolbar 172
Graphic Views Toolbar 172
Main Toolbar 173
Plans Toolbar 174
Auto Features Toolbar 175
Tools Toolbar 176
Status Bar 177
Using Shortcut Menus 177
Shortcut Menus on the Toolbar Area 177
Shortcut Menus for Feature IDs or CAD GD&T Callouts 178
Shortcut Menus in the Plan List 180
Configuring 3Dconnexion 3D Mouse Devices 182
Editing the 3DxWare Configuration File 182
3Dconnexion 3D Mouse Device Modes 186
Updating the Software 187
Index 191
Glossary 193

PC-DMIS Planner

PC-DMIS Planner 2013 MR1: Introduction

Important: These CAD DCI or DCTs are only supported in the 32-bit (x86) version of the software:

- CATIA5
- DXF Translator
- PROE
- NX

PC-DMIS™ Planner is an independent software application that works in conjunction with your CAD software package and PC-DMIS to aid the Design Engineer or Quality Engineer in designing a plan, called an inspection plan, that determines which features of the geometry and GD&T/dimension to use in the inspection process.

Unlike its predecessor, Inspection Planner, PC-DMIS Planner 2010 or later is a stand-alone package that can create inspection plans that are used with PC-DMIS; however, PC-DMIS is no longer required to use PC-DMIS Planner.

PC-DMIS Planner uses the PC-DMIS CAD engine, providing access to IGES, STEP, and the other translators, as well as PC-DMIS's supported Direct CAD Interfaces™. This gives you wide coverage of available CAD engines and systems. You can save the inspection plan directly into a .cad file or you can output it to a text file (an .ip file). You can then use PC-DMIS to automatically generate a part program from an imported inspection plan. This part program can be used for either a CMM or CNC machine tool for in-process inspection.

The topics below will walk you through the steps to create an inspection plan from your CAD data and then store that plan in either a .cad file or exported as a text file for use with PC-DMIS.

In the standard product, PC-DMIS Planner can import the following file types: IGES, STEP, VDAFS, XYZ, IP, XML and CSV.
Additionally, PC-DMIS Planner can optionally support the following, which must be enabled on your portlock settings:
Direct CAD Translators (DCT): CATIA v4, CATIA v5, CATIA v6, JT, Parasolid, Pro/E, Unigraphics and SolidWorks.
Direct CAD Interfaces (DCI): CATIA v5, Pro/E, SolidWorks and Unigraphics.
For information on your portlock settings, contact a Hexagon representative.

This documentation covers these topics:

- Using PC-DMIS Planner: An Overview of Essential Steps
- Understanding the User Interface
- Setting Preferences
- Importing and Exporting Files
- Working with the Graphic Display window
- Working with CAD Files
- Working with Inspection Plans
- Using the Change Manager
- Creating Auto Features
- Constructing Features
- Adding Dimensions to Features
- Creating Alignments
- Inserting Comments

- Working with Section Cuts
- Creating Slot Edge Points
- Creating Tab Vector Points
- Adding GDT CAD Callouts
- Adding Features from GD&T Callouts
- ISO Dimension Tolerances
- Defining Material Thickness
- Viewing Toolbars and the Status Bar
- Using Shortcut Menus
- Configuring 3Dconnexion 3D Mouse Devices
- Updating Software

For information on working with inspection plans inside PC-DMIS, consult the "Using Inspection Plans in PC-DMIS" topic in the PC-DMIS Core documentation.

Using PC-DMIS Planner: An Overview of Essential Steps

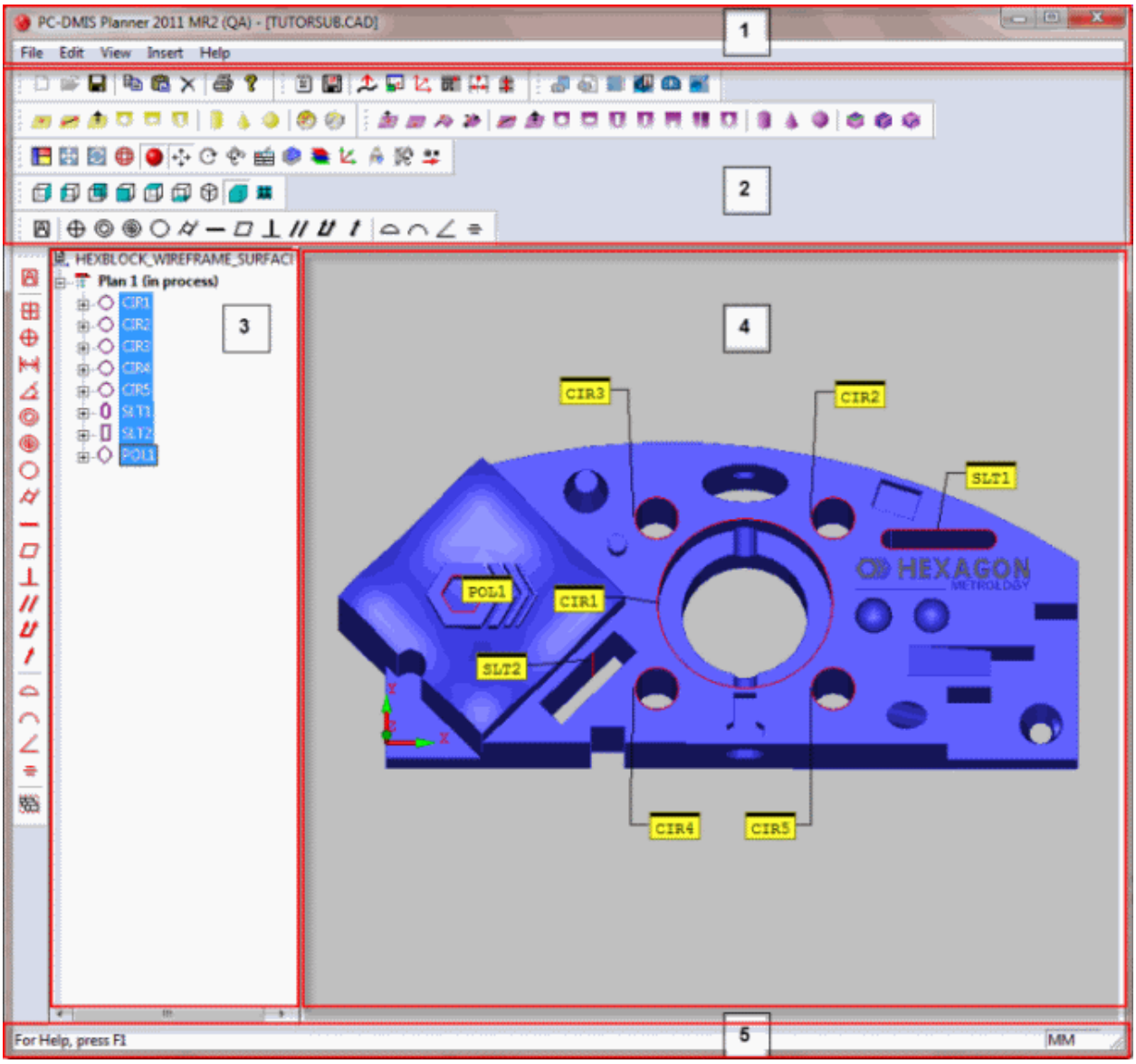
This topic provides a basic overview of steps to take inside PC-DMIS Planner so that you can end up with an inspection plan that can then be used inside of PC-DMIS for Windows.

1. Install PC-DMIS Planner - installation of Planner is covered in the Planner Readme documentation.
2. Create a new .cad file in Planner by selecting **File | New** or by clicking the **New** icon from the **Main** toolbar. PC-DMIS stores your inspection plans in this .cad file. For information on .cad files, see "Working with .Cad Files".
3. If this is your first time using Planner, see "Understanding the User Interface" for a basic overview of the user interface.
4. Import a part model into Planner, or directly interface with a part model using DCI. See "Importing CAD Data" or "Using Direct CAD Interfaces". You can also import existing inspection plans into the current model. See "Importing an Inspection Plan".
5. If you need to modify any settings, see "Setting Preferences".
6. If desired, add GD&T Cad Callouts to your part model. See "Adding GD&T CAD Callouts".
7. Add a new inspection plan to the Plan window. See "Working with Inspection Plans" for ways on how to do this.
8. Add commands to your inspection plan. You can use the **Insert** menu or appropriate toolbars to insert these types of commands:
    - Alignments. See "Creating Alignments".
    - Auto Features. See "Creating Auto Features".
    - Constructed Features. See "Constructing Features".
    - Section Cuts. See "Working with Section Cuts".
    - Dimensions. See "Adding Dimensions to Features"
    - Comments. See "Inserting Comments".

9. Finalize the inspection plan by "releasing it". This allows the inspection plan to be imported when the .cad file is imported into a part program in PC-DMIS. If desired, you can also manually export the inspection plan as a text file (.ip file).
10. Save your .cad file.
11. Import the .cad file or .ip file into PC-DMIS.

Understanding the User Interface

When you first launch PC-DMIS Planner you will need to either create a new .cad file (use **File | New**) or open an existing .cad file (use **File | Open**). Once you import a part model and create an inspection plan, the user interface will look something like this:



1. Title Bar and Menu Bar - This contains the title bar, listing your Planner version along with the name of the currently loaded .cad file. Below the title bar is the menu bar. The menu bar contains the standard menus available to PC-DMIS Planner. The menu bar helps you define desired settings, save and load .cad files, import information

into your .cad file, print the contents of the Graphic Display window, insert various commands into your inspection plan, and all other operations needed to create inspection plans.

2. Toolbar Area - This area contains any toolbars you choose to display. They provide a quick way of accessing most of the functionality available in the menu bar. By default, the **Dimensions (Plan)** toolbar is displayed to the left of the Plan window and the **Dimensions (CAD GDT)** toolbar is displayed to the right of the Graphic Display window. See "Viewing Toolbars and the Status Bar".

3. Plan Window- The Plan window on the left displays all the inspection plans contained within the current .cad file as well as the commands contained within each plan. See "Working with Inspection Plans".

4. The Graphic Display window - This window displays graphical information about your part model and inspection plans. Planner will use this window to display any graphical information as well as defined features, feature IDs, and CAD GD&T callouts. You can use this window to interact with your part to define CAD features to measure. See "Working with the Graphic Display window".

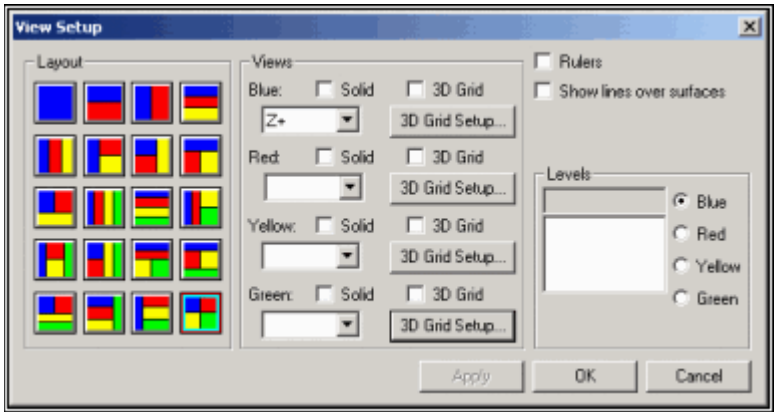
5. The Status Bar - The status bar displays instructions and other useful information. See "Status Bar".

Setting Preferences

Select the **Edit | Setup** menu and use the menu items there to set your desired preferences.

Setting Up the View

The **Edit | Setup | View Setup** option opens the **View Setup** dialog box. This dialog box lets you specify the number of views (maximum of four) of the part that Planner will display in the Graphic Display window. You can also determine the orientation of each view and if the particular view is displayed as a wire frame or a solid.



View Setup dialog box

Hint: To quickly change how the part is displayed in the 'blue' view, you can click on the icons available from the **Graphic View** toolbar.

Changing Layout and Views

To change the screen layout and the orientation of the views:

1. Access the View Setup dialog box (Edit | Setup | View Setup).
2. Select the desired screen style from the **Layout** area. For example, to split the screen into two different views of the part image, click on button two or three (in the first row). The window will be split horizontally or vertically, depending on the button that is chosen.

3. Using the drop-down list box, select the desired view orientation for each window that will be displayed. For example, to view the part image from the Z+ direction, simply select **Z+** from the drop-down list box. Or, to view the same image from the Y- axis, select **Y-**. All of the display options only affect how PC-DMIS displays the part image. They do not affect the measured data or inspection results.
4. Select the **Solid** check box if you want to display your part in the selected view of your part as a solid. Leaving the check box blank will display the view of the part as a wire frame.
5. Click either the **Apply** button or the **Ok** button.
   - If the **Apply** button is selected, PC-DMIS will re-draw the Graphic Display window to reflect the current settings, allowing you to preview your changes.
   - If the **Ok** button is selected, PC-DMIS will close the **View Setup** dialog box and apply the changes to the Graphic Display window. At any time before clicking **Ok**, you can click **Cancel** to close the dialog box and revert the views to what they were.

Adding a 3D Grid

Select a **3D Grid** check box to turn on the 3D grid for the selected view. Unlike the rulers, the origin of the 3D grid will always be located at the part's current alignment.

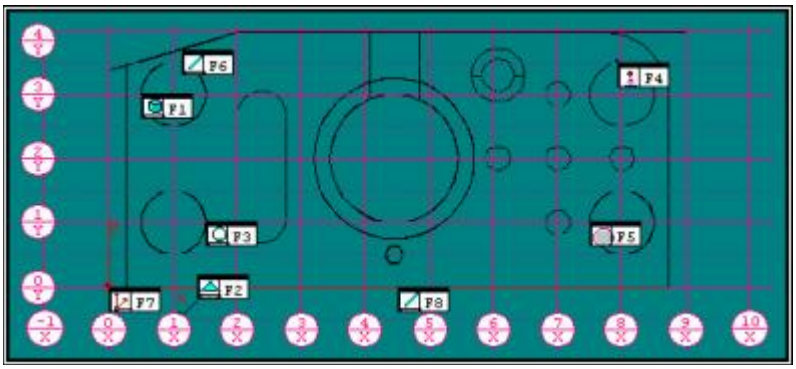
The **3D Grid Setup** button brings up the **3D Grid Setup** dialog box. From here you can modify the grid spacing.

3D Grid Setup dialog box

Modify the grid line spacing by typing a value in the **X Axis**, **Y Axis**, and **Z Axis** boxes or select the **Auto** check boxes to have PC-DMIS automatically compute reasonable spacing.

Grid line labels are placed around the outside of the window. These lines follow the current alignment defined in the Edit window. They can be inset by the value shown in the **Label Inset** box. This value determines the pixel buffer size that PC-DMIS places between the outer edge of the Graphic Display window and the 3D grid labels.

Click **OK** and PC-DMIS displays the 3D grid in the Graphic Display window.

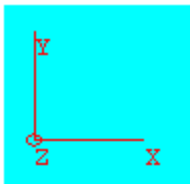


Example of a 3D—Grid with Label Inset of 10 in the X-Y Plane

There are three possible grids that can be drawn:

X-Y
Y-Z
Z-X

PC-DMIS draws only one grid at a time. The grid drawn depends on the axis plane rotated closest to your screen (determined by the axis indicator). For example, the following axis indicator from the Graphic Display window shows the X-Y axis plane facing the user; PC-DMIS draws the X-Y axis grid.



Axis Indicator showing the X-Y plane.

Note: The color of the 3D grid can be modified from the **Screen Color** dialog box. See "Changing Screen Colors".

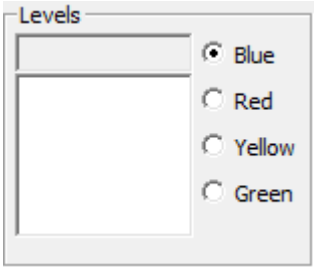
Showing lines over surfaces

This check box applies to solid view only. Select this check box to cause points and lines that hide behind surfaces to show through. This option is useful for looking at features that are behind surfaces. For example, suppose a line in your part is slightly underneath the CAD surface. Selecting this check box will force the line to be visible.

Displaying Rulers

Select the **Rulers** check box to turn on rulers in all views. The origin of the rulers will always be located at the CAD origin (not local part alignments).

Viewing CAD Levels



Levels area of the View Setup dialog box

You can easily view predefined levels of CAD elements in an associated "view" by clicking on the level from the list and then selecting the view by clicking the **Blue**, **Red**, **Yellow**, or **Green** option button.

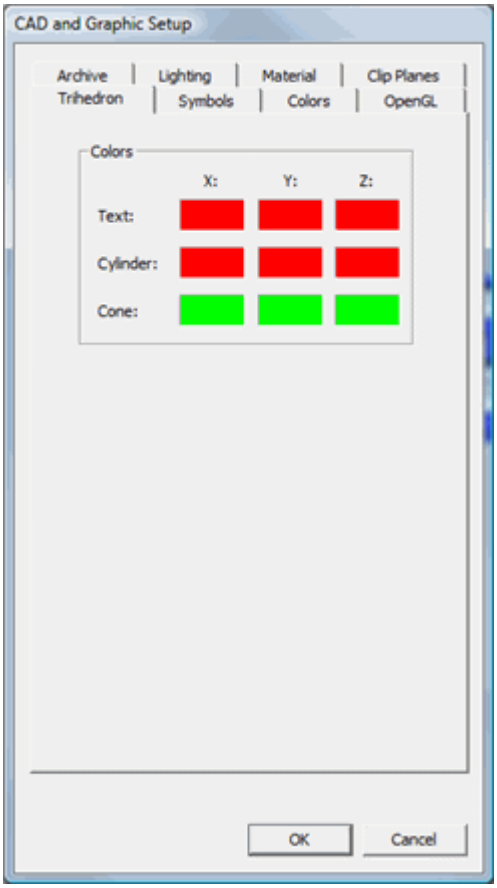
By default, even if you haven't defined a level, PC-DMIS will always display a hidden 0 level. The 0 level is a default level containing all the CAD elements in the CAD model. To create and work with CAD levels, see the "Working with CAD Levels" topic.

Scaling the Drawing

You can scale the CAD drawing to fit the size of your selected view(s) in the Graphic Display window. This option is helpful whenever the image becomes too large or too small. To alter the part image so all features and CAD elements are visible, select the **Edit | Scale To Fit** menu option.

Setting Up CAD and Graphics

The **Edit | Setup | CAD and Graphic Setup** option displays the **CAD and Graphics Setup** dialog box. This gives you a set of tools that allow you to further manipulate the display of CAD models—including your part model, probe model, machine model, and other fixture models—inside Planner's Graphic Display window. You can also use it to change other properties that affect how things are drawn on the screen.



CAD and Graphic Setup dialog box - Trihedron tab

This dialog box contains these tabs:

Trihedron - This tab lets you modify the colors of each axis and component of the trihedron symbol.

Symbols - This tab lets you set properties for the display of wireframe and surface objects, point symbols and path lines.

Colors - This tab lets you define the screen's background color, the highlight color, the mouse-over highlight color, the 3d grid color, and Vision's Field of View (FOV) color.

OpenGL - This tab lets you define OpenGL options that affect the display of the model in solid view mode.

Archive - This tab lets you save and recall these CAD configurations for later use.

Lighting - This tab lets you define and enable light sources, their colors, their directions, light models, and CAD transparency.

Material - This tab lets you apply texture mapping to the surfaces of your CAD models, you can also change the colors and brightness of the materials making up the textures.

Clip Planes - This tab lets you define up to four Clipping Planes that hide your part model in the clipped area. You can use this to cut your part model to show cross sections and cutaway views.

There are four buttons at the bottom of the dialog box:

OK - Accepts your changes and closes the dialog box.

Cancel - Rejects your changes and closes the dialog box.

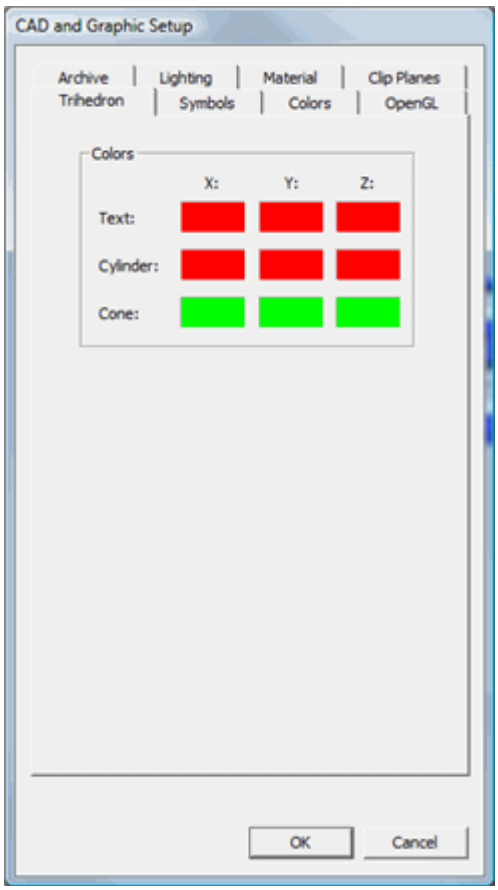
Save System Defaults - (**Lighting** and **Material** tabs only.) Saves your changes as the new default lighting and material settings. This overwrites the default settings that came with Planner.

Restore System Defaults - (**Lighting** and **Material** tabs only.) This brings back the original lighting and material settings that came with Planner.

Note: Any modifications you make to the **CAD and Graphic Setup** dialog box take place in real time in the Graphic Display window. This lets you immediately see the effects of your selected changes. Until you click **OK**, however, the changes you make are only temporary.

Trihedron Tab

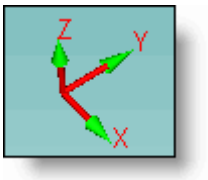
The **Trihedron** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you change the colors of the different components of the part alignment trihedron that appears inside the Graphic Display window.



CAD and Graphic Setup dialog box – Trihedron tab

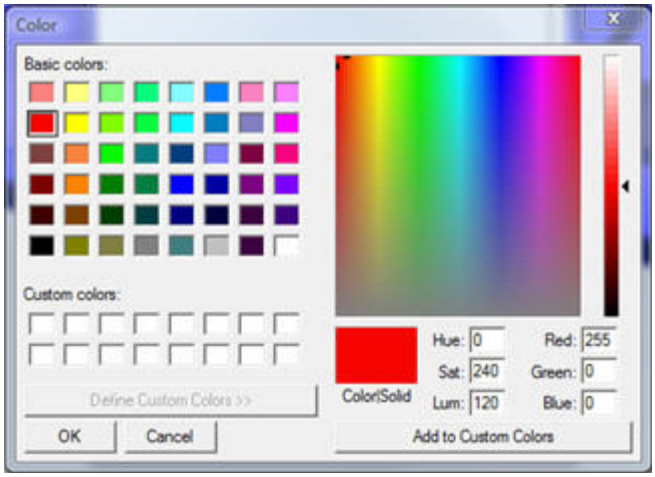
You may find this useful when the background color or other items in the Graphic Display window have similar colors, making it difficult to see the trihedron; or perhaps you just want to give each axis a different color to easily differentiate between the axes.

The original colors for the trihedron symbol are red and green, like this:



Original Colors of the Trihedron

You can change these colors by clicking on the colored boxes in the **Colors** area and choosing a new color from the resulting **Color** dialog box:

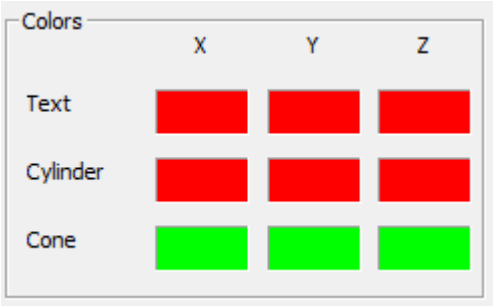


Color dialog box

When you click **OK**, the **Color** dialog box closes, and Planner instantly updates the color of the trihedron accordingly in the Graphic Display window. When you click **OK** in the **CAD and Graphic Setup** dialog box, Planner will store the color information in the registry so that all part programs will use these colors for the part alignment trihedron.

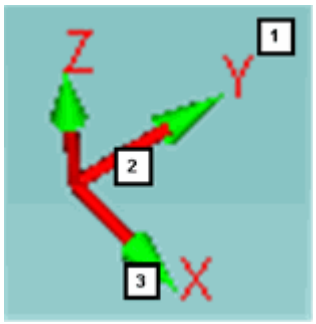
Colors area

The **Colors** area contains a grid of colored boxes representing the different components for each of the three axes of the trihedron.

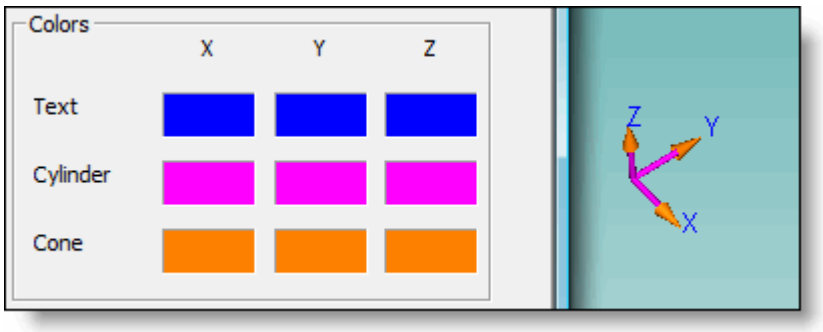


Colors area

- **Text** - This sets the colors of the "X", "Y", or "Z" letters (or text) displayed on the trihedron (1 in image below).
- **Cylinder** - This sets the color of the X, Y, or Z axis line (or cylinder) on the trihedron (2 in image below).
- **Cone** - This represents the color of the X, Y, or Z arrow head (or cone) on the trihedron (3 in image below).



For example, changing the **Text** colors to blue, the **Cylinder** colors to pink, and the **Cone** colors to orange would result in a trihedron like this:

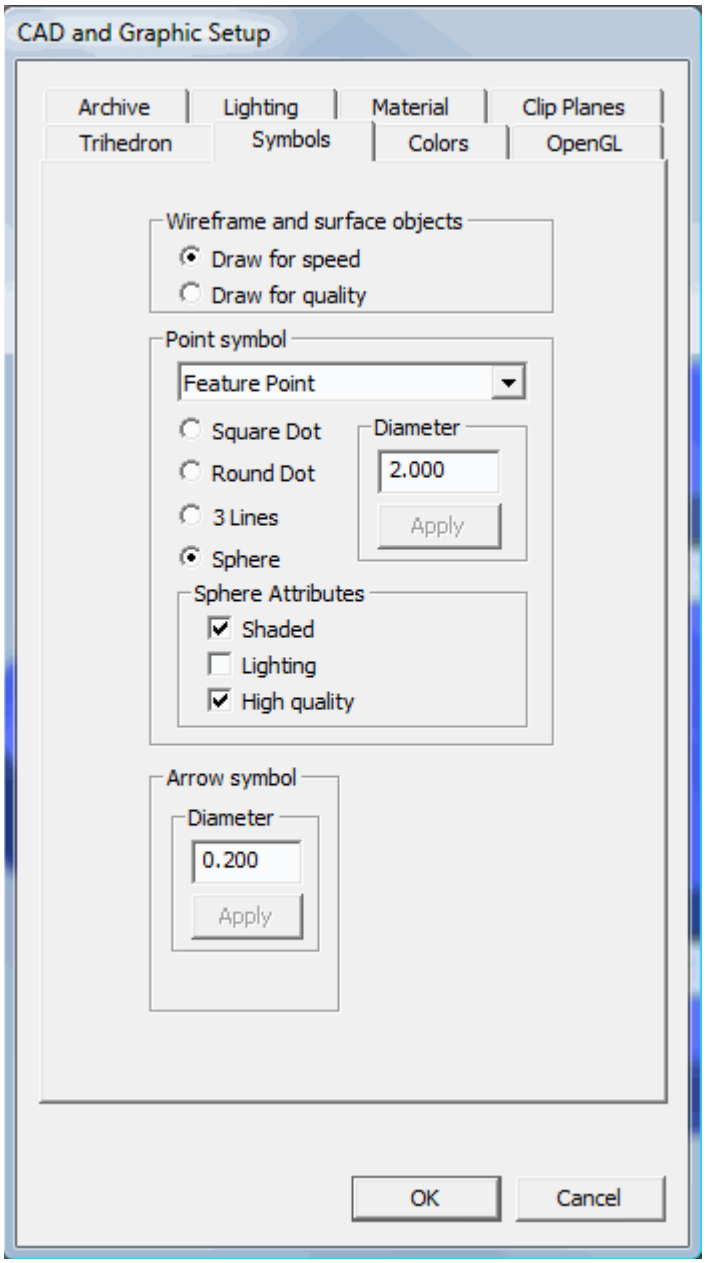


A Sample Modified Trihedron

Symbols Tab

The **Symbols** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you change how different symbols are displayed in the Graphic Display window.

Hint: You can also display this by clicking the **Display Symbols** icon from the **Graphic Modes** toolbar.



CAD and Graphic Setup dialog box – Symbols tab

The initial values for the settings on the **Symbols** tab come from the registry or hard-coded defaults.

Available symbols you can modify include **Wireframe and Surface objects**, the **Point Symbol**, the **Arrow Symbol**, and **Path Lines**. Corresponding areas exist on the **Symbols** tab. Whenever you make a change to one of the check boxes or option buttons, Planner will apply that change automatically so you can see what effect it will have.

Planner will only save your changes once you click the **OK** button. To see changes to the symbols' sizes click the appropriate **Apply** button.

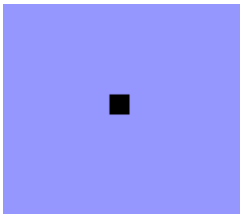
Draw for speed option
This option turns off anti-aliasing for wireframe and surface objects, optimizing the drawing speed of CAD objects in the Graphic Display window.

Draw for quality option
This option turns on anti-aliasing for wireframe and surface objects. This optimizes the drawing quality of CAD objects in the Graphic Display window.

Point Symbol drop-down list
This list defines the type of Point to modify. You can choose either **CAD Point** or **Feature Point**. The default item is **Feature Point**.

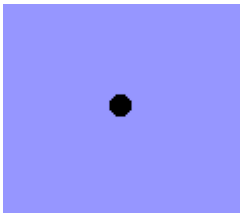
Diameter / Width in Pixels box
This box defines the size of the point symbol. **Square Dot** and **Round Dot** symbols use **Width** in pixels; **3 Lines** and **Sphere** use **Diameter** in plan units. Note that the maximum size for **Round Dot** is based on the computer's physical video card. If the size exceeds the limits of the current computer's video card, it will simply display the symbol at the largest size available to the current system's video card.

Square Dot option button
This option displays the point symbol as a square dot. This is the quickest symbol for Planner to draw.



Example Square Dot symbol

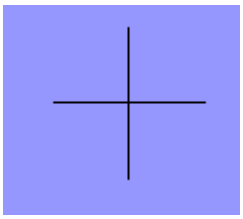
Round Dot option button
This option displays the point symbol as a round dot.



Example Round Dot symbol

Note: Square Dot and **Round Dot** symbols are drawn flat on the screen and may be clipped by the CAD model.

Three Lines option button
This option displays the point symbol as a three-lined cross hair.



Example Three Lines symbol

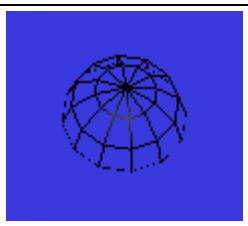
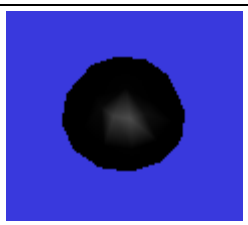
Sphere option button
This option displays the point symbols as a sphere. This is the slowest symbol for Planner to draw, especially if you have all the attributes selected.

These check boxes become available if you select the Sphere symbol. They provide you with additional attributes to further control the display of a sphere symbol in the Graphic Display window.

- **Shaded** - This check box produces a shaded (opaque) sphere symbol.
- **Lighting** - This check box adds OpenGL lighting to the sphere symbol.
- **High Quality** - This check box produces a smoother looking sphere symbol.

While these check boxes improve the image quality of the sphere symbol, they will also cause a slight increase in the time it takes to draw the sphere symbols whenever the screen gets refreshed.

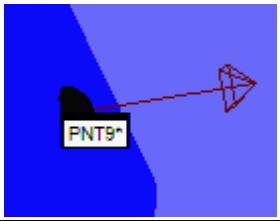
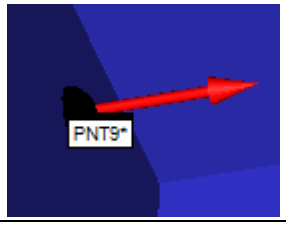
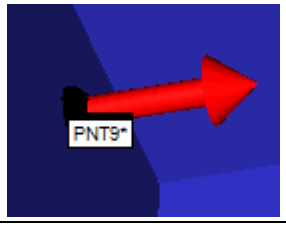
Some Examples:

	Shaded, High Quality
	Lighting, High Quality
	Shaded, Lighting, High quality

Arrow Symbol box
This box controls the diameter size (in plan units) of the arrows displayed in the Graphic Display window. The size of the actual arrow in the display will change only if the **Shaded** check box is selected.

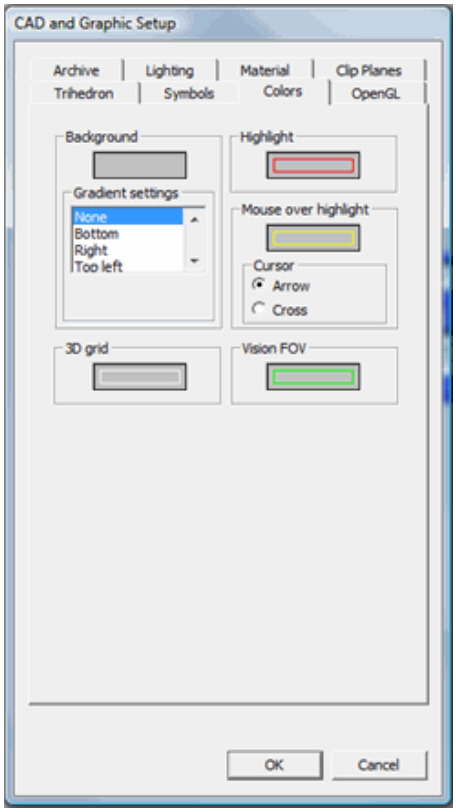
Shaded - This check box shades arrow symbols, in essence drawing arrows as cylinder tubes with a set diameter. If not selected, arrows are drawn as simple lines without size.

Some Examples:

	Not Shaded
	Shaded
	Increased Diameter

Colors Tab

The **Colors** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) allows you to determine the colors to display within the Graphic Display window.



CAD and Graphic Setup dialog box – Colors tab

These include the background color, a gradient color, highlight, and mouse-over-highlight colors, the 3D Grid color, and the Vision module's Field of View (FOV) color.

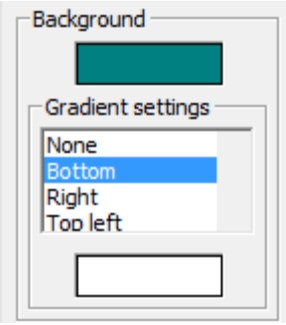
If you want to change the color of your part model, use the **CAD Assembly** dialog box. See the "Working with CAD Assemblies" topic.

To Change a Color:

To change the colors:

1. Access the Colors tab on the CAD and Graphic Setup dialog box (Edit | Setup | CAD and Graphic Setup).
2. Click on rectangular color box for **Background**, **Gradient**, **Highlight**, **Mouse Over Highlight**, **3D Grid**, or **Vision FOV**. The **Color** dialog box will appear.
3. Choose a new color for the option you selected. Planner immediately displays the selected color change, allowing you to immediately preview the change.
4. Click **OK**. The **Color** dialog box will close returning you to the **Colors** tab of the **Cad and Graphics Setup** dialog box.
5. When finished, click the **OK** button.

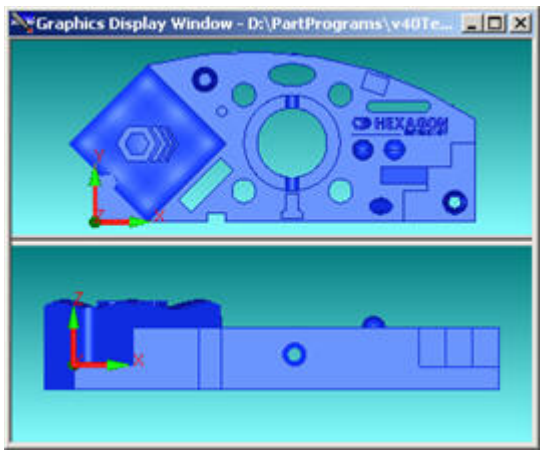
Background



The **Background** area allows you to change the background screen color. This color is also used as the background color for any Feature Control Frames displayed in the Graphic Display window. Follow the directions discussed in the "To Change a Color" chapter.

The **Gradient settings** area lets you define a secondary background gradient color. When you set the gradient to something other than **None** in the list, a rectangular color box appears. You can click that box to define a secondary gradient color. Then, when setting the screen's background, Planner will start from the setting location (for example the Bottom) and display the secondary gradient color, and then gradually change from that color to the primary gradient color as it draws the colors nears the other end of the screen.

For example, a **Bottom** gradient setting might look like this:

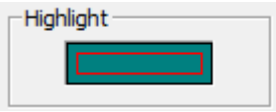


Gradient Colors Example

The available items in the **Gradients settings** list are:

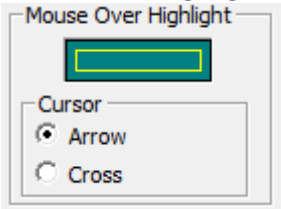
- **None** - No secondary gradient. The screen's background will be the solid primary color.
- **Bottom** - The secondary gradient color starts at the bottom of the screen and gradually changes to the primary color as it nears the top.
- **Right** - The secondary gradient color starts at the right side of the screen and gradually changes to the primary color as it nears the left side.
- **Bottom Right** - The secondary gradient color starts at the bottom right side of the screen and gradually changes to the primary color as it nears the top left side.
- **Bottom Left** - The secondary gradient color starts at the bottom left side of the screen and gradually changes to the primary color as it nears the top right side.
- **Top right** - The secondary gradient color starts at the top right side of the screen and gradually changes to the primary color as it nears the bottom left side.
- **Top left** - The secondary gradient color starts at the top left side of the screen and gradually changes to the primary color as it nears the bottom right side.

Highlight



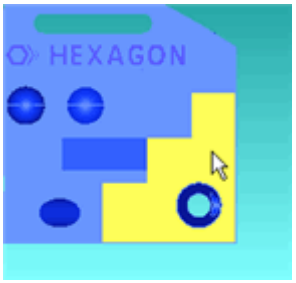
The **Highlight** area allows you to change the color used for highlighting within the Graphic Display window of Planner. Follow the directions discussed in the "To Change a Color" chapter.

Mouse Over Highlight

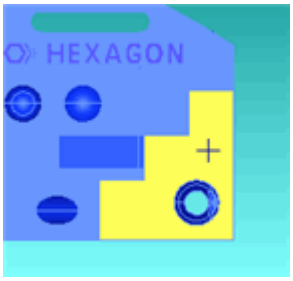


The **Mouse Over Highlight** area lets you change the color that Planner uses as well as the mouse pointer's shape when you highlight surfaces or curves with your mouse using Mouse Over Highlighting (MOHL). Follow the directions discussed in the "To Change a Color" chapter.

The **Cursor** area lets you display either an **Arrow** or a **Cross** shape for the mouse pointer when in MOHL mode:



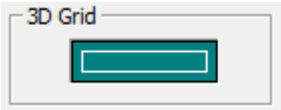
Arrow Shape Mouse Pointer



Cross Shape Mouse Pointer

See "Highlighting Surfaces and Curves with the Mouse" for information.

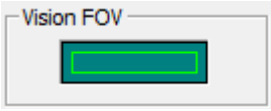
3D Grid



The **3D Grid** area allows you to change the color used for the 3D grid in the Graphic Display window. Follow the directions discussed in the "To Change a Color" chapter.

For more information on the 3D Grid, see "Adding a 3D Grid".

Vision FOV

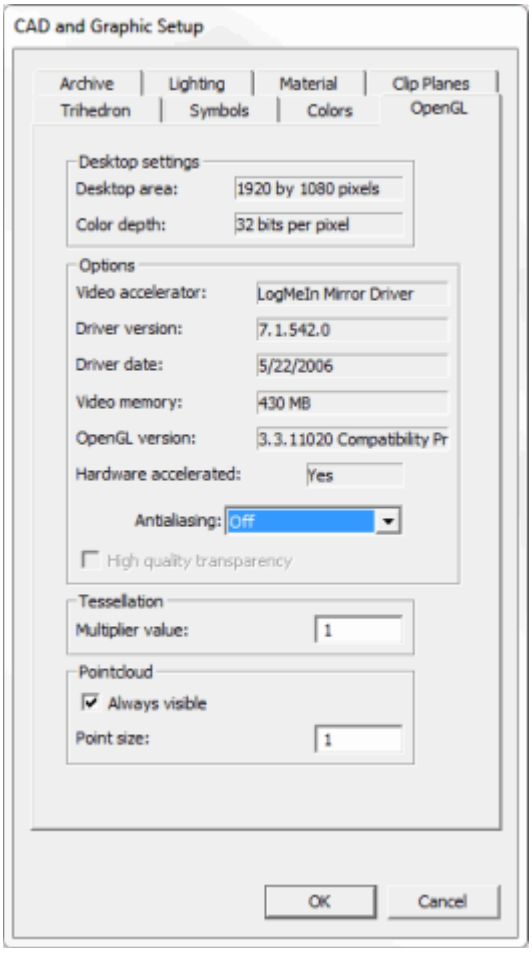


The **Vision FOV** area lets you change the color used for the Field of View (FOV) when using the PC-DMIS Vision module. Follow the directions discussed in the "To Change a Color" chapter. See the PC-DMIS Vision documentation for information on the FOV and on the Vision module.

OpenGL Tab

The **OpenGL** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you change the OpenGL options that affect the display of the model in solid view mode. To change the part to solid view, see "Setting Up the View".

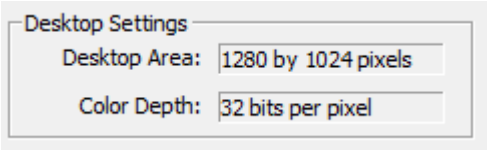
Hint: You can also access this tab by clicking the **OpenGL Settings** icon from the **Tools** toolbar.



CAD and Graphic Setup dialog box - OpenGL tab

This tab contains these items.

Desktop Settings area



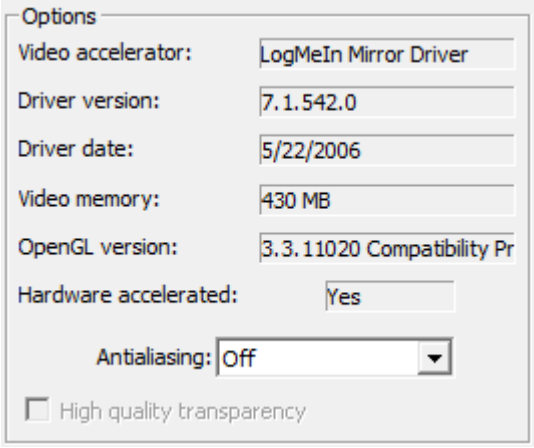
There can be different OpenGL options for each desktop display setting. The **Desktop Settings** area shows what the current desktop settings are.

Resolution Ratios of Different Monitor Sizes

Wide screen monitors need a 1.6 ratio instead of a 1.3333 ratio used by normal monitors. For example, a resolution of 1200x1600 has a 1.3333 ratio (1600/1200) and works well for a normal size monitor, while a resolution of 1680x1050 has a 1.6 ratio, good for a wide screen monitor. If you use a wide screen monitor and your

screen appears stretched (perhaps your circle features appear as ellipses in the Graphic Display window), use a 1.6 resolution ratio to resolve this problem.

Options area



The **Options** area displays information about the system's video card:

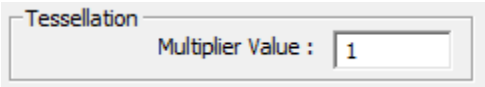
- **Video Accelerator** - Video card description
- **Driver version** - Video driver version
- **Driver date** - Video driver date of release
- **Video memory** - Amount of video card RAM
- **OpenGL version** - Version of OpenGL supported by the video driver
- **Hardware accelerated** - Displays Yes or No depending on whether graphics rendering will be hardware accelerated. Hardware acceleration is much faster than software acceleration.

Antialiasing - The **Antialiasing** combo box allows the level of antialiasing by specifying the number of multisamples. 2x antialiasing will sample each pixel twice. 4x antialiasing will sample each pixel four times and so on. When antialiasing is enabled, each pixel is sampled multiple times at slightly different locations within the pixel. An average color is computed from these samples to determine the final pixel color. This effectively reduces the model's jagged edges in the Graphic Display window. Higher antialiasing settings will produce better visual results at the expense of slower system performance.

The Antialiasing options are determined by the capabilities of the graphics accelerator. Some graphics accelerators may support 64x antialiasing while others may only support up to 16x or may not support it at all. If antialiasing is supported, the default is 8x - if not supported, the default is Off.

The **High quality transparency** check box controls the **`HighQualityTransparency`** registry setting. This feature is disabled if your graphics driver does not support OpenGL 4.2.

Tessellation area



The **Tessellation** area controls the drawn image by setting a tessellation multiplier in the **Multiplier Value** box. Planner multiplies the **Multiplier Value** by the tessellation value for the given CAD system. These values are then used in the generation of the shaded image.

Note: The tessellation value is the default value used to break up surfaces into patches for shading.

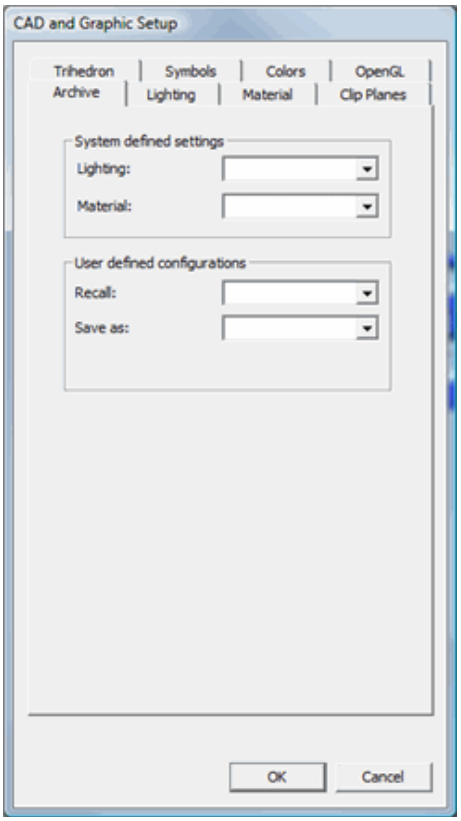
The tesselation multiplier updates the Graphic Display window immediately once you click outside of the **Multiplier Value** box or press TAB to move to a different item on the dialog box.

Notes:
The size of the CAD data file and the tessellation multiplier value used affects the amount of memory needed. These both affect the amount of tessellated facets needed to display the model. The smaller the tessellation multiplier value used, the more memory needed for the facets. For large CAD models this could cause an "Out Of Memory" error. If this occurs, the current PC-DMIS session will be left in an unstable state and should be terminated.
The default tessellation multiplier value is 1.0. Setting a tessellation multiplier of 0.1 will result in a 10 to 20 percent increase in the memory required over the default value of 1.0. Decreasing the tessellation multiplier further to 0.01 will result in an additional 50 to 65 percent increase of memory required.

Archive Tab

The **Archive** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you store and recall lighting and material settings.

Hint: You can also use the **CAD Lighting** icon from the **View | Toolbars | Graphics Modes** toolbar to access this tab.



CAD and Graphic Setup dialog box – Archive tab

This tab contains two areas, the **System defined settings** area and the **User defined configurations** area.

The **System defined settings** area lets you select from system defined lighting and material settings. These are stored in a configuration file named LightingMaterials.dat in the directory where you installed Planner.
The **User defined configurations** area lets you save and recall your own custom lighting and materials configurations.

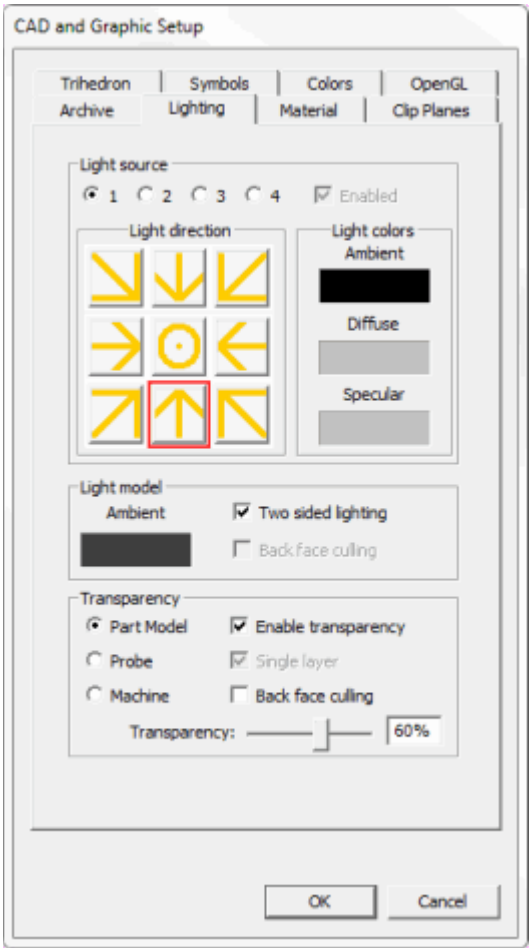
To save a custom defined configuration:

1. Make any changes you want in the other tabs of the **CAD and Graphic Setup** dialog box.
2. Click in the **Save as** box. A **Save Now** button appears.
3. Type a name for your saved configuration.
4. Click the **Save Now** button.

To recall a user defined configuration, select the saved configuration from the **Recall** list.

Lighting Tab

The **Lighting** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you define lighting settings to use with your part in the Graphic Display window.

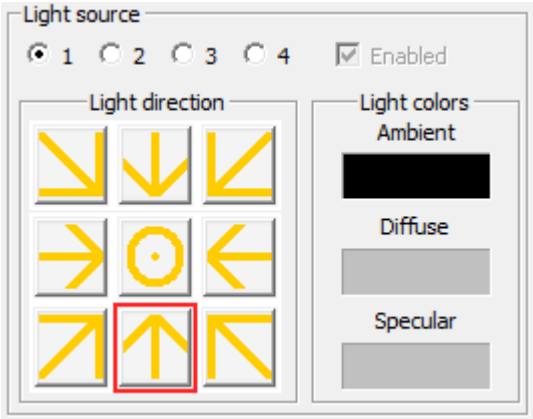


CAD and Graphic Setup dialog box – Lighting tab

The **Lighting** tab consists of three areas that allow you to define your light sources, your light model, and transparencies.

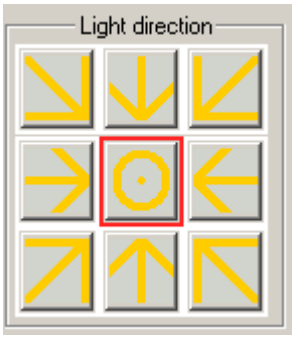
Light source

The **Light source** area defines up to four light sources, each specifying their respective light's direction and colors. You can create a new light source by selecting the appropriate number and clicking the **Enabled** check box.



With the exception of the first light source, which remains always enabled, the **Enabled** check box lets you turn the current light source on or off.

Once you enable a light source, an active light direction button becomes selected with a red outline. These buttons define the direction from which the light originates.



The center light direction button acts as if the light comes from the direction you are located as if shining through your computer screen onto the CAD part. The other direction buttons define light coming from other angles.

You can change the **Ambient**, **Diffuse**, or **Specular** light for the current light source by simply clicking on the colored rectangle beneath each type of light and selecting the new color from the **Color** dialog box that appears.

Note: Be aware that light source **1** has white as the default color for **Diffuse** and **Specular**, while the other three light sources use black as the default color.

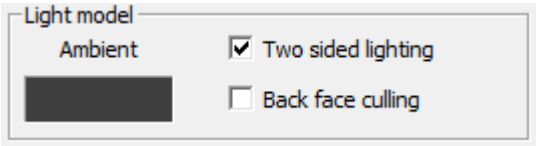
Ambient illumination is light that has been scattered so much by the environment that its direction is impossible to determine - it seems to come from all directions. Backlight in a room has a large ambient component, since most of the light that reaches your eye has bounced off many surfaces. A spotlight outdoors has a tiny ambient component; most of the light travels is the same direction, and since you are outdoors, very little of the light reaches your eye after bouncing off other objects. When ambient light strikes a surface, it's scattered equally in all directions.

Diffuse light comes from one direction, so it's brighter if it comes squarely down on a surface than if it barely glances off the surface. Once it hits a surface, however, it's scattered equally in all directions, so it appears equally bright, no matter where the eye is located. Any light coming from a particular position or direction probably has a diffuse component.

Specular light comes from a particular direction, and it tends to bounce off the surface in a preferred direction. A laser beam bouncing off a mirror produces almost 100 percent specular reflection. Shiny metal or plastic has a high specular component and chalk or carpet has almost none. You can think of specular light as "shininess".

If you disable a light source, the **Light direction** and **Light colors** buttons become unavailable for selection.

Light Model



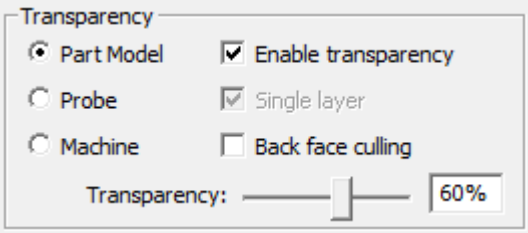
The **Light Model** area defines information applied to the entire scene, regardless of the light source.

Two sided lighting – This check box determines whether or not the front and back faces of surfaces should be lighted. You should select this check box for imported IGES files and for some other CAD formats when the surface normals are not correct.

Back face culling – This check box determines whether or not Planner should cull, or hide, the back faces of surfaces. You should select this check box for IGES files, and some other CAD formats, when the surface normals are not correct.

Ambient – This box defines the ambient color applied to the entire scene. Click on the box to change colors.

Transparency



Sometimes parts of a CAD model interfere with seeing other areas of the model. Enabling transparency reduces this problem by allowing you to "look through" the part image. In this way you can see sections of the part that may actually be hidden.

Once you have enabled transparency you can then select features and other items on your part by clicking on your part model *through the transparent model*.

Once you select the **Enable transparency** check box, you can select the other items in the **Transparency** area.

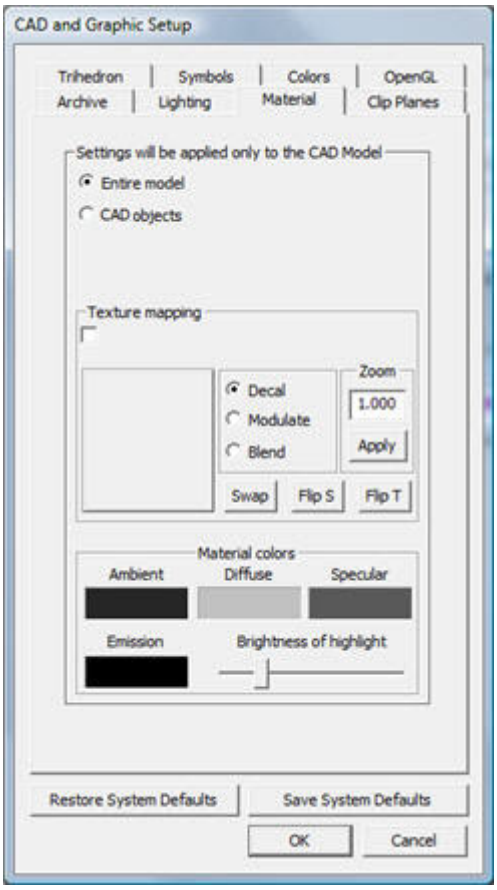
Single layer - This check box is only enabled when the **High quality transparency** check box from the **OpenGL** tab is enabled and checked. When the **Single layer** check box is selected, only the top-most transparent surface of the selected model (CAD Model, Probe, or Machine) will be rendered. In effect, this will hide the internal features of a transparent model even though other models can still be viewed through the transparent model. Note that rendering is faster when this option is selected.

Back face culling - This check box causes Planner to not draw the back faces of surfaces for the selected transparent object.
Transparency - This slider lets you determine the percentage of transparency for the selected object. Moving the slider to the left makes the display object more opaque, while moving the slider to the right makes it more transparent. The value (0 - 100) can also be updated in the **Transparency** input field to the right of the slider.

Once you define transparencies here you can easily switch them on and off using the **Toggle Graphic View Transparency** icon from the **Graphic View** toolbar. See the "Graphic View Toolbar" topic for more information.

Material Tab

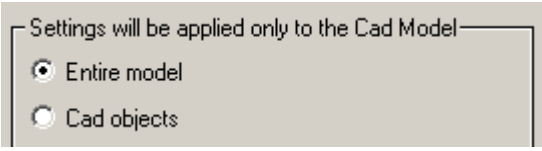
The **Material** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | CAD and Graphic Setup**) lets you define material (or texture) settings used on CAD objects or on your part model.



CAD and Graphic Setup dialog box – Material tab

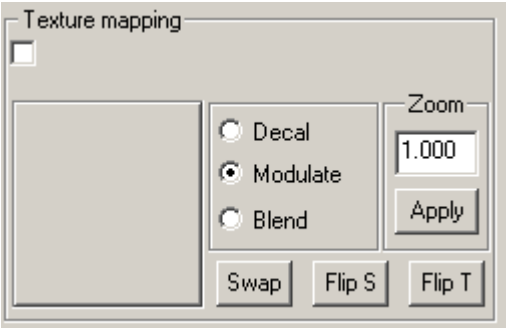
The **Material** tab consists of settings that will be applied to CAD objects making up the imported CAD part model or the entire part model altogether.

The first area of this tab contains two option buttons that allow you to determine the "active selection", either **Entire** model or **CAD objects**.



Entire model – Selecting this option applies the settings to the entire CAD model.
CAD objects – Selecting this option applies the settings to specific selected CAD objects making up the entire model.

Texture mapping



The **Texture mapping** area defines the texture to be applied to the active selection. PC-DMIS automatically applies the texture image as often as necessary to cover the entire active selection.

The check box displays the current bitmap name once you select a bitmap file to use for the texture. This check box is always selected when the active selection is a CAD object.

The large gray square area under the check box displays the current texture.

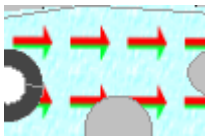
To apply a texture to your part:

1. Select click the gray square area. A standard **Open** dialog box appears.
2. Use the **Open** dialog box to navigate to and select a valid bitmap file. It must have a width and height of 2 to some power. For example 2 to the 5th power is 32 and 2 to the 4th power is 16. So if your bitmap is 32 X 16 pixels, it would be a valid bitmap. If you had something like 32 X 20, however, PC-DMIS would display an error message.
3. PC-DMIS displays a preview of the selected texture inside the gray square area.
4. Click **Open** to accept the texture.

To apply the check box to the active selection, click the **Enable** check box as needed.

The other items in the **Texture Mapping** area determine how PC-DMIS should apply the texture and surface colors:

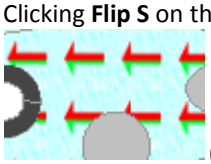
Decal prevents any of the surface's color from being emitted, so you will only see the texture's color.
Modulate and **Blend** use OpenGL algorithms to determine the final display colors.
Zoom controls the "zoom" factor to be applied to the current texture. Values greater than 1.0 zoom *into* the texture, repeating the texture less often. Values less than 1.0 zoom *out of* the texture, repeating the texture more often. For example, a value of 2.0 will make the bitmap twice the size (repeat half as often), and a value of 0.5 will make the bitmap half the size (repeat twice as often).
The buttons **Flip T**, **Flip S**, and **Swap** determine the orientation of the texture. The texture is a two dimensional image, the orientation is described in terms of **S** and **T**. A texture with an arrow drawn atop it helps visualize what happens:
The original texture looks like this, an arrow pointing to the right with the top half of the arrow red and the bottom half of the arrow green:



Clicking **Swap** on the original switches the orientation so that **S** becomes **T** and **T** becomes **S**. This causes the arrow to point up but to also flip the bottom and top of the arrow:

(Swap example)

Clicking **Flip S** on the original changes the direction of **S**. This causes the arrow to point to the left:


(Flip S example)

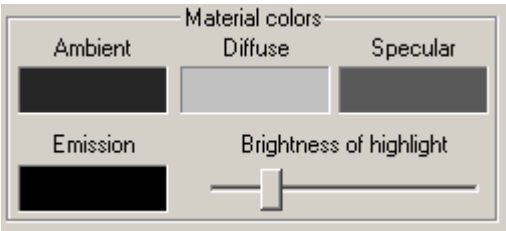
Clicking **Flip T** on the original changes the direction of **T**. This causes the arrow to flip, making the bottom and top switch places:

(Flip T example)

You can click these buttons in different combinations to get a variety of orientations for your texture.

Click **Apply** to see any changes you made to texture orientations or other material colors.

Material colors



The **Material colors** area defines the color information to be applied to the entire CAD Model.

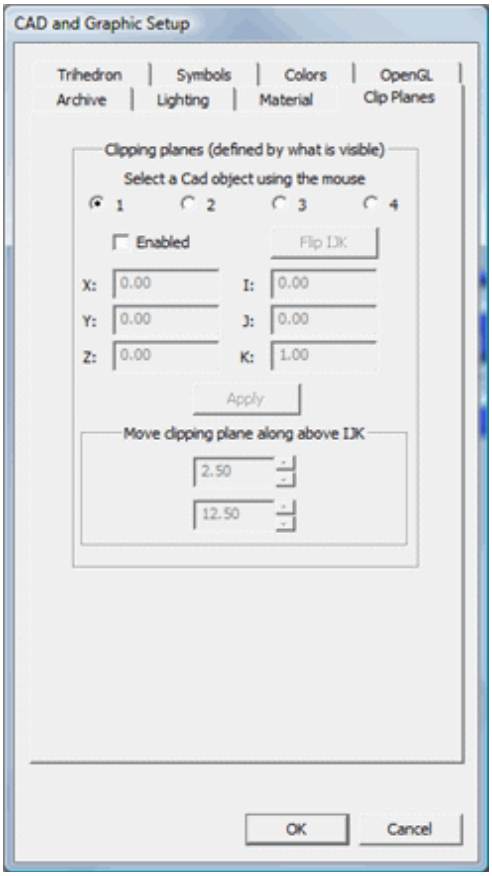
The **Ambient**, **Diffuse**, and **Specular** function similarly to those already explained in the **Light Colors** area of the **Lighting** tab. See "Lighting Tab".

Emission - Materials may have an emissive color, which simulates light originating from an object. In the OpenGL lighting model, the emissive color of a surface adds intensity to the object, but is unaffected by any light sources. Also, the emissive color does not introduce any additional light into the overall scene.

Brightness of highlight - This slider controls the intensity of the highlight when looking at a curved surface.

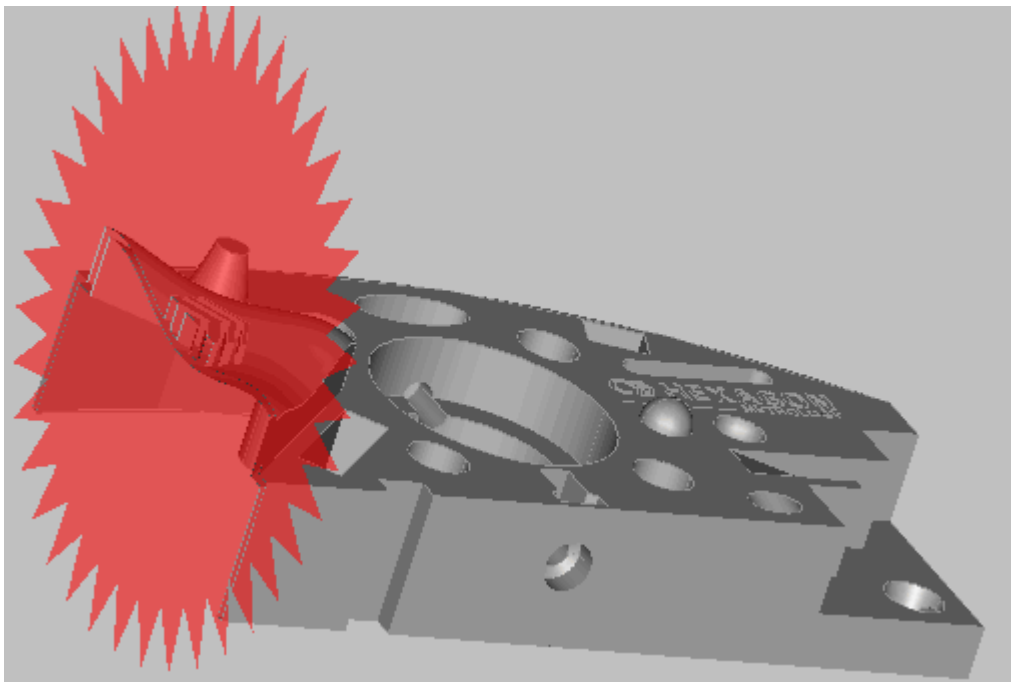
Clip Planes Tab

The **Clip Planes** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | Cad and Graphic Setup**) allows you to define up to four planes, called "clipping planes" that you can use to hide the display of your part model on one side of the plane. Your part model on the other side of the plane remains visible. These planes allow you to create cross sections of your part model.



CAD and Graphic Setup dialog box – Clip Planes tab

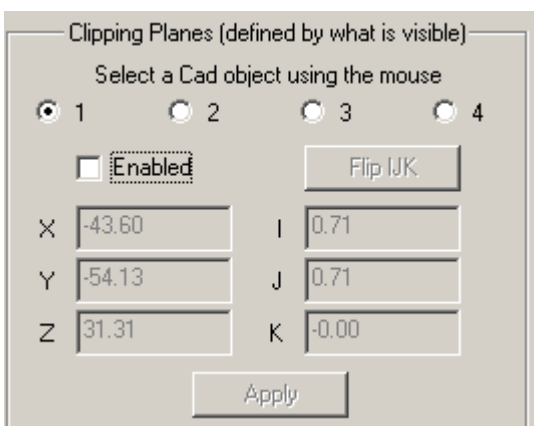
The clipping planes appear in the Graphic Display window as a round saw-like symbol.



Example of a clipping plane creating a cross section of the Hexagon test block

Defining a Clipping Plane

You only need two pieces of information to define your clipping plane: An XYZ location and an IJK vector. The IJK vector points in the direction of what will remain visible.



Clipping Planes area

There are two ways to define this information.

- You can click on your part in the Graphic Display window and PC-DMIS Planner will take the point that you clicked as the clipping plane's location. Once you click a location, Planner automatically enables your clipping plane and sets the vector to the surface normal at the location. You can then modify the IJK vector as desired.
- You can click the **Enabled** check box and then manually type in the XYZ location and IJK vector.

Note: In version 4.0 and later, the XYZ and IJK values are in the active part alignment coordinate system, not the CAD coordinate system as was the case in earlier versions.

The **Flip IJK** button flips the vector so that it points in the opposite direction.

Once you have your clipping plane modified to where you want it, click the **Apply** button to see your changes. Planner will hide all portions of your part model pointing away from your chosen clipping plane vector.

Deselecting the **Enabled** check box disables the current clipping plane. Disabled clipping planes have a yellow color and the ability to place and move your clipping plane is also disabled. Enabled clipping planes red color.

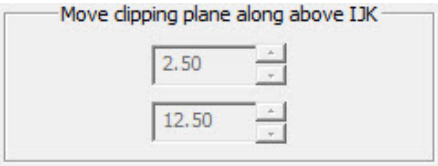
You can define multiple clipping planes by selecting a new clipping plane option button and then following the above instructions.

Adjusting a Clipping Plane

Once you have the clipping plane inserted, you can adjust it by:

- Clicking the **Flip IJK** push button to invert the IJK. This is useful when surface vectors from a CAD selection were incorrect, a relatively common problem with IGES files.
- Manually editing XYZ location and IJK vector information by typing in new values.
- Using the Move clipping plane along above IJK area.

The **Move clipping plane along above IJK** area contains two rows of boxes and up and down arrow buttons. The boxes define increments that the clipping plane will move along the defined vector when you click the up or down arrow buttons for that box.



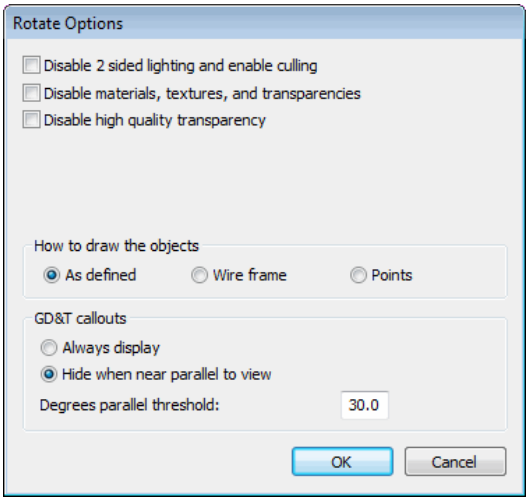
Move clipping plane along above IJK area

The top row allows you to move the clipping plane along the IJK vector in small adjustments.

The bottom row allows you to also move the clipping plane along the IJK vector, in larger increments.

Defining Rotation Options

You can modify how CAD models get displayed during rotation by accessing the **Edit | Setup | Rotate Options** menu item or click the **Rotation Options** button on the **Graphic Modes** toolbar to display the **Rotate Options** dialog box.



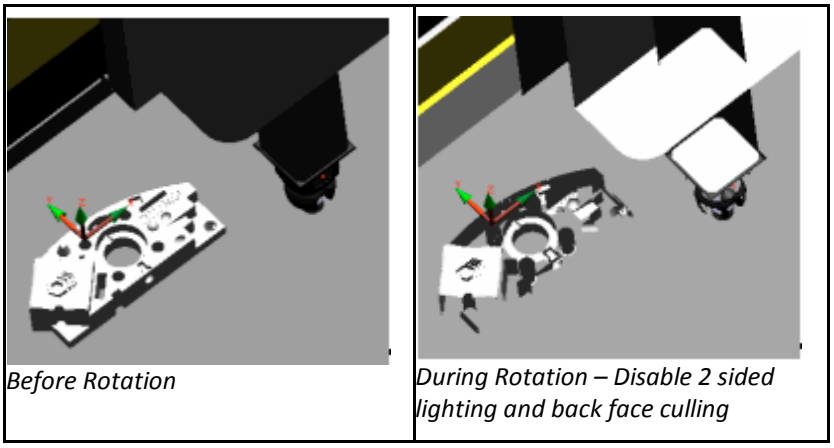
Rotate Options dialog box

This dialog box controls whether or not certain items get drawn and how they get drawn inside the Graphic Display window when you rotate your part model using the right mouse button. The options on this dialog box can help speed up the display of your rotation. When you click **OK**, Planner does not apply these settings globally to all plans; instead, they are only applied to your current plan. This allows you to set different rotation options for each part model.

Important: If you are using 3DxWare devices, this dialog box behaves just like the **Pan, Zoom, Rotate Options** dialog box but with a different dialog box title. 3DxWare devices extend the functionality described for rotation to zooming and panning as well.

Disable 2 sided lighting and back face culling

Selecting this check box disables two sided lighting and uses back face culling during rotation. The resulting lighting effects will roughly display only half of the CAD elements. See the comparison below.

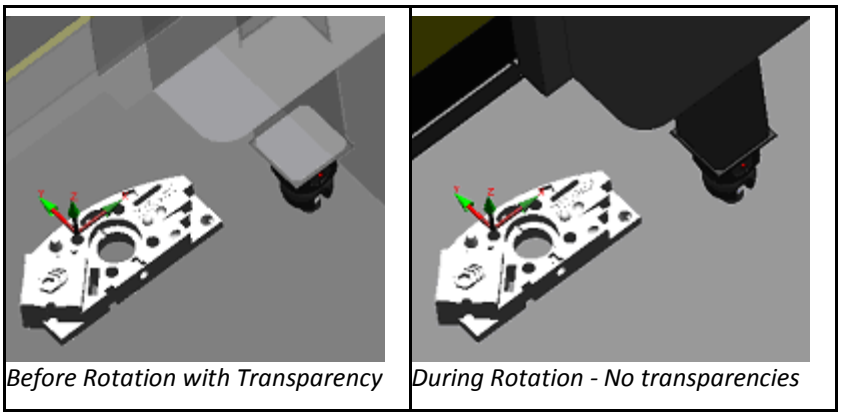


Before Rotation

During Rotation – Disable 2 sided lighting and back face culling

Disable materials, textures, and transparencies

Selecting this check box disables the rendering of applied materials, textures, and transparencies during rotation. See the comparison below.



Before Rotation with Transparency

During Rotation - No transparencies

Disable High Quality Transparency

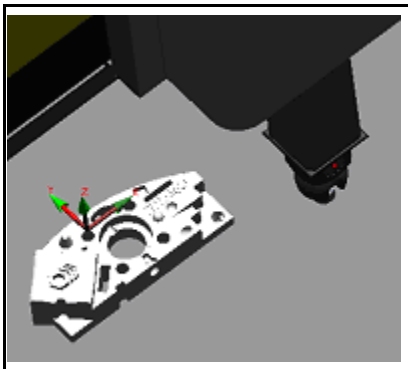
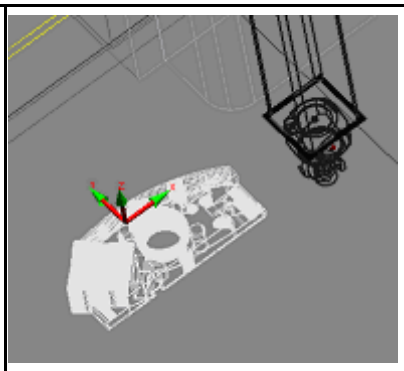
If high-quality transparency is enabled, it will be temporarily disabled when the graphic is rotated, zoomed, or panned. This can be particularly helpful for very complex models that take a long time to render using high-quality transparency.

How to draw objects - As Defined

Selecting this option draws objects as originally defined during rotation.

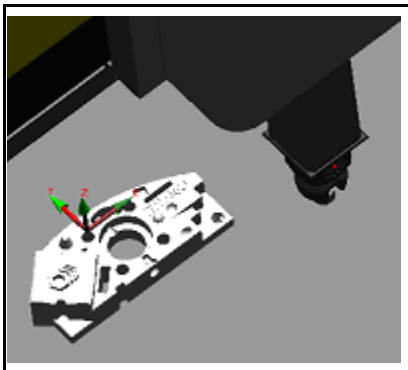
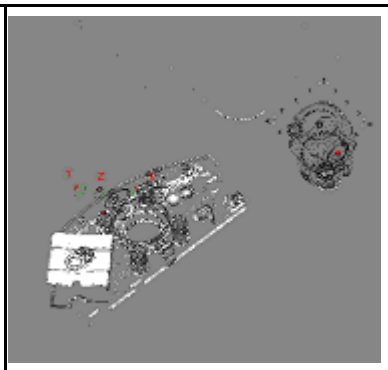
How to draw objects - Wire Frame

Selecting this option draws objects as wire frames during rotation. See the comparison below.

	
Before Rotation	*During Rotation - Draw Wire Frame*

How to draw objects - Points

Selecting this option draws objects as points. The **Points** option gives a representation of the objects using points. See the comparison below.

	
Before Rotation with Transparency	*During Rotation – Draw Points*

GD&T callouts

The **GD&T callouts** area lets you control the display of GD&T callouts within an imported CAD model. It helps reduce screen clutter by making such callouts temporarily hidden as they reach or approach a view angle parallel to your current view angle.

Note: Do not confuse these GD&T callouts with the Feature Control Frame dimensions. While they appear to contain text, the GD&T callouts are essentially CAD objects themselves, not actual dimensions. GD&T callouts are stored with the .cad file. They can be created using Planner or by some CAD software packages.

Always display option

This option indicates that the callouts will always be displayed, regardless of the viewing angle of the screen.

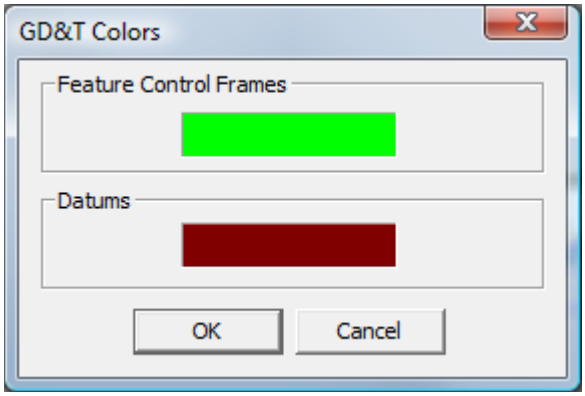
Hide when near parallel to view option

This option indicates that the callouts will be hidden when they cross a threshold of degrees parallel to your view angle. This option is disabled when **Always display** is selected.

Degrees parallel threshold box

This box lets you specify the threshold of degrees parallel to your view angle used to hide the callouts.

Defining GD&T Colors

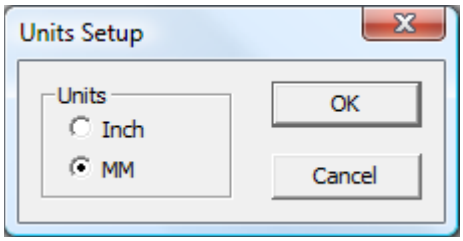


GD&T Colors dialog box

If you select the **Edit | Setup | GD&T Colors**, the **GD&T Colors** dialog box appears, allowing you to define how CAD-based GD&T elements, such as feature control frames and datums, are colored on the CAD model in the Graphic Display window. The default colors are black.

To define a new color, simply click on the rectangle-shaped box for either **Feature Control Frames** or **Datums**, and then from the resulting **Color** dialog box, choose the desired color for that GD&T element.

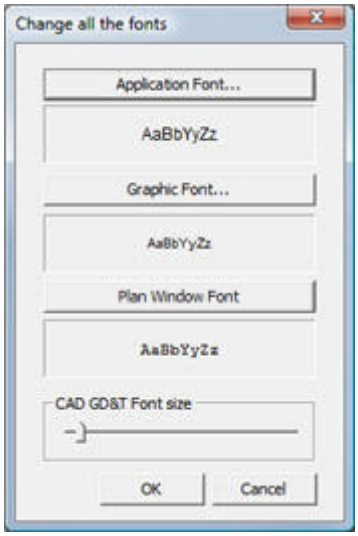
Setting the Measurement Units



Units Setup

If you select **Edit | Setup | Units Setup**, the **Units Setup** dialog box appears, allowing you to set the measurement units for your inspection plan.

Setting Fonts



Change all the fonts dialog box

To modify font attributes, access the **Change all the fonts** dialog box by selecting the **Edit | Setup | Font Setup** menu option. Using this dialog box you can change font attributes—including font, size, and style—for the main user interface, for the Plan window, for the Graphic Display window, and also for CAD GD&T elements within the Graphic Display window.

To change the application, Graphic Display window or Plan window fonts:

1. Select one of these option buttons and the standard **Font** dialog box appears:
   - **Application Font –** This changes the font of the status bar and the tool tips.
   - **Graphic Font –** This changes the font of text displayed on the Graphic Display window, mainly affecting feature labels.
   - **Plan Window Font –** This changes the font on the Plan window. See the note below.
2. Select your font changes from the available lists.
3. Click **OK** to close the **Font** dialog box.
4. Click **OK** at the **Changes all the fonts** dialog box to accept your change.

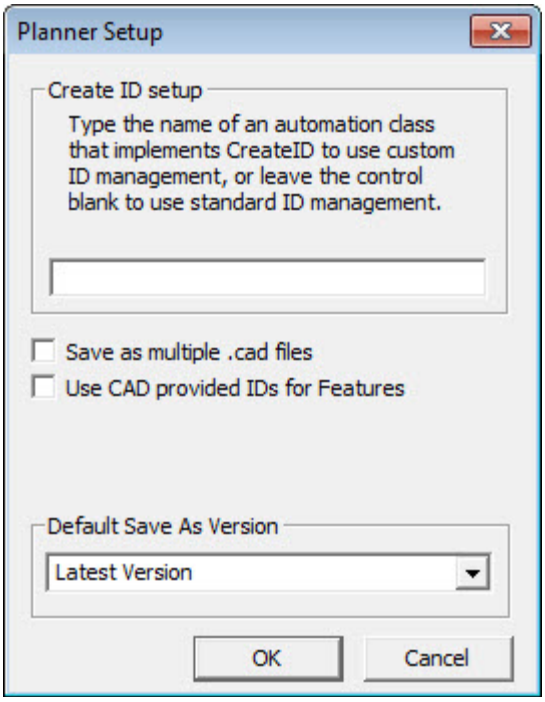
Important: You can use any font available on your computer system, however be aware that some fonts don't have a uniform spacing and may make some components more difficult to read. When viewing or printing the Plan window text, for example, we recommend that you use a monospace (or fixed-width) font (such as the *Courier New* font), otherwise characters, lines, and headings may not line up as expected.

CAD GD&T Font size

This slider lets you scale the size of any GD&T CAD elements displayed in the Graphic Display window. The font size changes dynamically as you adjust the slider. Click **OK** to keep the changed font size.

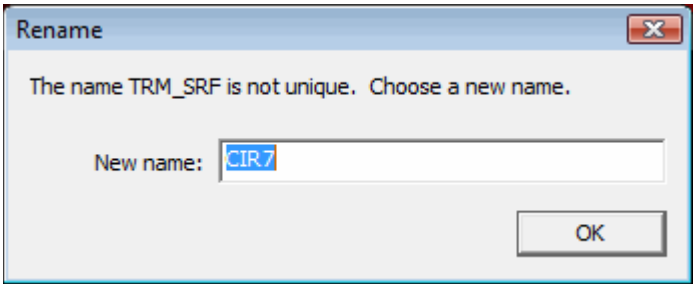
Setting Planner Options

Click **Edit | Setup | Planner Setup** to display the **Planner Setup** dialog box with the following options:



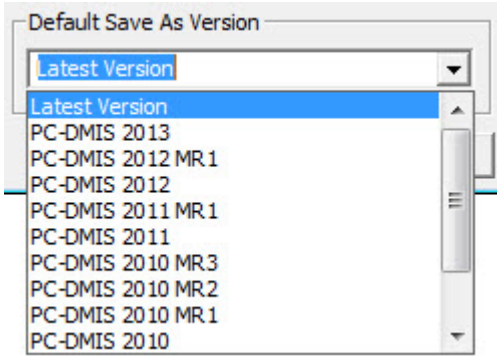
Planner Setup dialog box

Create ID setup: As described, you can provide a custom automation class for feature ID management.
Save as multiple .cad files: When selected, individual .cad files will be created for each model in the assembly. Otherwise the file is saved into a single .cad file for an entire assembly.
Use CAD provided IDs for Features: When selected, Planner automatically uses the feature IDs stored within the CAD file for each CAD feature selected to be added into an inspection plan. If a CAD ID is not unique from the other features already in the inspection plan, Planner will display a ***Rename*** *dialog box* to enter in a unique ID.



Rename dialog box

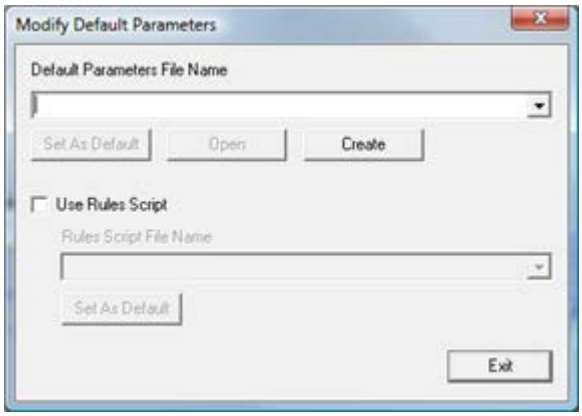
Default Save As Version: Use the drop-down list to select the Planner version that the inspection plan is saved as when you perform a **File | Save As** operation. If you save to an older version and your plan contains items that are not supported in the older version, the item(s) will either be converted into something the previous version can handle, or they will be removed.



Default Save As Version section of the Planner Setup dialog box

Modifying Default Parameters and Rules

Select the **Edit | Setup | Modify Default Parameters** menu option to open the **Modify Default Parameters** dialog box where you can edit the Inspection Plan default parameters.



Modify Default Parameters

The **PC-DMIS Planner Default (.ipd)** file is used to define the parameters for **PC-DMIS Planner** within PC-DMIS. This file allows you to determine how to take hits from features within the **Inspection Plan**. Feature parameters are specified for Auto Features, Constructed Features, and Dimensions. The install includes these two files that you can use as starting points for creating your own default parameters files:

- **Default.ipd** - A general defaults file.
- **VoidDetection.ipd** - A general defaults file for using void detection.

You can also specify how you want to display comments and features. See "Creating and Modifying a PC-DMIS Planner Default (.ipd) file".

The **PC-DMIS Planner Rules (.ipr)** script is used to specify rules for how automatic processes are performed on the **Inspection Plan**. Rules can be modified for certain criteria. For example, the number of hits taken on a circle based upon the diameter of the circle and so forth. See PC-DMIS Planner Rules (.ipr) script".

Before importing an Inspection Plan from a PC-DMIS Planner .cad file or an .ip file, you need to set up these files to apply to imported Inspection Plans.

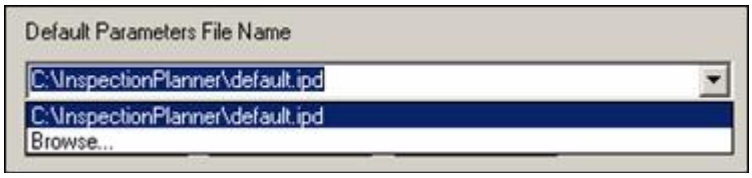
Creating and Modifying a PC-DMIS Planner Default (.ipd) file

To create a new **.ipd** file or make changes to an existing .ipd file, follow this procedure:

1. Select the Edit | Setup | Modify Default Parameters menu option. The Modify Default Parameters dialog box appears.

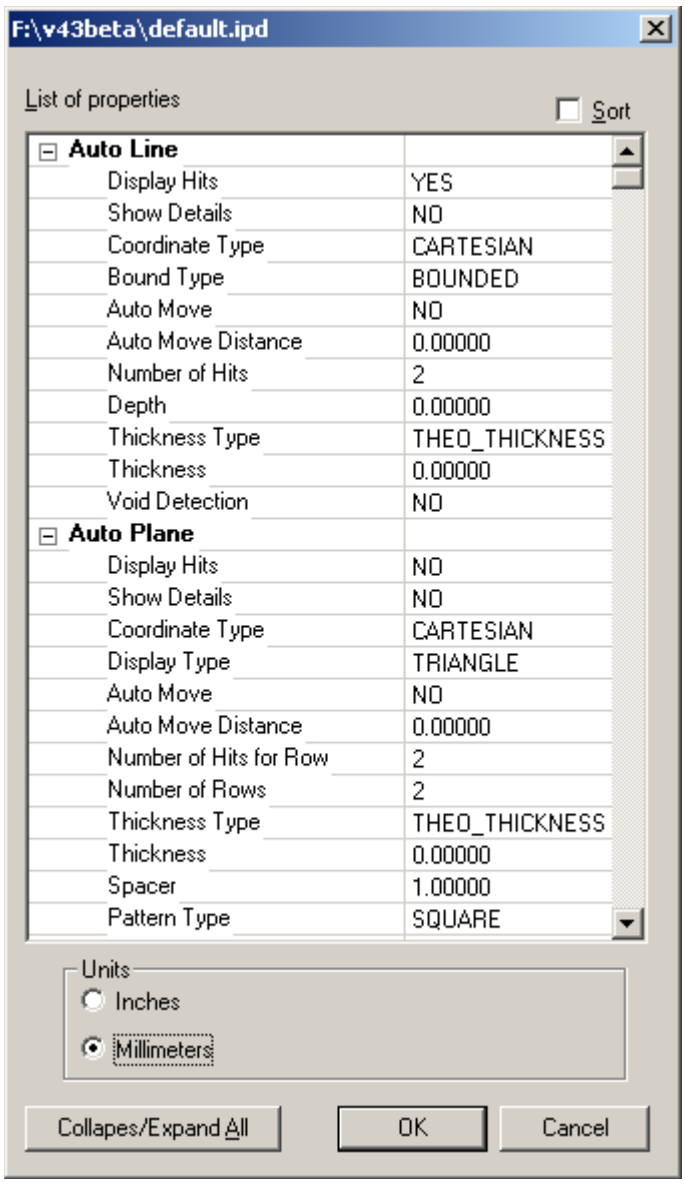
Modify Default Parameters – PC-DMIS Planner Default File

2. From the **Default Parameters File Name** drop-down select or browse to the file you want to modify or open. Click **Browse** to open a select file dialog box if the .ipd file is not already listed.



File Name list box - .ipd

3. Click **Open** to modify or open an existing .ipd file. Or if you want to create a new .ipd file, click **Create**. The **Default Parameters** dialog box opens for the selected or new .ipd file.



Default Parameters for PC-DMIS Planner

4. To edit a parameter, from the right column, click on that parameter's value. A drop-down list becomes visible.

5. Select the drop down list, and finally choose a different value. These values may represent distance, amount, or some measured value. You can select values from option lists or toggle fields.

   - Option lists provide a list of choices relevant to the associated item.
   - Toggle fields provide only two choices, such as ON/OFF, YES/NO, or TRUE/FALSE.

6. Click the **Sort** check box if you want to sort the **List of properties** alphabetically.

7. Select either **Inches** or **Millimeters** to determine how units are displayed in the **Default Parameters** dialog box. PC-DMIS will also convert imported Inspection Plans to the specified units if different from the Inspection Plan units.

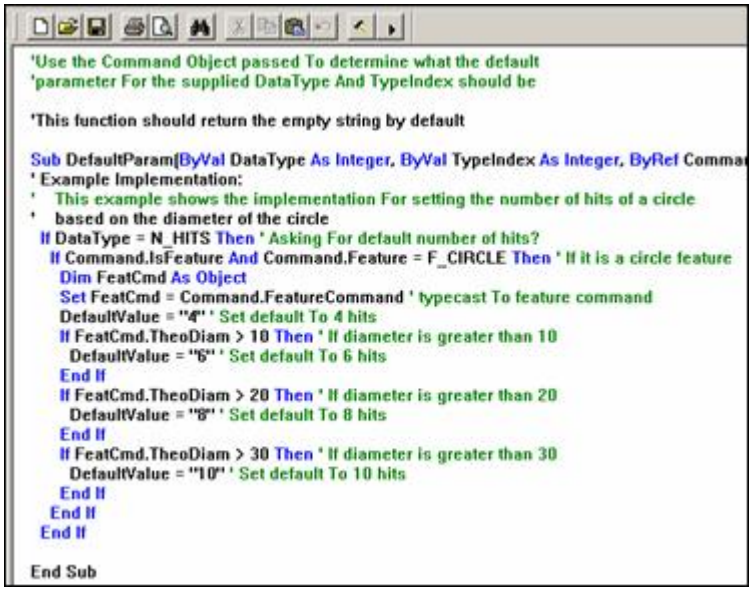
8. Click Collapse/Expand All to collapse or expand the List of properties.

Important! If you do not achieve the desired results for imported Inspection Plan items, you may need to update the needed parameter and recreate the .ipd file. Note that .ipd files from different versions of PC-DMIS are not exchangeable. For example, a .ipd created for PC-DMIS 4.3 is not compatible with PC-DMIS 2009.

9. Click **OK** to open the **Save As** dialog box, or **Cancel** to abort any changes that have been made.
10. Click **Save** to save the .ipd file.
11. Click **Set as Default** if you want to make the file listed in the **Default Parameters File Name** box the default file.
12. Click **Exit** when finished.

PC-DMIS Planner Rules (.ipr) script

.ipr scripts are basic scripts that determine how features are measured within your part program. This script is applied as your Inspection Plan is imported into your part program.
The PC-DMIS **Basic Script Editor** can be used to create **.ipr** scripts.



```
'Use the Command Object passed To determine what the default
'parameter For the supplied DataType And TypeIndex should be

'This function should return the empty string by default

Sub DefaultParam(ByVal DataType As Integer, ByVal TypeIndex As Integer, ByRef Comma
' Example Implementation:
'   This example shows the implementation For setting the number of hits of a circle
'   based on the diameter of the circle
  If DataType = N_HITS Then ' Asking For default number of hits?
    If Command.IsFeature And Command.Feature = F_CIRCLE Then ' If it is a circle feature
      Dim FeatCmd As Object
      Set FeatCmd = Command.FeatureCommand ' typecast To feature command
      DefaultValue = "4" ' Set default To 4 hits
      If FeatCmd.TheoDiam > 10 Then ' If diameter is greater than 10
        DefaultValue = "6" ' Set default To 6 hits
      End If
      If FeatCmd.TheoDiam > 20 Then ' If diameter is greater than 20
        DefaultValue = "8" ' Set default To 8 hits
      End If
      If FeatCmd.TheoDiam > 30 Then ' If diameter is greater than 30
        DefaultValue = "10" ' Set default To 10 hits
      End If
    End If
  End If

End Sub
```

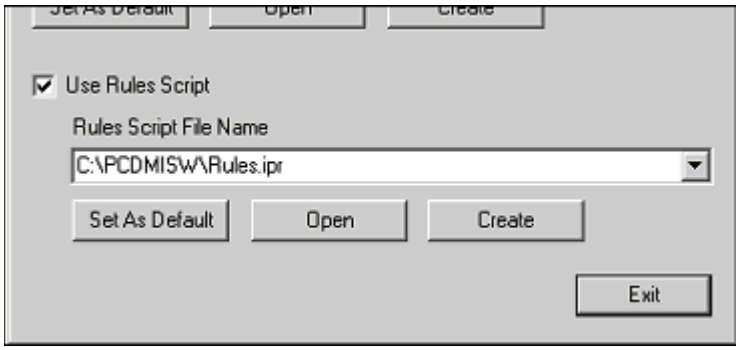
Basic Script Editor containing sample code for Planner rules

The example in the above figure shows rules that will change the number of hits that are taken depending on the diameter of the circle.

To make changes to the PC-DMIS Planner Rules (.ipr) script, follow this procedure:

1. Select the Edit | Setup | Modify Default Parameters menu option. The Modify Default Parameters dialog box appears.

   The **Modify Default Parameters** dialog box is password protected if password protection has been enabled for other settings in PC-DMIS. See the "Setup Options: General Tab" topic in the core PC-DMIS documentation for more information.



Modify Default Parameters – Rules Scripts Section

2. Select the **Use Rules Script** check box.
3. From the **Rules Script File Name** drop-down select or browse to the file you want to modify. Click **Browse…** to open a select file dialog box if the **.ipr** file is not already listed.

File Name list box - .ipr

4. Click **Open** to modify an existing **.ipr** file or click **Create** to open the default **.ipr** script. This will open the selected file in the **Basic Script Editor**. When using **Create** a new script will be populated with a default .ipr script inside of the **Basic Script Editor**.
5. Edit your **.ipr** script using the **Basic Script Editor**.
6. Click the **Save** Icon to save your finished script.
7. Click Close to exit the Basic Script Editor.

Setting the Default .ipr Script

1. Select the Edit | Setup | Modify Default Parameters menu option. The Modify Default Parameters dialog box appears.
2. From the **Rules Script File Name** drop-down select or browse to the needed file.
3. Click **Set as Default** to make the file listed in the **Rules Script File Name** box the default file.
4. Click **Exit** when finished.

Set as Priority

You can select a set of CAD surfaces for Find Nom processing using the **Set as Priority** check box. This speeds up the calculation of nominal values as it allows Planner to determine which surfaces to check first. There is no limit to the number of surfaces that can be selected. The order that the surfaces are selected in will determine the order of the search.

Whenever you open the **Edit CAD Elements** dialog box and select the **Set as Priority** check box, a surface or set of surfaces can be selected and will be tagged as priority surfaces.

Box Selecting for an Auto Feature

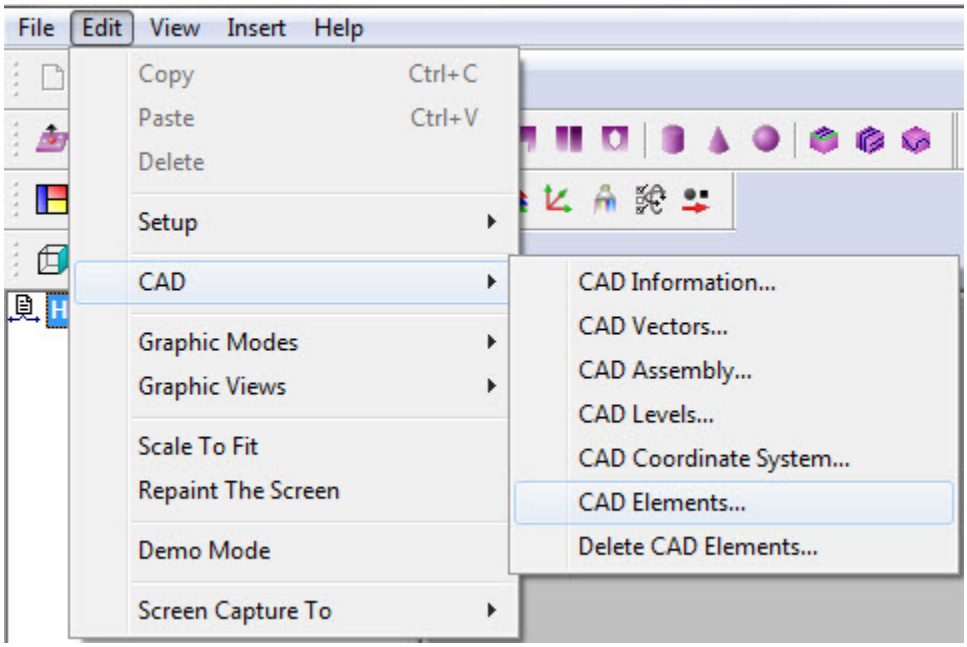
When box selecting with the Auto Features dialog open, all CAD surfaces of the feature type are selected inside the box.

The user can limit which surfaces are selected inside the box by specifying Priority Surfaces. That is, if Priority Surfaces have been defined, when box selecting from a surface auto feature for example, only the Priority Surfaces will be considered for that feature.

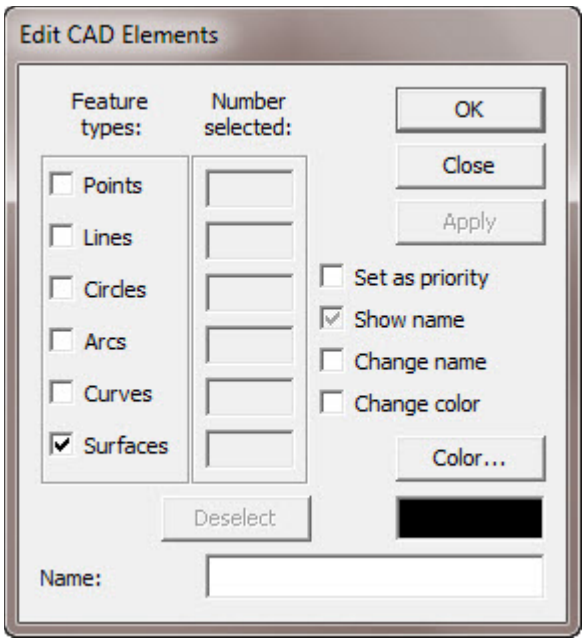
Note: Priority Surfaces are used for other functionality including finding nominals.

To set Priority Surfaces:

1. Select Edit | CAD | CAD Elements... from the Main menu



2. The Edit CAD Elements dialog is displayed. If the **Set as priority** check-box is not checked, click the check box to turn it on.



3. Add or remove surfaces to this list by selecting them in the graphics window.
4. Press the **Apply** or **OK** button to update the .prg file with the changes made to the list of Priority Surfaces.

To select a new set of surfaces

1. Open the Edit CAD Elements dialog box (Edit | CAD | CAD Elements).
2. Select the **Set as Priority** check box to turn this feature on.
3. With the **Set as Priority** check box selected, click on each feature in the graphics window to set as priority. To deselect a set of surfaces, click the **Deselect** button.
4. Click the **Apply** button.
5. Click **OK**.

Note: Planner does not store priority surfaces from one instance of Planner to the next. Once Planner is shut down, the next time it is started up, the **Set as Priority** feature will need to be redone.

Importing and Exporting Files

Before you begin creating your Inspection Plan, you should import CAD data from your CAD software package.

PC-DMIS Planner can also connect directly to your CAD software or import existing Inspection Plans. Once you have created your Inspection Plan, you can export it for use with PC-DMIS.

See "Importing CAD Data" for available CAD translators. See "Using Direct CAD Interfaces" for available DCIs.

Importing CAD Data

PC-DMIS Planner allows you import CAD using the following translators:

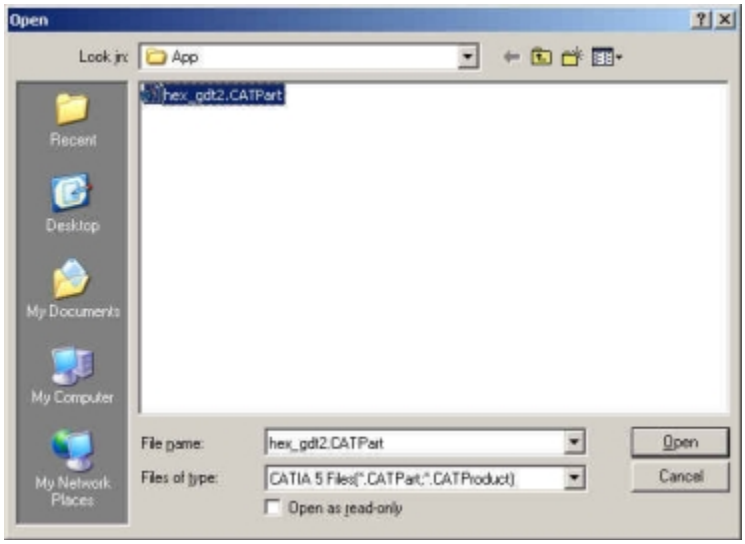
- CATIA V6 DCT
- CATIA V5 DCT
- CATIA V4 DCT

- DXF/DWG
- IGES
- Parasolid DCT
- Pro-Engineer DCT
- SolidWorks DCT
- STEP
- STL
- Unigraphics DCT
- VDAFS
- XYZ
- JT

Note: The following PC-DMIS Planner options must be enabled on your portlock to use them: **CATIA V4 DCT, CATIA V5 DCT, CATIA V6 DCT, UG DCT, PRO-E DCT**, and **Parasolid DCT**.

To import CAD data:

1. Select the **File | Import | <CAD File Type>** menu item. The Open dialog box appears.



Open dialog box

2. Browse to and select the needed CAD file.
3. Click Open to import the file. PC-DMIS Planner imports the CAD data with any accompanying GD&T/dimension information.

After saving, the default PC-DMIS Planner .cad file name is now the same as the imported CAD file.

Note: If you re-import your part's original CAD model file, any existing GD&T Cad callouts that you've added in the Graphic Display window will not be included.

Importing CATIA Files

PC-DMIS Planner allows you to translate various CATIA file types and import it as a CAD file into your inspection plan. This type of importation differs from interfacing directly with the CATIA CAD file. See "Using Direct CAD Interfaces" topic for directly interfacing with CAD models.

To import a CATIA file:

1. Open the part program in PC-DMIS into which you are going to import the CATIA file.
2. Select the **File | Import | CATIA** and select the appropriate CATIA file option from the list of CATIA file types. An **Open** dialog box will appear.
3. Choose the appropriate CATIA type from the **Files of Type** list at the bottom of the dialog box. PC-DMIS lists program files ending with extensions according to what you selected.
    - If you selected CATIA4 Files, then PC-DMIS lists files with .mod, .exp, .iso, and .cat filename extensions.
    - If you selected CATIA5 Files, then PC-DMIS lists files with .CATPart and .CATProduct filename extensions.
    - If you selected CATIA6 Files, then PC-DMIS lists files with .3dxml filename extensions.
4. Navigate through the available directories until you find the directory containing the desired CATIA file and select that file from the list.
5. Click the **Import** button. Planner imports the CAD file, and a progress bar appears indicating its progress.

CATIA v5 Assemblies

CATIA v5 assemblies can contain embedded CATIA v4 model files. When you use the CATIA v5 DCI or DCT to import a CATIA v5 assembly, Planner uses the CATIA v4 DCT to translate any embedded CATIA v4 model files. As such, your portlock must be programmed for the CATIA v4 DCT. Otherwise, the embedded CATIA v4 model files will not be translated. For information on assemblies, see "Working with Assemblies of Parts" topic.

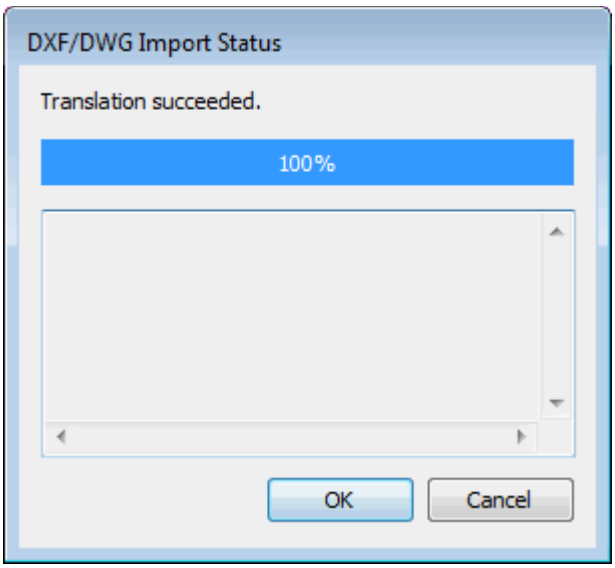
CATIA v5 Captures

PC-DMIS 2010 MR3 and later support CATIA v5 captures. For more information, see the "Working with CATIA Captures" topic.

Importing a DXF/DWG File

Note: DXF files do not contain units information. It is assumed that the imported DXF is in the same units as the part program. After importing a DXF file, if the scale is incorrect, import the DXF file into a part program with different units or use the CAD Transform dialog to scale the model to the appropriate units.

1. Select the **File | Import | DXF** menu option. An **Open** dialog box appears.
2. Select the data type you want to import from the **Files of Type** list.
3. Navigate to the directory containing the DXF file to import.
4. Click **Import** when ready to import the file. PC-DMIS will display the **DXF/DWG Import Status** dialog box.



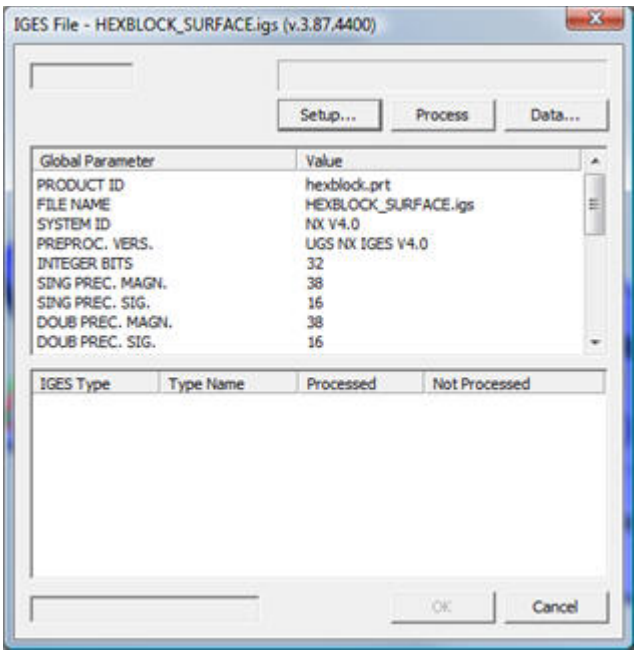
DXF/DWG Import Status dialog box

5. Click **OK** when the DXF or DWG file has been translated. If the translation fails, the **OK** button will not be available. Click **Cancel** to decline the addition of the imported file.

Note: 2D drawing dimensions are imported for Planner only. In Planner, the imported 2D drawing dimensions can be selected to add tolerances to the Inspection Plan.

Importing an IGES File

1. Select the **File | Import | IGES** menu option. An **Open** dialog box appears.
2. Navigate through the available directories to the directory containing your part file.
3. Select the file.
4. Click the **Open** button. Planner next displays the **IGES File** dialog box. The **IGES File** dialog box displays all of the pertinent information regarding the indicated data file.

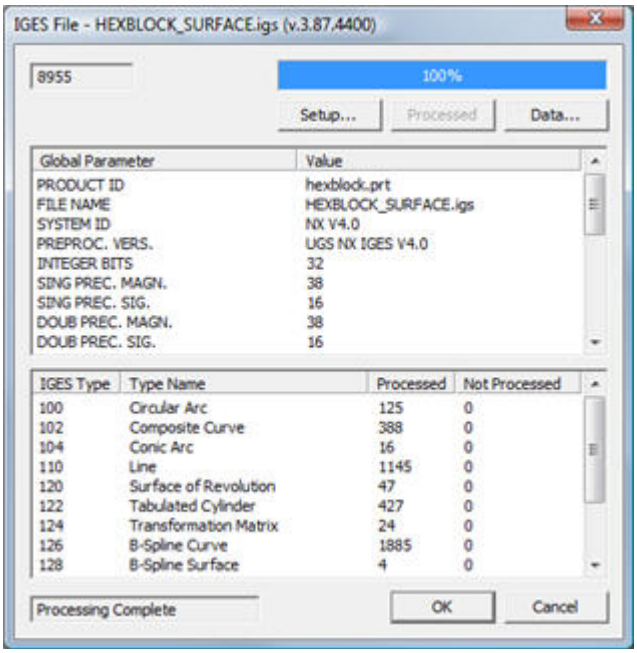


IGES File dialog box

5. If you want to determine what CAD data gets processed and displayed, click the **Setup** button (see "Altering Display of Imported CAD").

6. To view specific IGES feature data, click the **Data** button (see "Using the IGES Data Dialog Box").

7. To import this data into Planner, simply click the **Process** command button. Planner will indicate when the file is 100% processed.



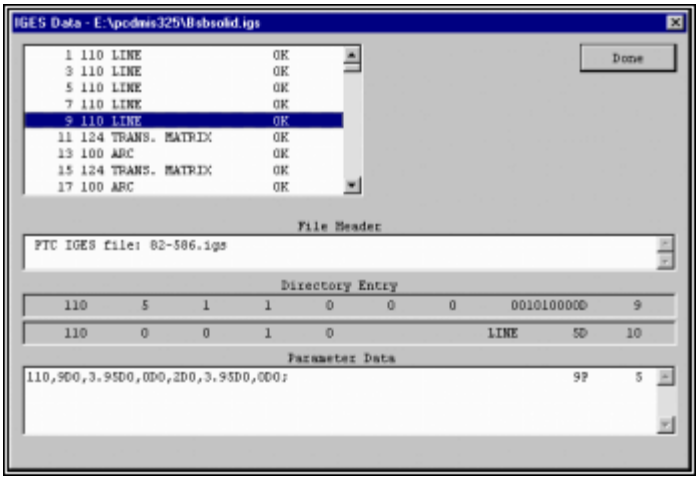
IGES File dialog box after a file is processed

8. To complete the action, click the **OK** button, and Planner will display the imported IGES model. Selecting the **Cancel** button will terminate the entire operation and close the **IGES File** dialog box.

If you want to manipulate a 2D CAD drawing three-dimensionally, thereby creating desired 3D levels, you can use the CAD levels functionality. If you do this, your original data should be defined in a single plane parallel to the Z (= 0) plane. For complete information on CAD levels, see the "Working with CAD Levels" topic in the "Editing the CAD Display" chapter.

Using the IGES Data Dialog Box

The **IGES Data** dialog box allows you to click on a specific feature in the list box to display information regarding that feature's diameter, XYZ nominal, etc.

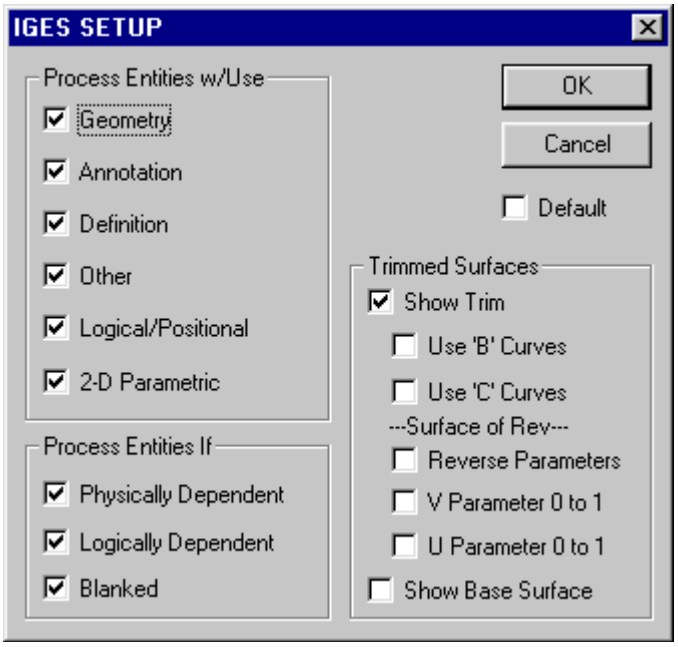


IGES Data dialog box

To view information about the CAD data you're importing inside the **IGES Data** dialog box, follow this procedure:

1. Begin importing a CAD or IGES file (see "Importing an IGES File").
2. Once the **IGES File** dialog box appears, click **Process** to import the data.
3. Once it's complete, click the **Data** button. The **IGES Data** dialog box appears.

Altering the Display of Imported CAD



IGES Setup dialog box

To access the **IGES Setup** dialog box and alter how the IGES image is displayed:

1. Begin importing your CAD file (see "Importing an IGES File").
2. When the **IGES File** dialog box appears, before clicking **Process**, click the **Setup** button.
3. Select the desired check box options from the **IGES SETUP** dialog box. See "Understanding the IGES Setup Options" for information on the different check boxes.
4. Click the **OK** button.

Planner will draw the selected CAD image with the requested features.

Understanding the Available Options

The **Surface of Revolution** check box is available if trimmed surfaces are being used that have a surface of revolution as a base surface and it is not being read in correctly. If this is the case, simply select the appropriate options in the **Trimmed Surface** area of the dialog box.

If the **Default** check box is selected, Planner will use the current setting to process all future entities.

The 'Process Entities with Use' Area

The **Process Entities with Use** area contains the following six check boxes that allow you to select the possible CAD image setup options.

These options are described below:

Geometry

Determines whether or not to import the geometric entities from the file. Geometric entities are the curves and surfaces that define the model.

Annotation

Determines whether or not to import the annotation entities from the file. These entities are used to add annotation or description to a file. These include angular dimension, diameter dimension, flag note, general label,

linear dimension, ordinate dimension, point dimension, radius dimension, general symbol, and sectioned area entities.

Definition
Determines whether or not to import the definition entities from the file. These entities are used in definition structures of the file. These include general note, leader, associativity definition, line font definition, macro definition, subfigure definition, text font definition, text display template, and network subfigure definition entities.

Other
Determines whether or not to import the other entities from the file. These entities are used for other purposes such as defining features in the file. Currently, these include the color definition entity and the dimensioned geometry associatively.

Logical/Positional
Determines whether or not to import the logical/positional entities from the file. These entities are used as logical or positional reference by other entities. Currently, Planner does not process any entities that are of this type.

2D Parametric
Determines whether or not to import the 2-D parametric entities from the file. These entities are positioned in two-dimensional XY parameter space, and are considered a subset of three-dimensional XYZ space, by ignoring the Z coordinate. This entity type is intended for use in defining trimming curves on surfaces. Currently, Planner ignores this check box. For more information, see the "The 'Trimmed Surfaces' Area" chapter.

If you select all six check boxes, then all IGES entities in the IGES file will be imported.

The 'Process Entities If' Area

The **Process Entities If** area contains check boxes that allow you to process certain entities when they meet certain conditions. The following describes the available conditions that are met and the description of each check box:

Process entities if Physically Dependent
Determines whether or not the base surface of a Curve on Parametric Surface will be shown.

Process entities if Logically Dependent
Determines whether or not logically dependent entities will be imported from the IGES file.

Process entities if Blanked
Determines whether or not blanked entities will be imported.

The 'Trimmed Surfaces' Area

The **Trimmed Surfaces** area contains check boxes that you may use to properly process Trimmed Surfaces.

Show Trim check box
Shows the trimmed surface.

Show Base Surface check box
Shows the untrimmed surface (base surface).

Use ‘B’ Curves check box
If ‘B’ curves are present in the IGES file, select this check box to use them as the trim definition. You may also need to select this check box if the model has poorly defined ‘C’ trimming curves. By selecting this check box, the ‘C’ trimming curves will be regenerated from the ‘B’ trimming curves.

Use ‘C’ Curves check box
If ‘C’ curves are present in the IGES file, select this check box to use them as the trim definition. You may need to use this option if the model has poorly defined ‘B’ trimming curves. By selecting this check box, the ‘B’ trimming curves will be regenerated from the ‘C’ trimming curves.

If you select the **Show Trim** check box, the other check boxes under it become available for selection. You can use either 'B' or 'C' curves as your trim definition, depending on the attributes of the trimmed surface.

Note: Be aware that if you use 'C' curves as your trim definition, Planner will have to calculate the 'B' curves that can be quite time consuming.

Also, if the IGES file contains trimmed surfaces that have a surface of revolution as a base surface these check boxes become available for selection:

Reverse Params check box
Determines whether or not to fix ‘B’ trimming curves that reverse the U and V parameters.

V Param 0 to 1 check box
Determines whether or not to fix the ‘V’ parameter that gets defined in the range 0 to 1, instead of in radians.

U Param 0 to 1 check box
Determines whether or not to fix the 'U' parameter that gets defined with a range of 0 to 1 instead of in radians when the generator of the surface of revolution is an arc.

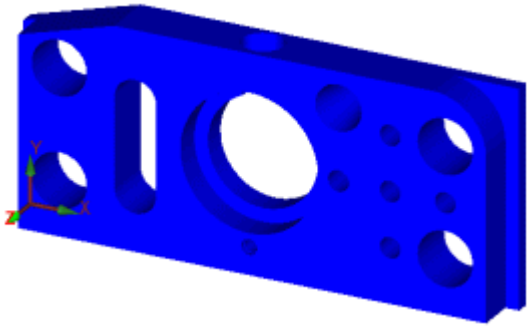
Show Base Surface check box
Determines whether or not to show the untrimmed surface.

Defining Trimming Curves

You can define trimming curves in two ways:

- **Using 3-dimensional coordinates in model space (the coordinates the surface is defined in).** These are known as ‘C’ trimming curves because each coordinate requires three values to position it in model space (1, 2, 3; A, B, C). By definition, the curve must be defined so that it geometrically coincides with the surface. In other words, the curve must lie on the surface. However, a poorly defined ‘C’ trimming curve may break this rule.
- **Defining the position of the curve using relative positions on the surface.** These are known as ‘B’ trimming curves because each coordinate requires two values to position it in parametric space. For example, a rectangular surface could have the parametric position (0,0) at one corner and (1,1) at the opposite corner. Every position on the surface could then be represented using two-value pairs between these two extremes. The parametric position (0.5,0.5) would be in the center of the surface. The parametrically defined curve is represented using these 2-dimensional parameter-space points. By definition, the curve must be defined so that it does not go outside the parameter-space of the surface it lies on. However, a poorly defined surface may break this rule.

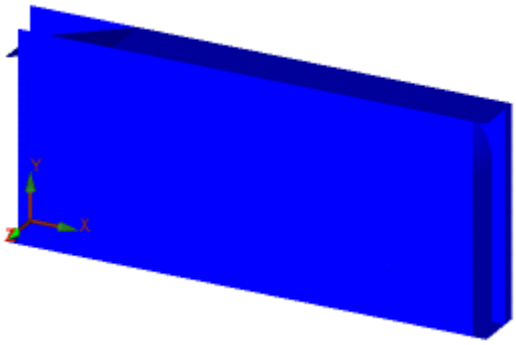
Example of Showing Trim



*IGES File with **Show Trim** check box selected*

Example of not Showing Trim

Using the IGES file in the "Example of Showing Trim" topic above, if you deselect the **Show Trim** check box, you will end up with an empty model. This is because this particular IGES file contains *only* trimmed surfaces. However, if you deselect the **Show Trim** check box and select the **Show Base Surface** check box, this is the result:



*IGES File with **Show Trim** check box not selected*

Notice that all the holes in the surfaces are gone and that the outer boundaries of the surfaces are not trimmed.

Importing a SolidWorks File

PC-DMIS Planner allows you to translate SolidWorks file types and import them into a Planner part program.

Note: The SolidWorks DCT option must be programmed in your PortLock in order for the SolidWorks option to be available.

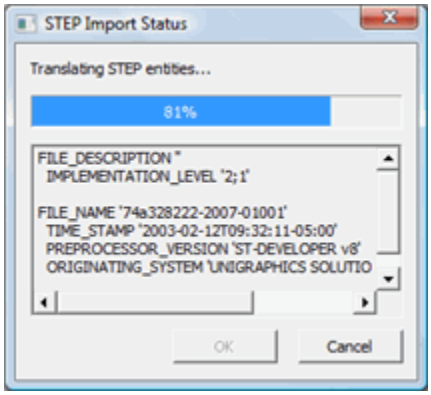
To import a SolidWorks file:

1. Open the PC-DMIS Planner inspection program that you are going to import the SolidWorks file into.
2. Select **File, Import, SolidWorks** from the main menu. An **Open** dialog box will appear.
3. The SolidWorks file types are shown in the **Files of Type** list at the bottom of the dialog box. Planner lists program files ending with extensions according to what is currently supported which are:
    - File types ending in .sldprt.
    - File types ending in .sldasm.
4. Navigate through the available directories until you find the directory containing the desired SolidWorks file and select that file from the list.

5. Click the **Open** button. PC-DMIS Planner imports the SolidWorks file, displaying a progress bar to indicate the progress of the import.

Importing a STEP File

1. Select the **File | Import | STEP...** menu option. An **Open** dialog box appears.
2. Select the data type you want to import from the **Files of Type** list.
3. Navigate to the directory containing the file to import and select the file.
4. Click **Open** when ready to import the file. Planner will display the **STEP Import Status** dialog box. This dialog contains a status bar on the top part of the dialog box and an information box on the bottom half. This information box displays header data contained within the imported STEP file.

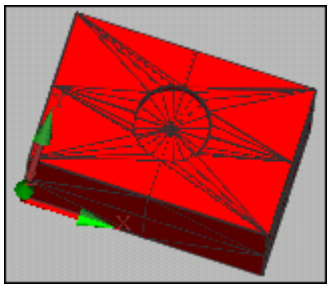


STEP Import Status dialog box

5. Click **OK** when the STEP file has been translated. If the translation fails, the OK button will not be available. Click **Cancel** to decline the addition of the imported file.

Importing an STL File

The stereolithography (STL) format, an ASCII or binary file used in manufacturing, provides a list of the triangular facets that describes a computer generated solid model. This format is the standard input for most rapid prototyping machines.



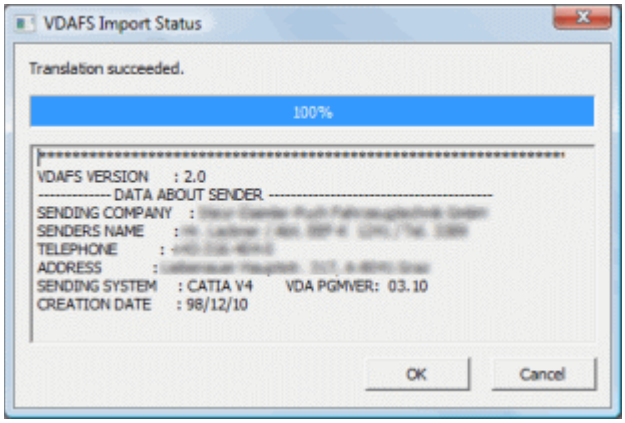
Sample Imported STL File

To import an STL file,

1. Select the **File | Import | STL...** menu option. An **Open** dialog box will appear.
2. Navigate through the available directories until you find the directory containing the desired STL file and select that file. The STL file can be in ASCII or binary format.
3. Click the **Open** button. Planner imports the data as a list of facets that form a solid model.

Importing a VDAFS File

1. Select the **File | Import | VADFS...** menu option. An **Open** dialog box appears.
2. Select the data type you want to import from the **Files of Type** list.
3. Navigate to the directory containing the file to import and select that file.
4. Click **Open** when ready to import the file. Planner will display the **VDAFS Import Status** dialog box. This dialog contains a status bar on the top part of the dialog box and an information box on the bottom half. This information box displays header data contained within the imported VDAFS file.



VDAFS Import Status dialog box

5. Click **OK** when the VDAFS file has been translated. If the translation fails, the **OK** button will not be available. Click **Cancel** to decline the addition of the imported file.

A Note on Importing Point Data from PSET (Point Set) Objects

If the .VDA file you are importing contains point data within PSET Objects, Planner may end up creating a .cad file with fewer CAD entities than what is actually displayed on the screen.

To get a number of CAD entities equivalent to the number of displayed points on the Graphic Display window, follow this work-around:

1. Close down Planner.
2. Start the PC-DMIS Settings Editor.
3. Expand the **ImportSettings** section.
4. Set `ExplodePolylines` to TRUE.
5. Set `KeepPolylines` to FALSE.
6. Save your changes to the registry, close the Settings Editor, and restart Planner.
7. Import the .VDA file again.

For information on modifying registry entries, see the help documentation associated with the PC-DMIS Settings Editor.

Importing an XYZIJK File into CAD

XYZIJK Files have a .xyz extension. You can view the values contained in a .xyz file by using any standard text editor.
XYZIJK file types are simple files with feature information in the form of xyz coordinates with or without a vector.

To import an XYZIJK file into the Graphic Display window:

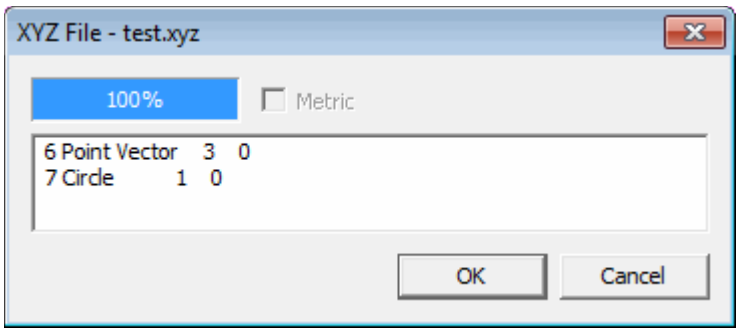
1. Select the **File | Import | XYZ into CAD...** menu option. An **Open** dialog box appears.
2. Navigate to the directory containing the file to import.
3. Click **Open** when ready to import the file. Planner interprets and imports the selected XYZIJK file into the Graphic Display window and draws the imported CAD features.

How many numbers there are on each line determine what feature is created. Planner reads the rows of the .xyz data file and converts the points to the following feature depending on the amount of numbers per row:

Numbers per Row	Description	CAD Feature Generated
3	First number = X value Second number = Y value Third number = Z value	Point with no vector
4	First number = X value Second number = Y value Third number = Z value Fourth number = diameter	CAD circle with a workplane based vector
5	First number = X value Second number = Y value Third number = Z value Fourth number = nothing Fifth number = nothing	Point with no vector
6	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value	Point with a vector
7	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value Seventh number = diameter	Circle with vector
8+	First number = X value Second number = Y value Third number = Z value Fourth number = I value Fifth number = J value Sixth number = K value Seventh number = nothing Eighth number = nothing	Point with a vector

Example: If your XYZ file contained a line with `25,280,750,25` Planner would create a circle with the centered at x=25, y=280, and z=750 with a diameter of 25.

After selecting the file to import, Planner displays the features constructed from the XYZ file in the **XYZ File** dialog box.



XYZ File dialog box

Note: Be aware that the order in which the features are created in the .xyz file and the order in which they are generated inside Planner may not be the same.

After clicking **OK**, Planner displays the created CAD features in the Graphic Display window.

Notes on XYZIJK Files

Planner will read in any text file that contains XYZ (and possibly IJK) data. The file should contain the nominal (theoretical) inspection points that need to be measured. The file must follow these rules:

1. Columns 1 - 6 in the first line of the file must contain the characters "XYZIJK". This is the keyword that lets Planner distinguish this file type from DES and IGES files. *It must be in capital letters*.

   In the first line of the file:
   Col. 1 = X
   Col. 2 = Y
   Col. 3 = Z
   Col. 4 = I
   Col. 5 = J
   Col. 6 = K

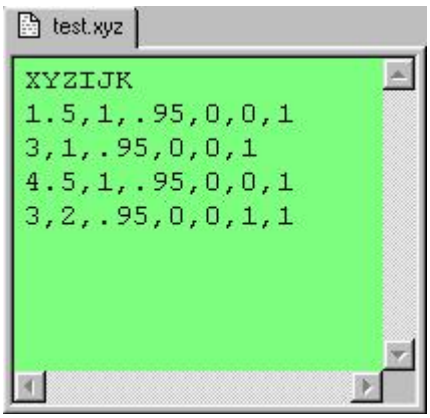
2. If the file contains metric data, columns 11 - 16 of the first line must contain the word METRIC. That is:

   Col. 11 = M
   Col. 12 = E
   Col. 13 = T
   Col. 14 = R
   Col. 15 = I
   Col. 16 = C

The default, if the METRIC keyword is not present, is INCH.
The rest of the first line should be blank

3. Each line thereafter *must* contain three to eight floating point numbers *separated by a comma*.

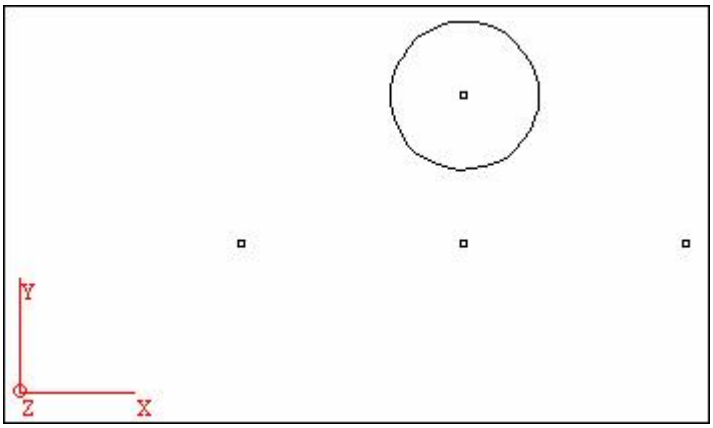
For example, suppose your XYZIJK file looked like this sample:



Sample XYZIJK file in a text editor

- Line 1 contains the XYZIJK keyword, and since the keyword METRIC isn't on columns 11 through 16, this defaults to INCH.
- Line 2 produces a point with a vector.
- Line 3 produces a point with a vector.
- Line 4 produces a point with a vector.
- Line 5 produces a circle with a vector and a diameter.

Once imported, it will produce this result in the Graphic Display window:



Graphic Display window showing the data imported from the sample XYZIJK file

Importing a JT File

The JT file format is a relatively light-weight file format capable of containing large assemblies of parts with faceted data, lighting, textures, curves and surfaces, product and manufacturing information (GD&T callouts) and other information. It is useful for visualization and collaboration.

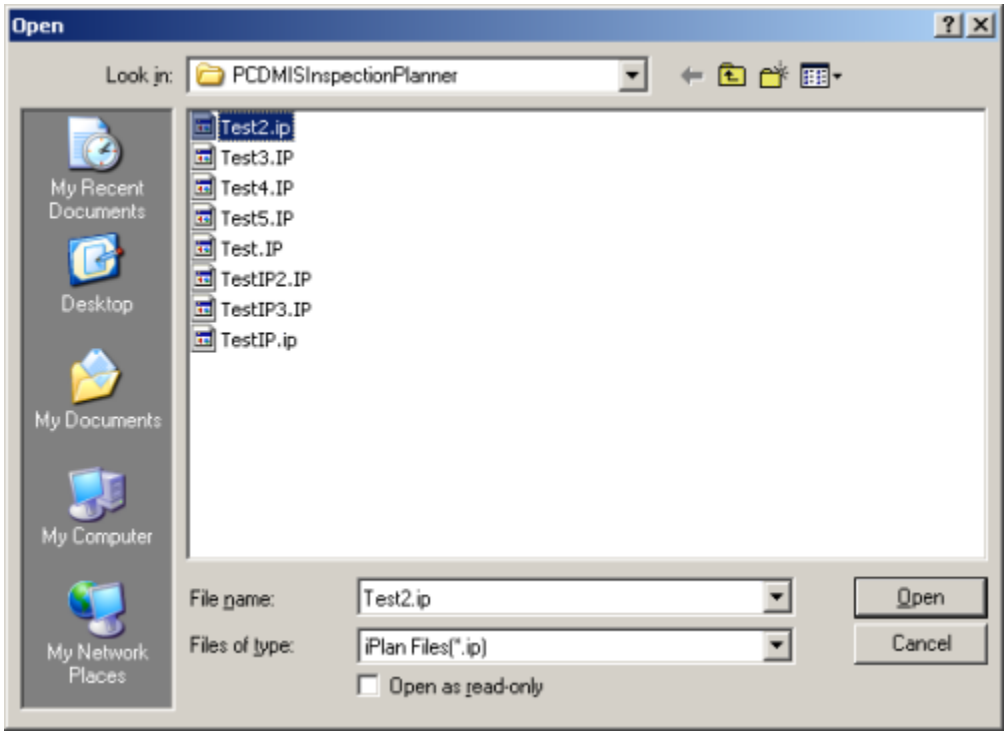
To import a JT file,

1. Select the **File | Import | JT...** menu option. An **Open** dialog box will appear.
2. Navigate through the available directories until you find the directory containing the desired JT file and select that file from the list.
3. Click the **Open** button. PC-DMIS imports the CAD data from the JT file.

Importing an Inspection Plan

To import an existing Inspection Plan:

1. Select the **File | Import | Inspection Plan** menu item. The **Open** dialog box appears letting you select a file with an ".ip" filename extension.

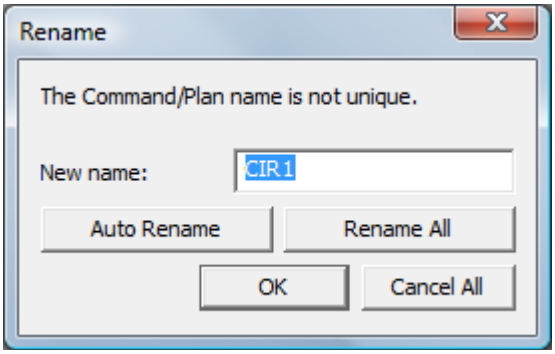


Open dialog box

2. Browse to and select the needed .ip file.
3. Click **Open** to import the file. PC-DMIS Planner imports the Inspection Plan.

Using the Rename dialog box to Ensure Unique IDs

If you attempt to import an inspection plan file, Planner expects imported command names and plan names to be unique. If the .ip file contains commands with identical names to any commands inside the **Plan** list, Planner will display a **Rename** dialog box, allowing you to rename any conflicting IDs:



Rename dialog box

New name - Lets you manually type a new ID to use for the imported ID.

Auto Rename - This automatically appends an underscore character to the conflicting ID and uses that to generate a unique ID.

Rename All - This automatically appends underscores as needed to all conflicting IDs.

OK - Applies the unique name specified in the **New name** box for conflicting ID.

Cancel All - Does not add or rename any conflicting ID into the **Plan** list.

Importing an XYZIJK File into an Inspection Plan

To import an XYZIJK file into an inspection plan:

1. From the Plan window, select the plan into which you want to import the features.
2. Select the **File | Import | XYZ into Plan...** menu option. An **Open** dialog box appears.
3. Navigate to the directory containing the file to import.
4. Click **Open** when ready to import the file. Planner interprets and imports the selected XYZIJK file into the selected plan.
5. Planner displays the features added to the plan in the **XYZ File** dialog box.

Note: While this procedure deals specifically with importing XYZIJK features into the inspection plan, the information on XYZIJK files discussed in the "Importing an XYZIJK File into CAD" topic is applicable here as well.

Using Direct CAD Interfaces

Direct CAD Interfaces (DCI) allow you to connect directly to your CAD software. As the CAD file is updated in your CAD software, the corresponding changes are reflected in PC-DMIS Planner. See the "Direct CAD Interface" documentation for detailed information.

Instead of translating CAD data into the internal CAD format of PC-DMIS, the Direct CAD file types are accessed directly. That is, the CAD data of these file types remains in its native format after being imported. Routines native to the CAD type are used for all calculations done on the CAD data. Accessing the CAD data directly eliminates problems that can occur because of translation inaccuracies or limitations.

The "Direct CAD Interfaces" documentation discusses importing Direct CAD Files in detail.

PC-DMIS Planner supports the use of the following DCIs:

- CATIA 5
- Pro/Engineer (including Creo)
- Solidworks
- Unigraphics

Note: The following **PC-DMIS Planner** options must be enabled on your portlock to use them: **CATIA V5 DCI**, **UG DCI**, **PRO-E DCI**, and **Solidworks DCI**.

Exporting an Inspection Plan

It is not necessary to export an Inspection Plan (.ip file) if you have released plans with the saved PC-DMIS Planner .cad file. The PC-DMIS Planner .cad file contains any embedded Inspection Plans that have been released. Otherwise, exporting a plan for use in PC-DMIS is another method to make Inspection Plans available for PC-DMIS.

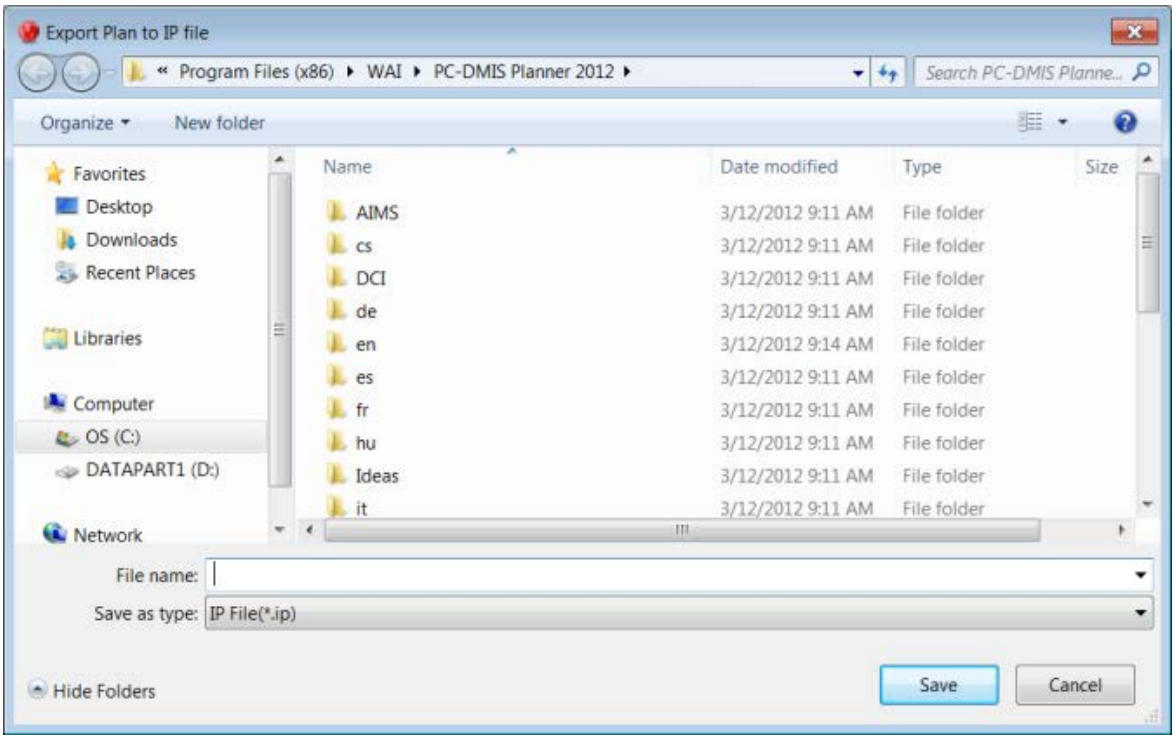
When importing CAD files into PC-DMIS, only plans that were released in Planner are visible. This allows you to be sure that the plans you see are only those that are meant to be seen.

To export a new PC-DMIS Inspection Plan, follow this procedure:

1. Select the needed plan from the Plan list, and do one of the following:

   - Click **Export Plan** from the **Tools** toolbar.
   - Select the File | Export | Inspection Plan... menu item.
   - Right-click the needed plan, and select the **Export Plan** option from the shortcut menu.

   The **Export Plan to IP** dialog box appears.



Export Plan to IP dialog box

2. Browse to the location where you would like to export the file.
3. Type a name for the PC-DMIS Planner .ip file in the **File name** box.
4. Click **Save** to export the file. This text file can be imported into PC-DMIS. See "Importing an Inspection Plan".

Using Command Line Options

PC-DMIS Planner lets you launch the application and load a cad model into a new "Untitled" inspection plan, by using simple command line switches. These switches can be used with shortcuts, batch files, or command window prompts to help save time.

/DCTLOAD to load a supplied part model using the direct cad translators. You must specify the complete path and filename of the model before this switch.
/DCILOAD to load a file using Direct Cad Interfaces (DCI). You must specify the complete path and filename of the model before this switch.
/ISPROE to specify that the .prt file will be used in ProE (both UG and ProE use the same .prt filename extension). This follows the /DCILOAD switch.

DCI/DCT Examples

- This command line launches PC-DMIS and loads a .igs cad model using PC-DMIS Planner's Direct CAD Translators:
  ```
  C:\Program Files\Wai\Pcdmis Planner\InspectionPlanner.exe
  /DCILOAD:d:\partmodels\hexblock_wireframe_surface.igs
  ```
- This command line launches PC-DMIS and loads a .prt cad model from the UG DCI:
  ```
  C:\Program Files\Wai\Pcdmis Planner\InspectionPlanner.exe
  /DCILOAD:d:\partmodels\hexblock.prt
  ```
- This command line launches PC-DMIS and loads a .prt cad model as a ProE DCI:
  ```
  C:\Program Files\Wai\Pcdmis Planner\InspectionPlanner.exe /DCILOAD:d:\partmodels\proe.prt
  /ISPROE
  ```

Importing a CSV File into Planner

Note: The information from a CSV file can be imported only into the plan.

To import information from a CSV file into Planner, navigate to **File | Import | CSV** from the menu option.

Each row in the CSV file represents a feature and a feature command will be created for each row of data. If tolerance values are specified, a location dimension command is also added to the plan.

The first row of the CSV file contains the format designators. This designator row indicates the data for each column. There are predefined column headers. Columns that do not include a format designator or that have an unrecognized column header are ignored. Columns can be in any order. If the “Type” column is missing or if there is no feature type specified for a row of data, this row of information will be treated as a “Point” feature.

If a row has only an asterisk in column 1 of the first column with no other information in the row, this is a signal that the next row will again be a format designator row. This allows the user to switch formats inside a single CSV file.

File Format

A CSV file is a **C**omma **S**eparated **V**alues file that can be created by many software packages, including Microsoft Excel. Here is an example of an Excel table used to create a CSV file:

TYPE	NAME	X	Y	Z	X2	y2	z2	D	A	L
Point	PNT1	67.98	89.88	13.99						
Line	LIN1	51.75	0	-13.1	62.25	0	-13.1			
Plane	PLN1	100.25	0	-25.6						
Circle	CIR1	124	50	0				60.5		
Ellipse	ELL1	179.9	10	0						14.125
SlotR	SLTR1	191.05	72.998	0				7.02		33.045
SlotS	SLTS1	69.7	23.35	0				11.041		12.018
SlotN	SLTN1	76.5	6	0				6		12
Polygon	POL1	26.35	49.5	15				22		
Cylinder	CYL1	124	50	0				60.5		
Cone	CON1	69	90	14					30	14
Sphere	SPH1	188.5	45.65	0				12.75		
*										
type	name	x	y	z	i	j	k			
point	ppnt1	0	10	0	0	0	1			
point	pnt2	0		0	0	0	1			
plane	ppln	0	10	20	1	0	0			

Partial spreadsheet table example 1 used to create a CSV file - continued below

D	A	L	W	i	j	k	I2	J2	K2	+TOL	-TOL
				0	0	1				0.05	0.05
				1	0	0				0.05	0.05
				0.707	0.707	0				0.05	0.05
60.5				0	0.707	0.707				0.05	0.05
		14.125	7.23	0.707	0	0.707	1	0	0	0.05	0.05
7.02		33.045	12.8	0.707	0	0.707	1	0	0	0.05	0.05
11.041		12.018	4.2	0.707	0	0.707	1	0	0	0.05	0.05
6		12	3.5	0.707	0	0.707	1	0	0	0.05	0.05
22				0.707	0	0.707	1	0	0	0.05	0.05
60.5				0.707	0	0.707				0.05	0.05
	30	14		0.707	0	0.707				0.05	0.05
12.75				0	0	1				0.05	0.05

Continuation of the spreadsheet table example from above used to create a CSV file

Download the above CSV example file here: CSV_Example_File.csv

Another example of a spreadsheet table used to create a CSV file is shown below:

TYPE	NAME	X	Y	Z	+TOL	-TOL
Point	X48a001	68	90	14	0.05	0.05
Point	X48a002	74	85	12	0.05	
Point	X48a006	76	84	11		
Point	X48a011	79	83.5	10.75	0.035	0.035
Point	X48a021	85	83.25	10.67	0.035	0.035
Point	X48a022	88	82.375	10.5	0.035	0.035
Point	X48a029	97	82.125	10.375	0.05	0.05
*						
TYPE	NAME	X	Y	D	+TOL	-TOL
CIRCLE	X48b989	124	50	12	0.065	0.065

Spreadsheet table example 2 used to create a CSV file

Note that in the second example, row two is a point feature and that both +TOL and –TOL are specified. Both a point feature command and a location dimension command will be added to the plan based on this row of information.

Neither +TOL nor –TOL are specified in the point feature in row four. Only a feature command will be added to the plan. A location dimension command will not be added to the plan.

Row nine includes no data except for an asterisk in column 1. This indicates that the next row, row ten, will include a new format designator row.

Format Designators

The following is a list describing the valid entries allowed when creating a CSV file that will be imported into Planner.

TYPE – this is the feature type. The legal entries for this column are: **Point**, **Line**, **Plane**, **Circle**, **Ellipse**, **SlotR**, **SlotS**, **SlotN**, **Polygon**, **Cylinder**, **Cone** and **Sphere**.

NAME – Feature name

X - Feature's nominal **X**-value

Y - Feature's nominal **Y**-value

Z - Feature's nominal **Z**-value

X2 - Feature's **second** nominal **X**-value

Y2 - Feature's **second** nominal **Y**-value

Z2 - Feature's **second** nominal **Z**-value

D - Feature's nominal **D**iameter

A - Feature's nominal **A**ngle in degrees

L - Feature's nominal **L**ength

W - Feature's nominal **W**idth

i - Feature's nominal **i**-vector

j - Feature's nominal **j**-vector

k - Feature's nominal **k**-vector

i2 - Feature's **second** nominal **i**-vector

j2 - Feature's **second** nominal **j**-vector

k2 - Feature's **second** nominal **k**-vector

+TOL - Feature's **plus TOL**erance

-TOL - Feature's **minus TOL**erance

File Import to Planner

The following describes the sequence of events that occur when a CSV file is being imported into Planner:

1. Each row of data is read into Planner from the CSV file.
2. The information is matched with the previously defined column headers to determine the meaning to each item of information.
3. Based on the feature type the information is used to create a feature command in the plan.
4. If one or both tolerance values are specified, a location dimension command is also added to the plan.

Working with the Graphics Display Window

There are many tools available to adjust the way CAD graphics are displayed, selected and edited in the Graphic Display window.

Changing the Graphic Modes

The **Graphic Modes** toolbar allows you to change the attributes and functionality of the **Graphic Display Window**.

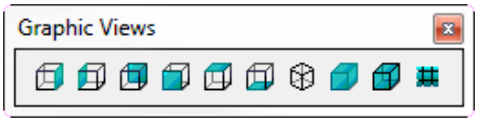
Graphic Modes Toolbar

You may also toggle the display for surface, wire frame and window grid under the **Edit | Graphic Modes** submenu and the viewing angles under the **Edit | Graphic Views** submenu.

See the "Graphic Modes Toolbar" topic for a description of the different graphics modes.

Changing the Graphic Views

The **Graphic Views** toolbar allows you to quickly change how your part is displayed inside the main view of the Graphic Display window.



Graphic Views Toolbar

The following buttons are available:

- Displays X+ View

- Displays Y+ View

- Displays Z+ View

- Displays X- View

- Displays Y- View

- Displays Z- View

- Displays ISO View

- Displays Toggle Solid View

- Displays 3D Grid View

You may also toggle the viewing angles under the **Edit | Graphic Views** submenu and the display for surface, wire frame and window grid under the **Edit | Graphic Modes** submenu.

See "Graphic Views Toolbar".

Repainting the Screen

To refresh the Graphic Display window and redraw all the views of the part, select the **Edit | Repaint The Screen** option or the button on the Graphic Modes toolbar.

Changing Screen Modes

PC-DMIS Planner allows you to switch between these Screen Modes. The different modes tell Planner how to interpret mouse clicks.

Translate Mode

The **Edit | Graphic Modes | Translate** option places PC-DMIS Planner into Translate Mode. This mode allows you to move and zoom in or out on the part while in the Graphic Display window. The following functions are available in the Translate Mode:

- Shrink the size of the part within the Graphic Display window.
- Enlarge the size of the part within the Graphic Display window.
- Select a portion of the part within the Graphic Display window.
- Reposition the part within the Graphic Display window.

Shrinking the Part Image within the Graphic Display window

To shrink (or zoom out) the entire image of the part by clicking:

1. Move the mouse pointer to a point *above* the imaginary, horizontal centerline of the graphic part display.
2. Click the right mouse button.

The farther away the cursor is from the centerline, the more the drawing will shrink.

If you have a mouse with a mouse-wheel you scroll your mouse wheel away from you to shrink the drawing. PC-DMIS centers the zoom-out operation at the mouse pointer's location.

Enlarging the Part Image within the Graphic Display window

To enlarge (or zoom in on) the entire image of the part:

1. Move the mouse pointer to a point *below* the imaginary, horizontal centerline of the graphic part display.
2. Click the right mouse button.

The farther away the cursor is from the centerline, the more the drawing will grow.

If you have a mouse with a mouse-wheel you scroll your mouse wheel towards you to enlarge the part. PC-DMIS centers the zoom-in operation at the mouse pointer's location.

Enlarging a Portion of the Part Image within the Graphic Display window

To enlarge a specified section of the displayed part:

1. Place the mouse pointer at a corner of the selection area.
2. Hold *both* left and right mouse buttons simultaneously.
3. Move (drag) the arrow icon over the portion of the Graphic Display window to be selected. PC-DMIS will begin drawing a box.
4. When the box contains the proper geometry, release both mouse buttons. PC-DMIS will zoom in on the selected area.

Note: Once you reach a certain point, the image doesn't enlarge any further.

Repositioning the Part Image within the Graphic Display window

To change the position of the part image within the Graphic Display window*:*

1. Place the mouse pointer over the image of the part.

2. Hold down the right mouse button and drag it to a new position.
3. Release the mouse button.

2D Rotate Mode

Rotating a drawing in 2 dimensions only changes the display. It does not in any way change the actual part's origin or datums.

The **Edit | Graphic Modes | 2D Rotate** option rotates the part in two dimensions. The part can be rotated a full 360 degrees or any fraction thereof.

The following sections describe the different ways to rotate a drawing in two dimensions.

Rotate 2D by Dragging

PC-DMIS Planner redraws the image dynamically as the mouse is being moved.

A drawing can be rotated in two dimensions by dragging the drawing around the *imaginary center* of the Graphic Display window.

To rotate by dragging:

1. Click the **2D Rotate Mode** icon from the **Graphic Modes** toolbar or select the **Edit | Graphic Modes | 2D Rotate** menu item. This places PC-DMIS Planner in 2D Rotate mode.
2. Position the mouse pointer in the Graphic Display window (in any area other than at the center).
3. Hold down the right mouse button.
4. Move the pointer about the imaginary center of the display window. PC-DMIS Planner redraws the image dynamically as you move the mouse.
5. Release the mouse button. PC-DMIS Planner keeps your current rotation.

Hint: To quickly rotate your part from within any mode, press Alt + right-click and drag the mouse. Note that this does not work if you have the **Rotate** dialog box for 3D rotation open.

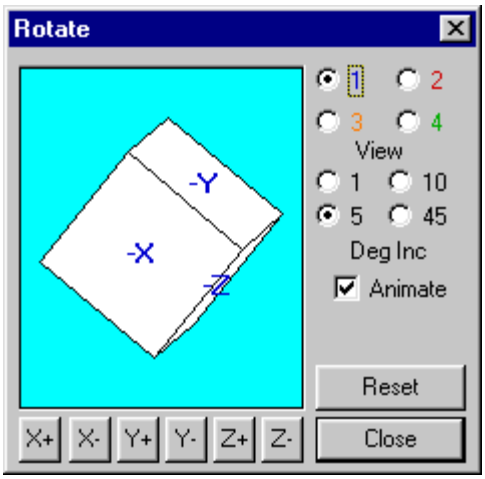
Rotate 2D to an Element (Square up the Drawing)

To "square-up" the part to the screen, from the **Graphic Modes** toolbar, select the **2D Rotate Mode** icon , and then *click* on a feature with the right mouse button. (Do not hold down the button.) PC-DMIS Planner will rotate the drawing so that the selected feature is parallel to the closest screen axis (vertical or horizontal).

Hint: You can square-up the drawing to the closest X, Y, or Z axis at any time, without having the **Rotate** dialog box open, by clicking twice on the Graphic Display window with your middle mouse button.

3D Rotate Mode

The **Edit | Graphic Modes | 3D Rotate** menu option allows you to rotate a part drawing in three dimensions. The part can be rotated up to 45 degrees per rotation. Selecting this option opens the **Rotate** dialog box.



Rotate dialog box

The "Rotate 3D using Rotate Dialog Box", "Rotate 3D by Dragging", and "Rotate 3D to an Element (Square up the Drawing)" topics describe three different ways to rotate a drawing in three dimensions.

Rotate 3D by Dragging

Part rotation can also be accomplished by dragging the mouse. To do this:

1. Access the **3D Rotate** dialog box by selecting the **3D Mode** icon from the **Graphic Modes** toolbar.
2. Click and hold the right mouse button.
3. Drag the mouse.

Hint: To quickly rotate your part with or without the **Rotate** dialog box open, press Ctrl + right-click and drag the mouse. If your mouse has a center mousewheel button, hold the mousewheel button and drag the mouse. (Note, however, that if you have a Vista operating system, clicking the mousewheel button instead activates the Switch Between Windows scrolling functionality.) If your mouse does not have a center mousewheel button, hold the center button on the mouse and drag the mouse.

Rotate 3D using Rotate Dialog Box

To rotate the drawing using the dialog box:

1. Access the **Rotate** dialog box by selecting the **3D Rotate Mode** icon from the **Graphic Modes** toolbar.
2. Select the view to be altered (numbered as 1, 2, 3, and 4 option buttons).
3. Determine the amount of rotation by selecting the **Degree Increment** (1, 5, 10, or 45) option.
4. Click on the desired axis buttons to rotate the drawing.
5. Select the **Close** button. PC-DMIS Planner displays the changes to the CAD image in the Graphic Display window.

Clicking the **Reset** button will snap the rotation of the drawing in the **Rotate** dialog box and in the Graphic Display window to the closest axis.

If you select the **Animate** check box, PC-DMIS Planner dynamically re-draws the part in the Graphic Display window as it is rotated inside the dialog box. If you clear this check box, and then use a rotation button in the dialog box, the rotation of the actual part on the screen does not take place until you release the clicked button.

Rotate 3D to an Element (Square up the Drawing)

To "square-up" the part to the screen, from the **Graphic Modes** toolbar, select the **3D Rotate Mode** icon , and then *click* on a feature with the right mouse button. (Do not hold down the button.) PC-DMIS will rotate the drawing so the selected feature will be parallel to the closest screen axis (vertical, horizontal, or perpendicular to the screen).

Hint: You can square-up the drawing to the closest X, Y, or Z axis at any time, without having the **Rotate** dialog box open, by clicking twice on the Graphic Display window with your middle mouse button.

Text Box Mode

The **Edit | Graphic Modes | Text Box** menu option places PC-DMIS Planner into a mode that lets you manipulate feature IDs and CAD GD&T callouts.

The following are the shortcut menus available inside Text Box Mode:

- If you right-click on the feature, feature's ID label, or text box, PC-DMIS Planner displays a shortcut menu for that feature. See "Feature Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" chapter.
- If you right-click on a CAD GD&T callout, PC-DMIS Planner displays a shortcut menu for that callout. See "Feature Shortcut Menu" in the "Using Shortcut Keys and Shortcut Menus" chapter.

Note: When your mouse isn't over a feature or text box, PC-DMIS Planner allows you to perform the normal translate mode operations of zooming and rotating the part with the right mouse button. See "Translate Mode" for additional information.

Switching Between Curve and Surface Modes

Curve Mode

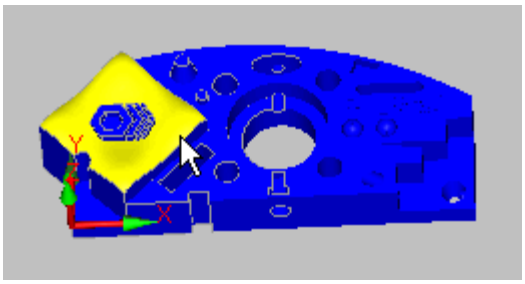
The **Edit | Graphic Modes | Curve** option makes a wire frame's model of curves and lines selectable when clicking on the CAD data in the Graphic Display window. You must import a wire frame model for this option to become available.

Surface Mode

The **Edit | Graphic Modes | Surface mode** option makes a solid model's surfaces selectable when clicking on the CAD data in the Graphic Display window. You must import a solid model for this option to become available.

Highlighting Surfaces and Curves with the Mouse

PC-DMIS Planner gives you the ability to quickly and easily highlight CAD surfaces and wireframe elements by simply hovering over them with your mouse. This lets you get a quick overview of the available surfaces or curves on your part model. This is called Mouse Over Highlighting (MOHL).



Example of Mouse Over Highlighting

To use the MOHL functionality, hold down the SHIFT key and move the mouse over your part.

If you have selected **Surface** mode from the **Graphic Modes** toolbar, PC-DMIS Planner highlights in the highlight color the current surface under the mouse.

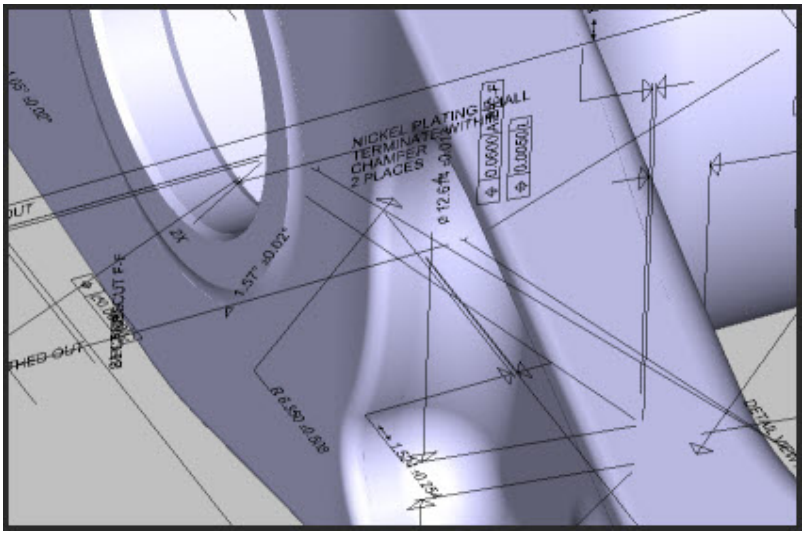
If you have selected **Curve** mode from the **Graphic Modes** toolbar, PC-DMIS Planner highlights the current curve or wireframe element under the mouse.

If using MOHL with a textured bitmap applied to surfaces of your part, you should set your **Z-Buffer Depth** to a value lower than 32 bits. A setting of 32 or higher may cause jagged highlight lines to remain on your image until your refresh the Graphic Display window. See "OpenGL Tab".

Working with CATIA Captures

The CATIA v5 and later versions of the CATIA software provide the ability to add "captures" directly into the CATIA part model. Captures store visibility, view orientation, translation, and scaling information of CAD elements. Generally, captures are also used to make many GD&T elements readable on the model.

Planner 2010 MR3 and later provides you with tools to work with these captures inside the Graphic Display window. The captures are supported through the CATIA v5 DCT and DCI.

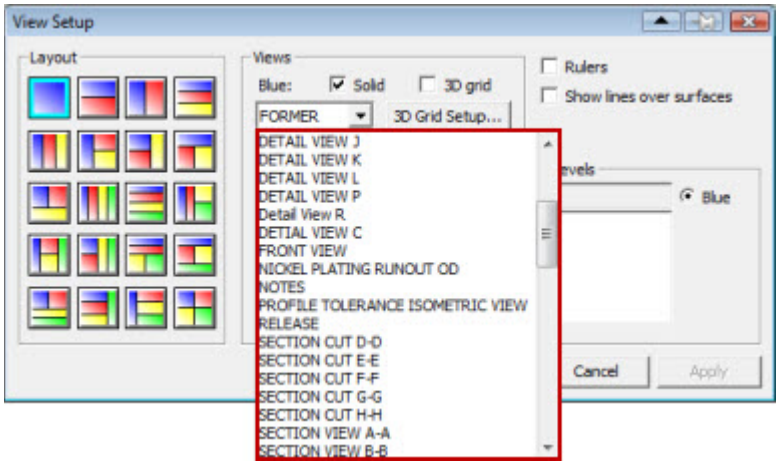


Portion of a CATIA Model Showing Captures (the GD&T or other text-like CAD elements)

PC-DMIS Planner adds the CATIA Captures into the **View Setup** dialog box providing you with the ability to orient the view to desired captures.

Using the View Setup dialog box to Orient the Screen View to Captures

The drop-down lists in the **View Setup** dialog box hold the view orientations associated with the captures.



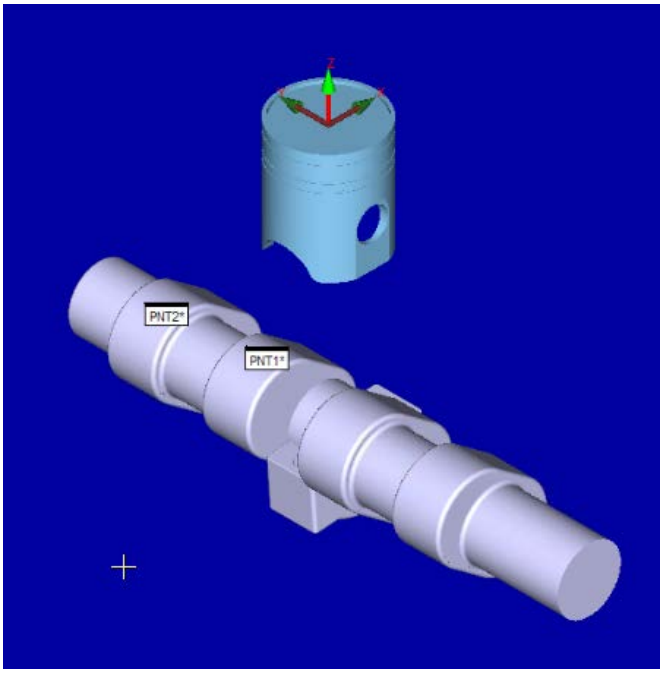
The View Setup Dialog Box Showing Captures in a drop-down list

You can orient the screen to a CATIA capture by simply selecting it from the drop down list on the desired colored view or views. Click **Apply** to preview the orientation change. When you click **OK**, the orientation for that view will change to that capture.

For information on the other **View Setup** dialog box operations, see the "Setting Up the View" topic.

Working with Assemblies of Parts

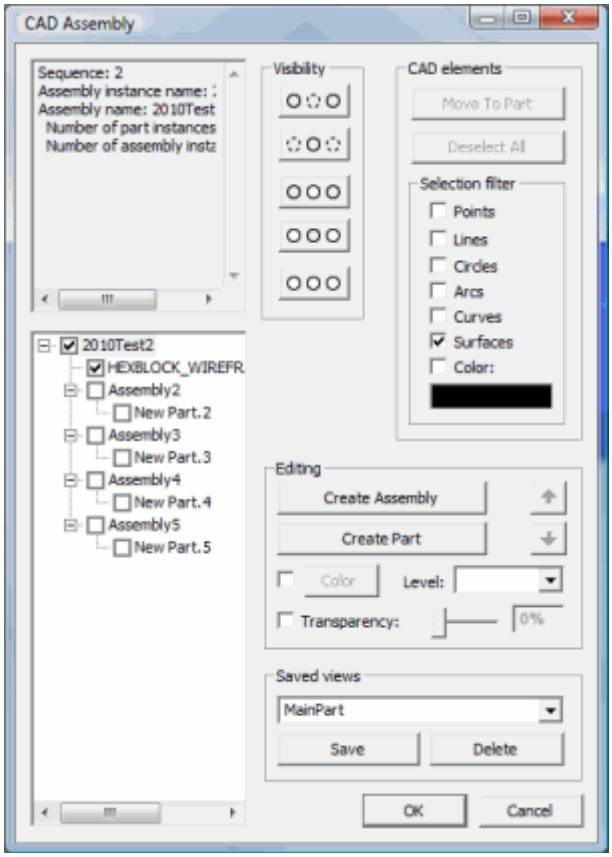
A CAD Assembly consists of two or more imported part models merged together in the Graphic Display window. For example, the graphic below shows an assembly of two visible parts.



A Sample CAD Assembly

You can have multiple parts and even multiple assemblies together in a single .cad file.

To work with assemblies, select the **Assembly** icon from the **Graphic Modes** toolbar or select **Edit | CAD | CAD Assembly** to display the **CAD Assembly** dialog box.



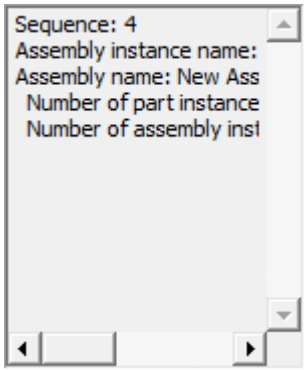
CAD Assembly dialog box

This dialog box contains a list of all imported and merged parts or assemblies used in the Graphic Display window. You can use the **CAD Assembly** dialog box to view, edit, and manage assemblies of parts.

The **Rename**, **Delete**, **Transform**, **Coordinate System**, and **Expand Descendent** buttons from PC-DMIS Planner 2009 and earlier have been moved to the right-click shortcut menu in the **Assembly Tree View**.

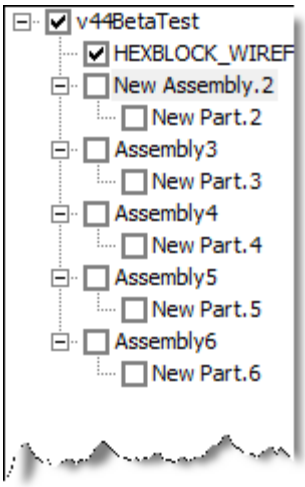
This dialog box contains the following options:

Information box



The gray box at the top of the dialog box displays information about the item currently selected on the **Assembly Tree View**. The information changes depending on the item selected.

Assembly Tree View area



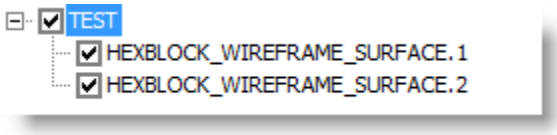
The left side of the dialog box has a tree view representing the CAD assembly structure. Each item in the tree view represents an assembly component.

- If you select a component from the tree view, Planner displays information about that item in the gray box above the list.
- If you double-click on a component in the assembly, Planner selects and highlights it in the Graphic Display window. Conversely, with the dialog box open, if you press Ctrl and select a CAD object in the Graphic Display window, Planner selects its corresponding component in the tree view. This makes it easy to identify which CAD objects belong to which assembly component.

Each component has its own check box to show or hide that component immediately within the Graphic Display window.

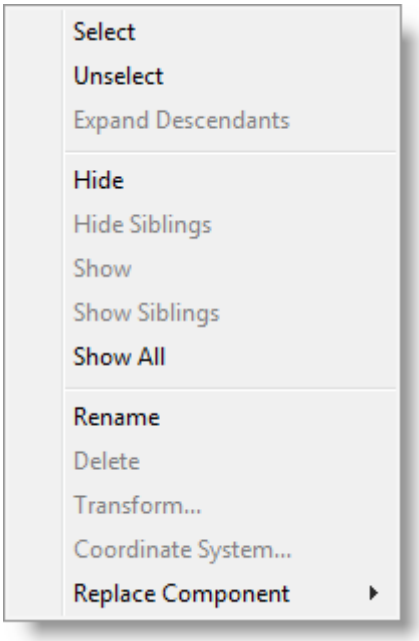
- If you mark a check box, Planner shows that component.
- If you clear a check box, Planner hides that component (and any children components, regardless of their visibility state).
- Once you mark a check box, you can also change the visibility state of that component or of its sibling or children components by using the buttons in the **Visibility** area.

Your original imported image name has the number 1 appended to its name. Any CAD transformation that creates a new instance of this same CAD file will then increment to have a unique number. See "Transforming a CAD Model".



Mirrored CAD File names with unique numbers inside a CAD Assembly Tree View

A shortcut menu appears when you right-click in the tree view.



It contains these items:

- **Select** - This functions as if you had double-clicked on the assembly component.
- **Unselect** - This deselects the assembly component in the Graphic Display window.
- **Expand Descendants** - This expands an assembly to show a list of child parts making up the assembly.
- **Hide** - This hides the selected component.
- **Hide Siblings** - This hides all of the components except the selected component at the same level on the assembly tree.
- **Show** - This shows the selected component.
- **Show Siblings** - This shows all of the components except the selected component at the same level on the assembly tree.
- **Show All** - This shows all the components on the assembly tree.
- **Rename -** This renames the selected component in the assembly.
- **Delete** - This deletes the selected component from the assembly and removes it from the Graphic Display window. You cannot delete the root tree view component.
- **Transform** - This displays the **CAD Transform** dialog box. This dialog box allows you to transform (translate, scale, and rotate) a single component in the assembly. You cannot transform the root tree view component. See "Transforming a CAD Model" for more information.
- **Coordinate System** - This displays the CAD Coordinate System dialog box. This dialog box lets you create and manage different coordinate systems. See "Working with CAD Coordinate Systems" for more information.

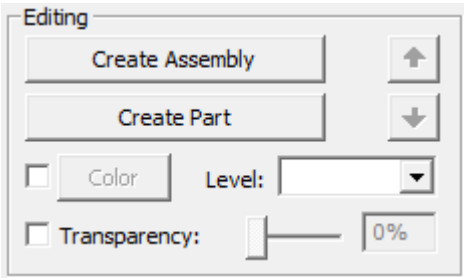
- **Replace Component** - This allows the selected assembly component to be replaced with another CAD model. This can either be an imported CAD model or a model connected with the DCI. The sub-menus have the same structure as the **File | Import** and **File | Direct CAD Interfaces** menu options respectively. The new model will replace whatever model was associated with the selected node. The new model on the node retains whatever transformation was applied to the previous model. For information on importing model files, see the "Importing and Exporting Files" section.

Visibility area

The **Visibility** area provides convenient buttons to change the hidden state of assembly component groups. Select the Hide button to hide the selected component.

- **Hide** - This hides the selected component.
- **Hide Siblings** - This hides all of the components except the selected component at the same level on the assembly tree.
- **Show** - This shows the selected component.
- **Show Siblings** - This shows all of the components except the selected component at the same level on the assembly tree.
- **Show All** - This shows all the components on the assembly tree.

Editing area



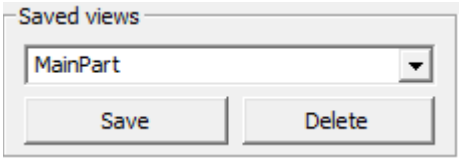
The **Editing** area lets you edit the currently selected component from the Assembly Tree View.

- **Create Assembly** - This creates a new assembly node in the currently selected assembly. Initially, it is empty but you can then move other assembly components into it by using the **Up Arrow** and **Down Arrow** buttons on those components.

- **Create Part** - This creates a new part node in the currently selected assembly. Initially, it is empty, but you can move CAD objects into it by using the **Move to Part** button in the CAD Elements area.
- **Up Arrow** - This moves the part or assembly component in the tree view up into the assembly just above it in the list.
- **Down Arrow** - This moves the part or assembly component in the tree view down into the assembly just below it in the list.
- **Rename** - This renames the selected component in the assembly.
- **Delete** - This deletes the selected component from the assembly and removes it from the Graphic Display window. You cannot delete the root tree view component.
- **Transform** - This displays the CAD Transform dialog box. This dialog box allows you to transform (translate, scale, and rotate) a single component in the assembly. You cannot transform the root tree view component. See "Transforming a CAD Model" for more information.
- **Coordinate System** - This displays the CAD Coordinate System dialog box. This dialog box lets you create and manage different coordinate systems. See "Working with CAD Coordinate Systems" for more information.
- **Color** - This check box enables the Color button and displays the Color dialog box. This dialog box lets you apply a color to the selected tree view component. This check box also enables or disables the display of the selected color. Once you choose the initial color, you can click the Color button thereafter to modify the color used.
- **Level** - This list lets you assign the CAD objects within the selected assembly component to a predefined CAD level. This overrides any CAD level assignments that were set using the CAD Levels dialog box until the assembly component is removed from the level or from the assembly within the CAD Assembly dialog box. See "Working with CAD Levels" for more information.
- **Transparency** - This check box applies the selected transparency percentage to the selected tree view component. You can drag the slider or change the value manually in the box to change the transparency percentage.

Note: You cannot perform some edit operations on the root-level component (usually the part name used in your part program) or DCI components. Those operations you cannot use are disabled when you select these types of components.

Saved Views area

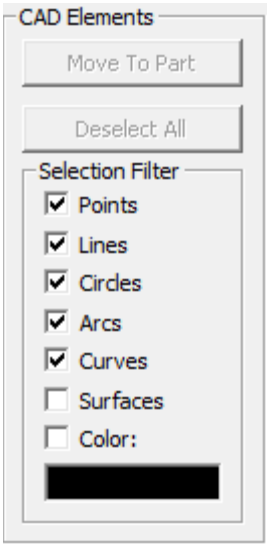


The **Saved Views** area lets you manage different visibility states of the **Assembly Tree View**.

- **Save** - This saves the current visibility state, giving it the name specified in the current **Saved Views** box. You must click the **OK** button to permanently save your view. Planner stores saved views in the .CAD file associated with the part program.
- **Delete** - This deletes the stored visibility state of the name selected from the **Saved Views** box.

To load a stored visibility state, select it from the **Saved Views** list. The check boxes in the **Assembly Tree View** are immediately modified according to the loaded view.

CAD Elements area



The **CAD Elements** area lets you select CAD objects and move them to a different part. You can select or deselect items by simply clicking on them within the Graphic Display window. You can also box-select multiple CAD objects.

- **Move To Part** - This moves the selected CAD objects to the currently selected part in the **Assembly Tree View**.
- **Deselect All** - This deselects all previously selected **CAD** objects.
- **Selection Filter** - This area lets you filter the CAD objects that Planner can select. If an item is selected, Planner will have the ability to select that type of CAD object when you click or box-select CAD objects in the Graphic Display window.

You can also filter the selection by the color of the CAD object. If you select the **Color** check box, the color of the next CAD object that you select in the Graphic Display window is then used as the filter color. For subsequent selections, Planner will only select CAD objects with that color.

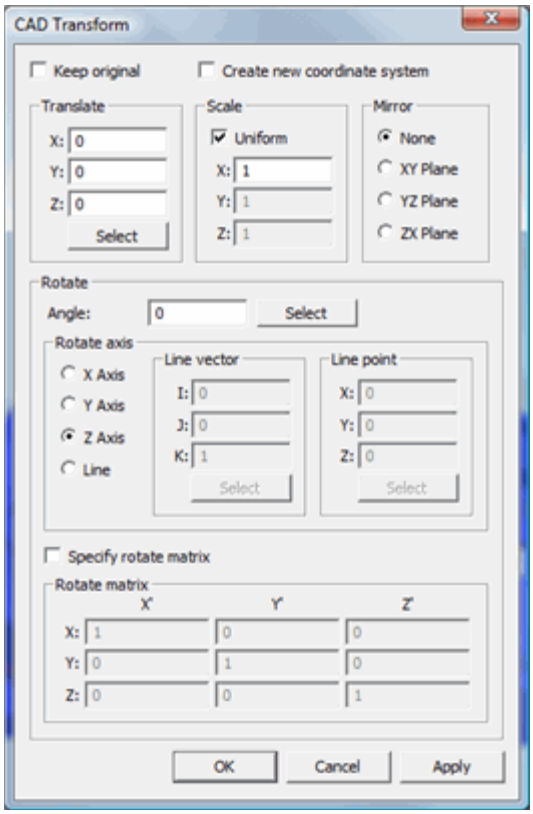
Item	Description
Box	
	•
Visibility area	
Editing area	

area	
area	

Transforming a CAD Model

The CAD Transform dialog box allows you to transform (translate, scale, and rotate) your CAD model, and if desired, keep a copy of the original unmodified CAD model, as well as create a new coordinate system for the transformed model.

To transform your CAD model, access the **CAD Transform** dialog box by clicking the Transform button within the **CAD Assembly** dialog box. Once the **CAD Transform** dialog box appears, use the items on the dialog box as needed, and then click **OK** or **Apply**.



CAD Transform dialog box

The following items are available:

Keep original check box

This check box lets you keep a copy of the original, unmodified CAD model.

Create new coordinate system check box

This check box lets you create a new coordinate system from the newly translated CAD model. See "Working with CAD Coordinate Systems" for more information.

Translate area

This area defines the XYZ offsets to translate the model.

You can either type the specific location to which you want Planner to move the coordinate system, or if you don't know the coordinates, you can use the **Select** button to select a specific CAD entity to which the coordinate system will move. See "Transforming by Selecting" below.

Scale area

This area defines how the CAD model will be scaled. This can be useful for fixing models that are not scaled correctly due to improper measurement unit identification. For example, if the model is sized for millimeter units but you know they should be inch units, you would scale the model by 25.4.

The **Uniform** check box uniformly scales the model. If you want to scale a selected axis of the model, clear the **Uniform** check box. Leave the value as 1 for axes that you don't want scaled, and change the axis that you want scaled. You can also scale axes by negative values. This is useful if you want to mirror an axis. In this case you would enter -1 for that axis.

Mirror section

In this section you can mirror the CAD view of a part. Mirroring provides the same functionality that could be done using **Scale** with -1 specified for the mirroring axis.

Mirroring your CAD model is particularly useful when measuring automotive parts that have identical left and right-hand pieces. If CAD information is available for the right side of a part, you can mirror the appropriate axis and create a CAD view of the left side of the part.

Mirroring the CAD Data

Select the axis you want to mirror. If you want to keep your old CAD data so that after mirroring you'll have CAD data for both the symmetrical halves, click the **Keep Original** check box.

Once you click **Apply** or **OK** Planner will mirror the CAD drawing in the specified axis and display the image in the Graphic Display window.

Mirrored Elements in CAD Assemblies

If you access the **CAD Assembly** dialog box, you will see that your original imported image name has the number 1 appended to its name. Any CAD transformation that creates a new instance of the same CAD file (such as a mirror operation) will then increment to have a unique number. See "Working with Assemblies of Parts".

Mirrored CAD File names with unique numbers inside a CAD Assembly Tree View

Hidden Assembly Components During a Mirror Operation

If you hide elements of your CAD Assembly prior to mirroring your CAD data, hidden components are still mirrored but their mirrored counterparts also continue to remain hidden in the Graphic Display window until their visibility state is modified in the **Assembly Tree View** of the **CAD Assembly** dialog box.

Rotate area

This area controls how the CAD model will be rotated. Type the angle you want to rotate the model by in the **Angle** box.

You can either type the specific location to which you want Planner to move the coordinate system, or if you don't know the coordinates, you can use the **Select** button to select a specific CAD entity to which the coordinate system will move. See "Transforming by Selecting" below.

Rotate axis area

This area defines the line about which the CAD model will be rotated. The model will rotate about this line by the specified angle. The direction of rotation follows the *right-hand rule*.

> **Right Hand Rule**: If you extend the thumb on your right hand in the direction of the line vector, and curl your other fingers into your palm, your fingers indicate the direction of positive angle rotation.
> You can use one of the coordinate system axes as the line to rotate about by selecting the appropriate **X**, **Y**, or **Z** axis option button.
> If you don't want to rotate about one of the coordinate system axes, you can rotate about an arbitrary line by selecting the **Line** option button. This enables the **Line Vector** and **Line Point** areas. Fill these areas out to determine the point and vector that comprise the arbitrary line.
> **Rotate matrix** area
> As you determine your CAD model's new transformation, this area automatically gets filled out with the values to use in a 3x3 matrix. This 3x3 matrix rotates the CAD model.

Note: Usually, you won't need to fill out anything in this area as it is generally for informational purposes only.

For Advanced Users, click here

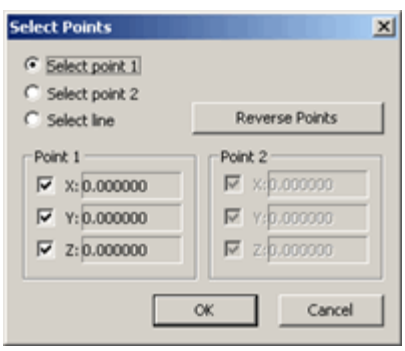
You can select the **Specify Rotate Matrix** check box to type the values for the rotation matrix manually. The columns specify the axes for the rotation. These restrictions apply:

- Each axis of the matrix must be orthogonal to the other two axes. Thus, each axis pair must form a 90-degree angle.
- Each axis must be unit length. That is, the length of the axis must be one.

When you apply the transformation, if either of these restrictions is not met, a message appears indicating the problem, and PC-DMIS automatically corrects the rotation matrix.

Transforming by Selecting

When you click the **Select** button Planner displays the **Select Points** dialog box.



Select Points dialog box

Instead of typing an offset value, you can use this dialog box to select an offset by simply picking a CAD entity from the Graphic Display window.

The following dialog box options are available:

Select point 1 option

This option defines the translation location. With this option selected, click on a desired CAD entity. This anchors the point to that location.

Select point 2 option

This option specifies the angle with respect to point 1 and the axis of rotation. With this option selected, click on a second CAD entity on your CAD model to define the angle.

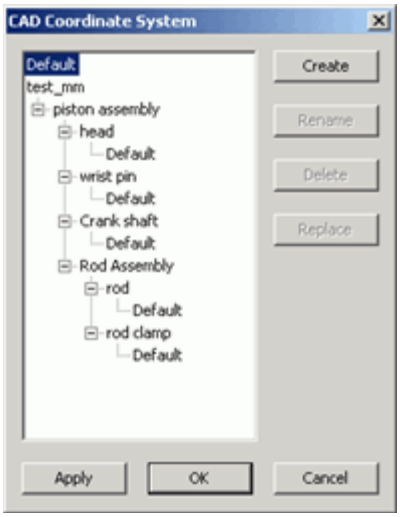
Select line option
Instead of selecting two points for your translation, this option lets you select a single line. Planner then sets the values of the **Point 1** and **Point 2** areas to match the start and end point of the selected line.

Reverse Points button
This switches the XYZ values of Point 1 with the XYZ values of Point 2.

Point 1 and Point 2 areas
These areas define the XYZ center point of the CAD entity selected with the **Select point 1** and **Select point 2** options.

The check boxes in these areas allow you to selectively update the X, Y, or Z value of the point, enabling you to specify points where there is no actual geometry for you to click on. For example, for Point 1, suppose you wanted the X and Y value of one point but the Z value of a different point. To do this, you would clear the **Z** check box and then select one point. Then you would clear the **X**, **Y** check boxes, select the **Z** check box, and then select the other point.

Working with CAD Coordinate Systems



CAD Coordinate System dialog box

The **Edit | CAD | CAD Coordinate System** menu option displays the **CAD Coordinate System** dialog box. The **CAD Coordinate System** dialog box lets you create or select new coordinate systems for your CAD model.

The **Coordinate System** list on the left side of the dialog box shows all the coordinate systems in the CAD model. The coordinate systems listed first are the global coordinate systems. You will always see the **Default** coordinate system listed.

If the CAD model is an assembly of component parts, then following the global coordinate systems will be listed all the coordinate systems contained in that assembly. These are listed in a tree view structure. To expand the tree view, click on a plus symbol (+). PC-DMIS displays the component parts' assembly coordinate systems. You will always see a **Default** coordinate system listed for each part component in the assembly. This coordinate system defines the part's default local coordinate system.

When you first open the **CAD Coordinate System** dialog box, PC-DMIS checks to see if the current active coordinate system matches a coordinate system in the list:

- If it finds a match, then PC-DMIS selects it from the list.

- If it doesn't match any of the coordinate systems, then PC-DMIS uses the **Default** coordinate system at the top of the list. This can happen if you transformed the CAD without creating a coordinate system (see CAD Transform dialog section for more information).

To Select and use Coordinate Systems

To use an existing coordinate system,

1. Select the coordinate system from the **Coordinate System** List. PC-DMIS shows that coordinate system in the Graphic Display window. If you select an assembly component, PC-DMIS selects the first coordinate system in that component instead.
2. Click **Apply** or **OK**. The selected coordinate system becomes the new active coordinate system and PC-DMIS redraws the CAD to reflect its new position.

To Create a Coordinate System

To create a coordinate system at the current CAD model's position, click the **Create** button. The CAD model's position can be changed by using the **CAD Transform** dialog box. See the "Transforming a CAD Model" topic for more information on transforming the CAD model.

To Rename a Coordinate System

To rename a coordinate system, select the coordinate system from the list, and click the **Rename** button. Then type the new name.

You cannot rename the **Default** coordinate systems.

To Delete a Coordinate System

To delete a coordinate system, select the coordinate system from the list, and click the **Delete** button. PC-DMIS deletes the selected coordinate system.

You cannot delete the **Default** coordinate system or assembly coordinate systems.

To Replace a Coordinate System

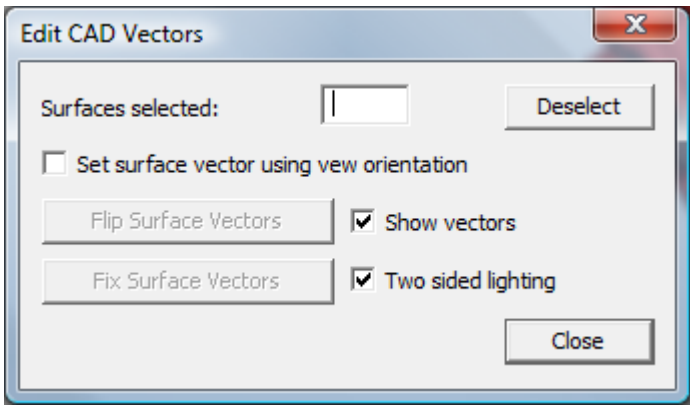
To replace or overwrite an existing Coordinate System with the current CAD model's position, select a coordinate system from the list, and click the **Replace** button.

You cannot replace the **Default** coordinate system or assembly coordinate systems.

Editing CAD Vectors

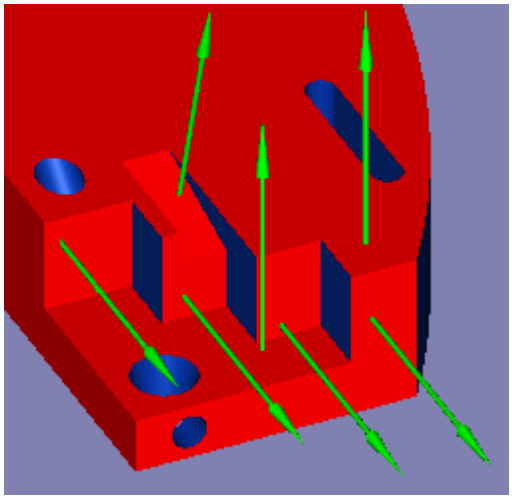
To edit the vectors for surfaces of your CAD:

1. Click **CAD Vectors** from the **Tools** toolbar or select the **Edit | CAD | CAD Vectors** menu item. The **Edit CAD Vectors** dialog box appears.



Edit CAD Vectors dialog box

2. Click on the surface(s) of the CAD where the surface vector needs to be changed. Notice that the current surface vector is shown with a green arrows at the location where you clicked if the **Show Vectors** options has been selected. Also, the number of **Selected Surfaces** is displayed. To start over with selecting surfaces, click **Deselect**.



Example of Surface Vectors

3. Click **Flip Surface Vectors** to flip the currently selected vectors so they point in the other direction.
4. Click **Fix Surface Vectors** to correct any surface vector that points into the part to point outwards.
5. Click **Close** when you are finished.

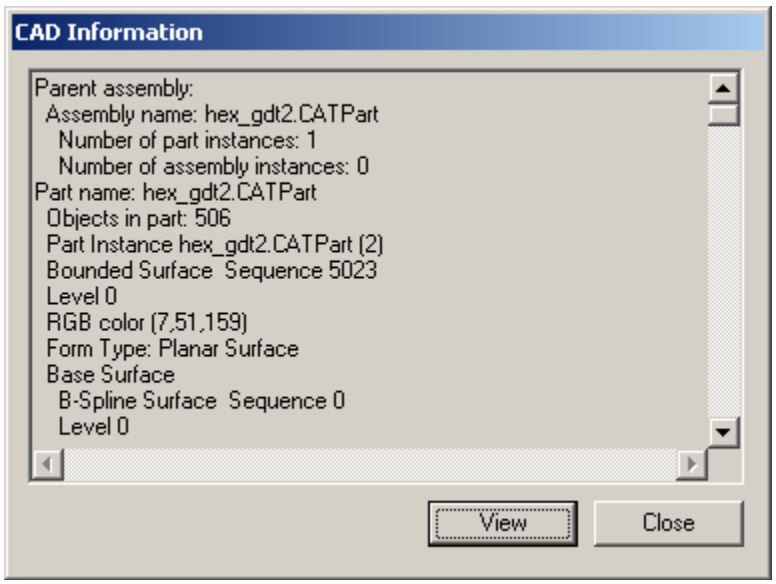
Viewing CAD Nominal Information

The **CAD Information** dialog box provides detailed information from the imported CAD file. Things such as assembly information (if applicable), number of parts in the assembly, number of objects in each of the parts that make up the assembly, surface type and sequence, current level, RGB color, form type, and detailed nominal information for the surface.

The nominal information includes information such as diameter, vector or location. For complex elements such as trimmed surfaces, information about other geometries (base surface, bounding curves) will also be displayed.

To view the specific information about CAD geometry:

1. Click **CAD info** from the **Tools** toolbar or select the **Edit | CAD | CAD Information** menu item. The **CAD Information** dialog box appears.



Cad Information dialog box

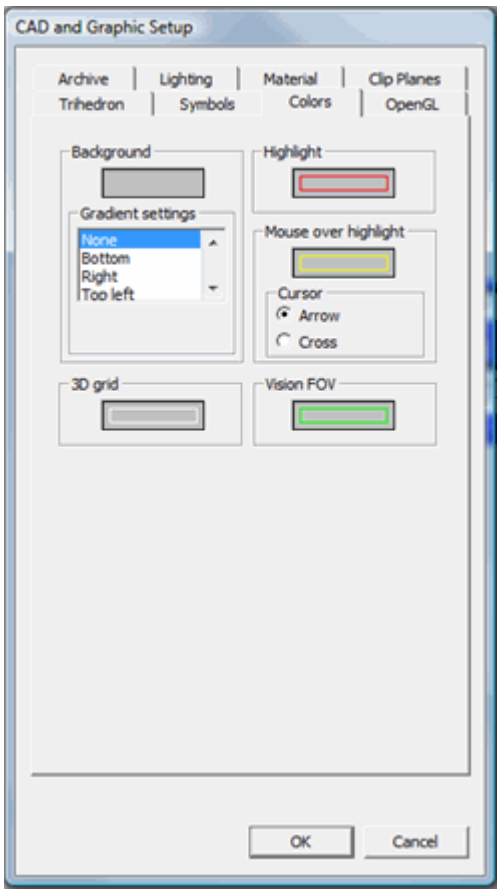
2. Select any CAD geometry element and the element information is displayed in the **CAD Information** dialog box.
3. Click **View** and the CAD element will flash on the screen.
4. Click **Close** when you are finished.

Editing Screen Colors

The **Screen Color** dialog box allows you to determine the colors to display within the Graphic Display window. These include the background color, a gradient color, highlight and mouse over highlight colors.

To edit the screen colors:

1. Click CAD Text Colors from the Tools toolbar or select the Colors tab from CAD and Graphics Setup dialog box (Edit | Setup | CAD and Graphic Setup).



Colors tab - CAD and Graphic Setup dialog box

2. Edit the following color values by clicking one of the color boxes and selecting the desired color from the **Color** dialog box. Click **OK** to return to the **Colors** tab.

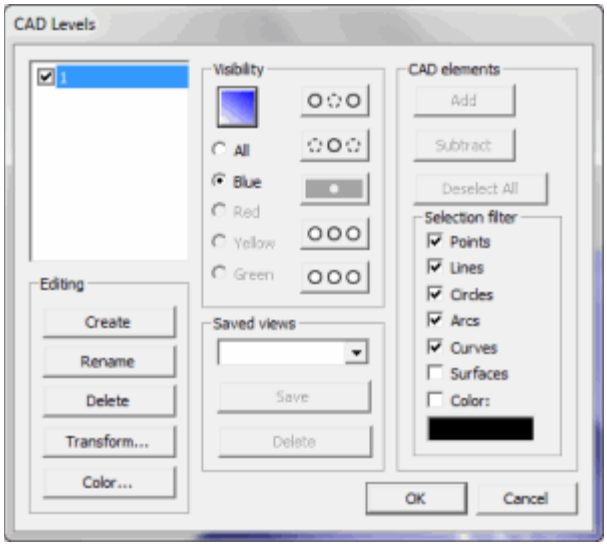
   - **Background** - Provides the background color for the screen or the primary background color when a gradient is specified.
   - **Gradient Settings** - When you set the gradient to something other than "None" in the list an additional color box appears. This box allows you to define a secondary background gradient color. PC-DMIS Planner will start from the setting location (for example the "Bottom") and display the secondary gradient color, and then gradually change from that color to the primary gradient color on other end of the screen.
   - **Highlight** - Allows you to change the color used for highlighting selected CAD elements within the Graphic Display window
   - **Mouse Over Highlight** - This color is used when you highlight surfaces or curves with your mouse using Mouse Over Highlighting (MOHL). The **Cursor** area lets you display either an Arrow or a Cross shape for the mouse pointer when in MOHL mode.
3. When finished, click the **OK** button, or click **Cancel** to undo any changes.

Working with CAD Levels

A CAD level consists of a group of user-selected CAD geometries or elements selected from the Graphic Display window. The level information is saved with the .CAD file so that if you import the .CAD into another part program the level information will be available there too.

You can define which levels will be displayed in the Graphic Display window by using the **CAD Levels** dialog box (**Edit | CAD | CAD Levels...**). Once levels are defined, you can view the levels using either the **CAD Levels** dialog box or the **View Setup** dialog box (**Edit | Setup | View Setup...**)

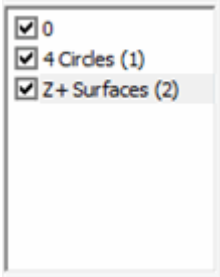
Clicking the **CAD Levels** icon from the **Graphics Modes** toolbar or selecting the **Edit | CAD | CAD Levels...** menu item displays the **CAD Levels** dialog box. This dialog box lets you define the CAD elements that will be assigned to the levels. You can also use it to modify levels, delete levels, and view levels in the Graphic Display window:



CAD Levels dialog box

The dialog box contains these items:

Levels list

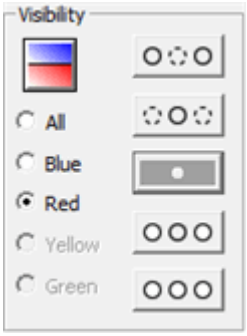


The list of levels in the upper-left area of the dialog box shows all the current levels in the CAD model. The list will always contain the default level 0, a level that automatically contains all the CAD elements in the CAD file.

Here are some operations you can perform using the levels list:

- Double-click on a level from the list to select all CAD elements assigned to that level.
- Select a CAD element in the Graphic Display window and press and hold Ctrl on the keyboard to select the corresponding level in the level list.
- Select the check box next to a level to show the selected level.
- Clear the check box to hide the selected level.

Visibility area



The **Visibility** area provides option buttons and icon buttons that allow you to run visibility operations in order to change the hidden state of levels.

The colored icon at the top of this area shows your current screen layout. It corresponds with the icon used in the **Layout** area of the **View Setup** dialog box. See "Setting Up the Screen View".

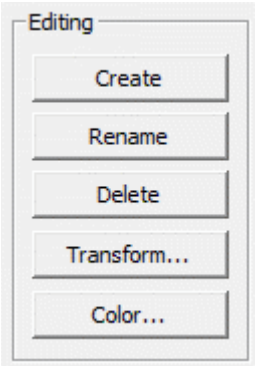
The **Blue**, **Red**, **Yellow**, and **Green** options limit the visibility operation to only that "view" of the Graphic Display window. The **All** option applies the visibility operation to all views.

These command buttons have pictures on them that depict their action:

- **Hide** - Hides the level selected in the list.
- **Hide Siblings** - Hides all the levels except the level selected in the list.
- **Show** - Shows the level selected in the list.
- **Show Siblings** - Shows all the levels except the level selected in the list.
- **Show All** - Shows all the levels in the list.

Note: These buttons will have a dark gray background and will be unavailable for selection until you select a level from the list of levels.

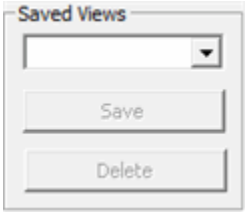
Editing area



The **Editing** area allows you to modify CAD levels. For all buttons except **Create**, you will need to first select a level from the levels list and the buttons will affect that selected level.

- **Create** - Creates a new level in the CAD model. Initially, there won't be any CAD elements assigned with this new level. You will need to use the **Add** button in the **CAD Elements** section to add CAD elements selected from the Graphic Display window to the new level.
- **Rename** - Renames the selected level. You can also rename a level by clicking on a selected level in the list until the cursor appears.
- **Delete** - Deletes the selected level. Planner will assign any CAD elements that were on that level to the default level 0. You cannot delete this default level.
- **Transform** - Transforms the selected level. When you click this button, Planner displays the **CAD Transform** dialog box. See "Transforming a CAD Model" for information on using this dialog box.
- **Color** - Changes the color of all CAD elements in the selected level.

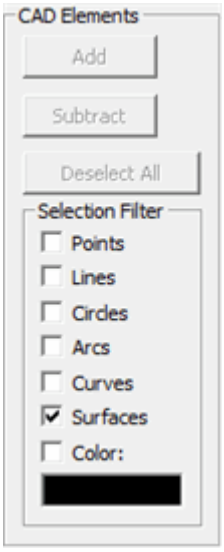
Saved View area



The **Saved Views** area stores the level visibility state in the CAD model. These are saved inside the .cad file. Use the drop-down combo box to either select and recall a saved view or to specify a name for a new view.

- **Save** - Saves the current level visibility state using the name entered in the combo box.
- **Delete** - Deletes the saved view selected in the combo box.

CAD Elements area



The **CAD Elements** area modifies the level of the selected CAD elements. You can select (or deselect) CAD elements by simply clicking on or box-selecting them in the Graphic Display window. Selected elements will be highlighted in red.

- **Add** - Adds the selected CAD elements to the currently selected level in the list.
- **Subtract**- Removes the selected CAD elements from the currently selected level in the list. The CAD elements will be assigned the default level 0, but only if they were on the selected level. If a selected CAD element is not on the selected level, then that CAD element will not be modified.
- **Deselect All** - Clears any selected CAD elements on the Graphic Display window.
- **Selection Filter** - The check boxes in the **Selection Filter** area filter the type of CAD elements assigned to a CAD level. You can also use the **Color** check box in this area to filter selection by the color of the CAD element. When you select the Color check-box, the color of the next CAD element that you select in the Graphic Display window will be used as the filter color. For subsequent selections, only CAD elements with that color can be selected.

Creating Levels

1. Access the **CAD Levels** dialog box.
2. Click **Create** from the **Editing** area. A new level number will appear in the list of levels.
3. Select the new level and rename it if desired. Even if you rename the level, Planner will still show the level number in parentheses.
4. Ensure that the CAD level is selected from the list.
5. From the Graphic Display window, select the CAD elements you want included in the CAD level. To limit which CAD element types are selectable, select the check boxes of CAD elements in the **Selection Filter** area.
6. Click the **Add** button. Planner assigns the selected elements to the selected level. Now when you show that level and hide other levels, PC-DMIS will show only those elements.
7. Click **OK** when finished to close the dialog box.

Modifying Levels

1. Access the **CAD Levels** dialog box.
2. Select the level from the level list.
3. To add elements to the level, select the CAD elements from the Graphic Display window and click **Add**. PC-DMIS modifies the level accordingly.
4. To remove elements from the level, select the level and display its assigned CAD elements. From the Graphic Display window, select the elements to remove and click **Subtract**. PC-DMIS modifies the level accordingly.
5. Click **OK** when finished to close the dialog box.

Deleting Levels

1. Access the **CAD Levels** dialog box.
2. Select a level from the level list.

3. Click **Delete**. PC-DMIS deletes the level and assigns all CAD elements of that level to the 0 level.
4. Click **OK** when finished to close the dialog box.

Showing a Level for a Specific Layout Pane

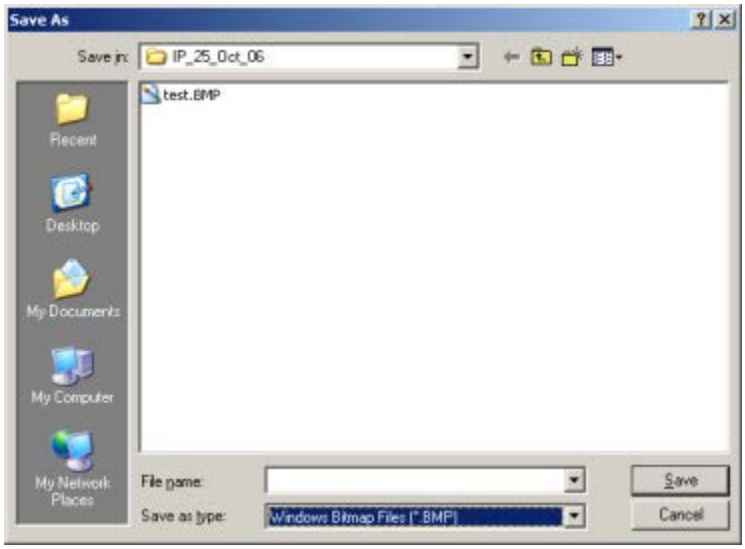
1. Access the **CAD Levels** dialog box.
2. From list of levels, select the level to use and ensure that its check box is selected.
3. From the **Visibility** area, select the **Blue**, **Red**, **Yellow**, or **Green** option button for the pane in which you want to view the level.
4. Either deselect the check boxes from other levels in the list of levels, or use the pictured **Hide Siblings** button in the **Visibility** area to hide other levels.
5. Click **OK** when finished to close the dialog box.

Note: You can also view levels within the **View Setup** dialog box.

Capturing the Graphic Display window

PC-DMIS Planner allows you to capture an image of the current view from the Graphic Display window to the clipboard or a bitmap file.

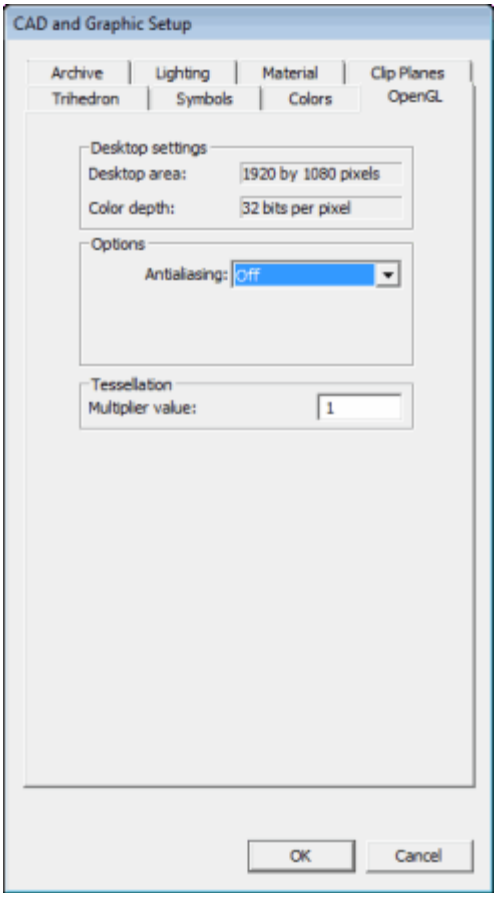
- Click **Capture To Clipboard** from the **Tools** toolbar or select the **Edit | Screen Capture To | Clipboard** menu item to capture the Graphic Display window to the clipboard. This image can then be pasted into any windows application where the paste option is available.
- Click **Capture To File** from the **Tools** toolbar or select the **Edit | Screen Capture To | File** menu item to open the **Save As** dialog box.



Browse to the folder where you want to save the bitmap and type the name in the **File Name** edit box. Click **Save** to save the file as a bitmap (.bmp) file.

Setting OpenGL Options

The **OpenGL** tab allows you to change the options that affect display of the model in "Surface Mode". Click **OpenGL Settings** from the **Tools** toolbar or select the **OpenGL** tab from the **CAD and Graphic Setup** menu item (**Edit | Setup | CAD and Graphic Setup**).



CAD and Graphics Setup dialog box - OpenGL tab

Notes:

The size of the CAD data file and the tessellation multiplier value used affects the amount of memory needed. These both affect the amount of tessellated facets needed to display the model. The smaller the tessellation multiplier value used, the more memory needed for the facets. For large CAD models this could cause an “Out Of Memory” error. If this occurs, the current PC-DMIS session will be left in an unstable state and should be terminated.

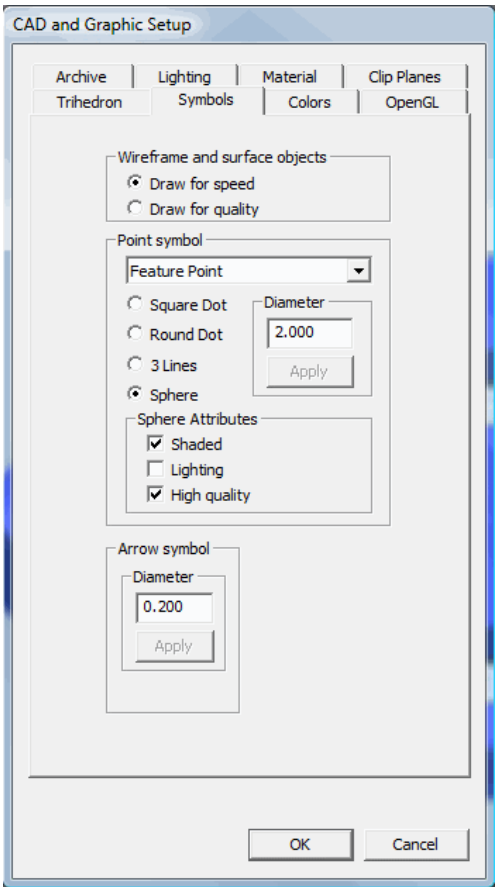
The default tessellation multiplier value is 1.0. Setting a tessellation multiplier of 0.1 will result in a 10 to 20 percent increase in the memory required over the default value of 1.0. Decreasing the tessellation multiplier further to 0.01 will result in an additional 50 to 65 percent increase of memory required.

For information, see "OpenGL Tab".

Modifying Display Symbols

The **Symbols** tab of the **CAD and Graphic Setup** dialog box (**Edit | Setup | Cad and Graphic Setup**) lets you change how different symbols are displayed in the Graphic Display window.

Hint: You can also display this by clicking the **Display Symbols** icon from the **Graphics Modes** toolbar.

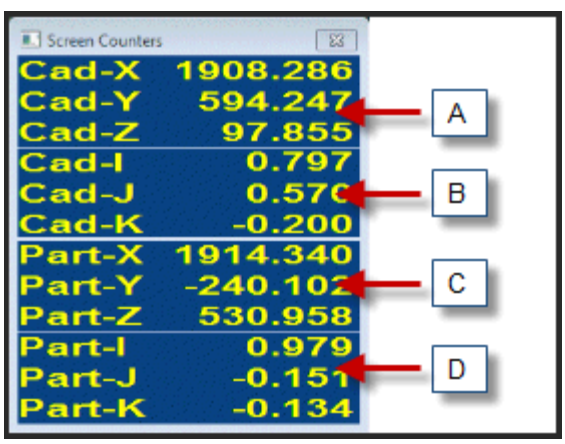


CAD and Graphic Setup dialog box—Symbols tab

See the "Symbols Tab" topic for information.

Displaying Mouse Coordinates in the Screen Counters Window

PC-DMIS Planner can provide you with a continuous update of the location of your mouse pointer's location in the **Screen Counters** window as it moves over the part model. To access this window, select **View | Screen Counters**.



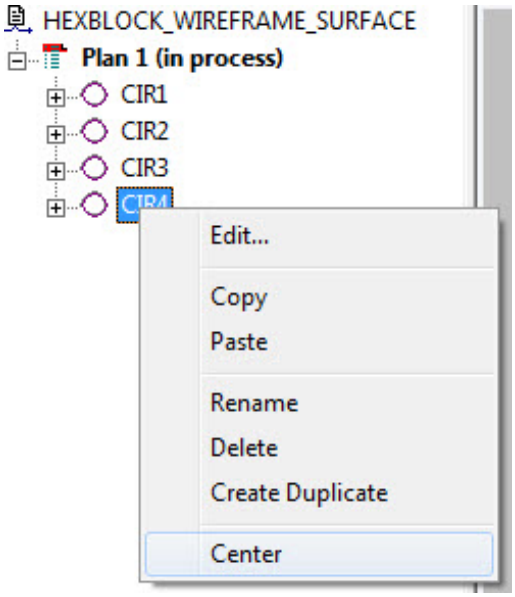
- **A** - Screen Counters showing mouse location using the CAD model's coordinate system.

- **B** - Screen Counters showing the IJK vector of the CAD at the mouse location using the CAD model's coordinate system.
- **C** - Screen Counters showing mouse location using the current part coordinate system.
- **D** - Screen Counters showing the IJK vector of the CAD at the mouse location using the current part coordinate system.

If you move the mouse away from the part over empty space, or if the **Screen Counters** mode is not active, it displays the last known location of the mouse when over the part.

Centering the Graphics Feature View

A feature can be rotated and centered in the **Graphic Display window** by right-clicking on the specific feature in the **Plan Window** and selecting the **Center** option.



Planner will automatically rotate and center the selected feature in the **Graphic Display window**.

Viewing the Graphic Display in Demo Mode

The **View | Demo Mode** menu item places the Graphic Display window in a mode where it continually rotates the part in a random manner, useful for demo purposes.

To exit this demo mode, simply click anywhere within the Graphic Display window.

Working with Planner CAD Files

PC-DMIS Planner uses .cad files to store the imported .cad and any embedded Inspection Plans. This allows you to store multiple Inspection Plans in the PC-DMIS Planner .cad file. When you import a Planner .cad file into PC-DMIS and it has one or more embedded plans, PC-DMIS allows you to select Inspection Plans using the **File | Import | Inspection Plan** menu item.

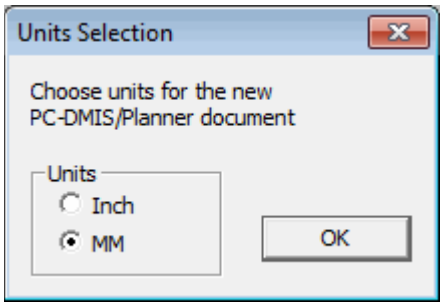
Inspection Plans are included with a Planner .cad file when the plan has been "released". To release a plan, right-click on an "in process" plan and select **Release Plan** from the pop-up menu. You can also release plans using the **Save** button from the **Main** toolbar. "Released" plans will be available for import from the saved Planner .cad file in PC-DMIS.

Any changes made to a plan will change its status to "in process", requiring that it be "released" again.

Alternately, you may choose to import an Inspection Plan into PC-DMIS that you have exported from PC-DMIS Planner via an .ip file.

The following commands are used to create, open, save and close PC-DMIS Planner .cad files:

- Click **New** from the **Main** toolbar or select the **File | New** menu item. This displays a **Units Selection** dialog box.



  This allows you to create new PC-DMIS Planner .cad file named "Untitled" based on the selected units. Be sure to select the same units as the part model you will use since you will not have the ability to change the measurement units later. The .cad file is not saved at this point. When you try to close the .cad file, Planner will request if you would like to save it.

- Click **Open** from the **Main** toolbar or select the **File | Open** menu item. This opens the **Open** dialog box. Browse to and select the needed .cad file. Click **Open** to open the file.

- Click **Save** from the **Main** toolbar or select the **File | Save** menu item (or **Save As**) to save the file or release plans. The **Release Plans** dialog box appears. Saving and releasing plans is discussed in more detail in the "Saving and Releasing Inspection Plans" subtopic of the "Working with Inspection Plans" topic.

- Select the **File | Close** menu item to close the current file without closing PC-DMIS Planner.

Working with Inspection Plans

The pane on the left side of the screen is called the Plan window. It houses your inspection plans and all commands contained within each inspection plan. Initially the Plan window will be blank when creating a new .cad file until you choose to create a new inspection plan or import an existing plan.

- Each inspection plan you create can be expanded or collapsed by clicking the + or - keys respectively.

- You can select multiple commands inside of inspection plans in contiguous order by holding down the SHIFT key and clicking on two different command IDs. All IDs between them will be selected. You can also select multiple non-contiguous commands by holding down the Ctrl key and clicking each command you want selected. Planner highlights associated features in the Graphic Display window.

- Selecting a plan command in the list highlights features in the Graphic Display window that are associated with that command.

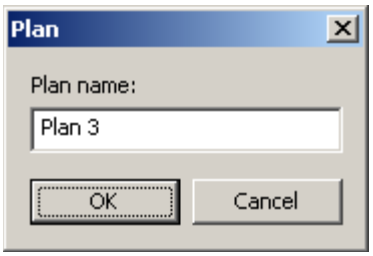
- Feature IDs copied from one plan and pasted into another share the same underlying feature information. Therefore, any change made to the feature ID in one plan is automatically made to all other copied feature IDs, regardless of the plan in which they're located.

- Until inspection plans are "released" for use, they will not be available for import into PC-DMIS, even though they remain embedded inside the .cad file. In this case, the Plan window displays "in process" next to the plan title. To make them ready for use with PC-DMIS, right-click on the inspection plan's name in the Plan window and select **Release Plan** from the shortcut menu. Next to the Plan name it displays "released". See also "Saving and Releasing Inspection Plans" below.
- You can only open a single .cad file at a time in Planner. As long as one is open, the **New** and **Open** menu and toolbar items are inactive. To switch to a new .cad file, first use the **File | Close** menu item to close the current file. The **New** and **Open** items will then be enabled for selection.

Creating a New Inspection Plan

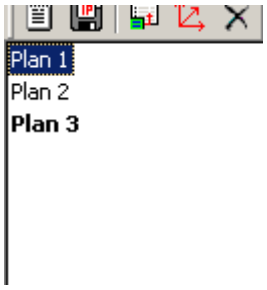
To create a new Inspection Plan, follow this procedure:

1. Open the **Plan** dialog box by doing one of the following:
   - Click **New Plan** from the **Plans** toolbar.
   - Right-click any existing plan in the Plan list and select the **Insert New Plan** option from the shortcut menu.



Plan dialog box

2. Type a name for the Inspection Plan in the **Plan name** box.
3. Click **OK** to create the Inspection Plan. This adds the specified plan to the list of available plans.



Plan List

By default, when you create a new PC-DMIS CAD file, an Inspection Plan named **Plan 1** is created. You can rename any of the Inspection Plan files by selecting it from the list, right-clicking the plan name, and then selecting the **Rename** option from the shortcut menu. Type the needed name and press **Enter**.

New Inspection Plans are embedded in the PC-DMIS Planner CAD file.

Saving and Releasing Inspection Plans

When you're done working on your inspection plan, and you want to either save it to your .cad file or you want to release a plan within the .cad file, follow this procedure:

1. Select **File | Save**, or access the **Main** toolbar (right-clicking on the toolbar area and select **Main**) and click the **Save** button.
2. The **Release Plans** dialog box appears.

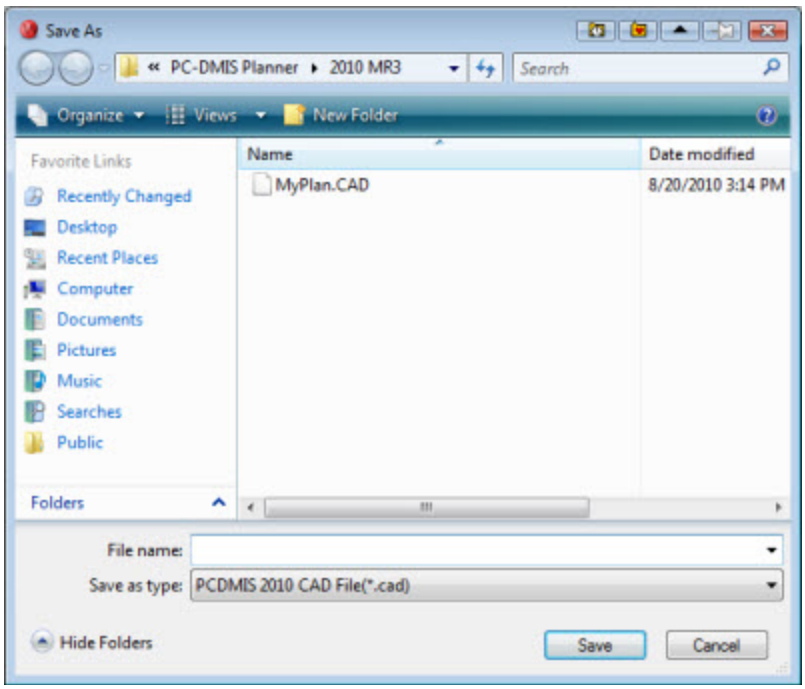
Release Plans dialog box

This dialog box provides you with a list of plans that have not been released and command buttons to easily release or save inspection plans embedded within the cad file.

This buttons are available:

- **Release All** - This removes all the inspection plans from the list, essentially marking them for release. Once you click the **Save** button, Planner actually releases the plans with the saved .cad file.
- **Release** - This removes the selected inspection plans from the list, essentially marking them for release. Once you click the **Save** button, Planner actually releases the plans with the saved .cad file.
- **Save** - This saves the .cad file and any changes made to embedded inspection plans. Any inspection plans that you selected to release will be released when you click this button as well.

  If this is the first time you've saved your changes, or if you used the **Save As** menu item, Planner will display a **Save As** dialog box.



Save As dialog box

The **Save As** dialog box provides you with the ability to save the .cad file as a different name, in a different location, or in a format that works with supported PC-DMIS versions. Supported versions are listed in the **Save as type** list. These include:

- PCDMIS CAD File (works with the most recent version of PC-DMIS)
- PCDMIS 2010 CAD File (works with the base release of PC-DMIS 2010)
- PCDMIS 2009 CAD File (works with the base release of PC-DMIS 2009)
- PCDMIS 4.3 CAD File (works with any PC-DMIS 4.3 version)

Click **Save** to complete the **Save As** operation.

- **Cancel** - Closed this dialog box.

Importing an Existing Inspection Plan

Select **File | Import | Inspection Plan** to import an existing .ip file into your current .cad file.

Adding Commands to an Inspection Plan

To add a command to the inspection plan, select where you want the new command to appear in the desired inspection plan, and then insert it into the command by using the **Insert** menu, by clicking on the GD&T symbols in the Graphic Display window, or by using available toolbars.

See the following topics to add those specific commands:

- Creating Auto Features
- Constructing Features
- Adding Dimensions to Features
- Creating Alignments
- Inserting Comments

Editing an Inspection Plan

To edit an existing inspection plan, right-click on the inspection plan command and select **Edit**. See "Shortcut Menus in the Plan List".

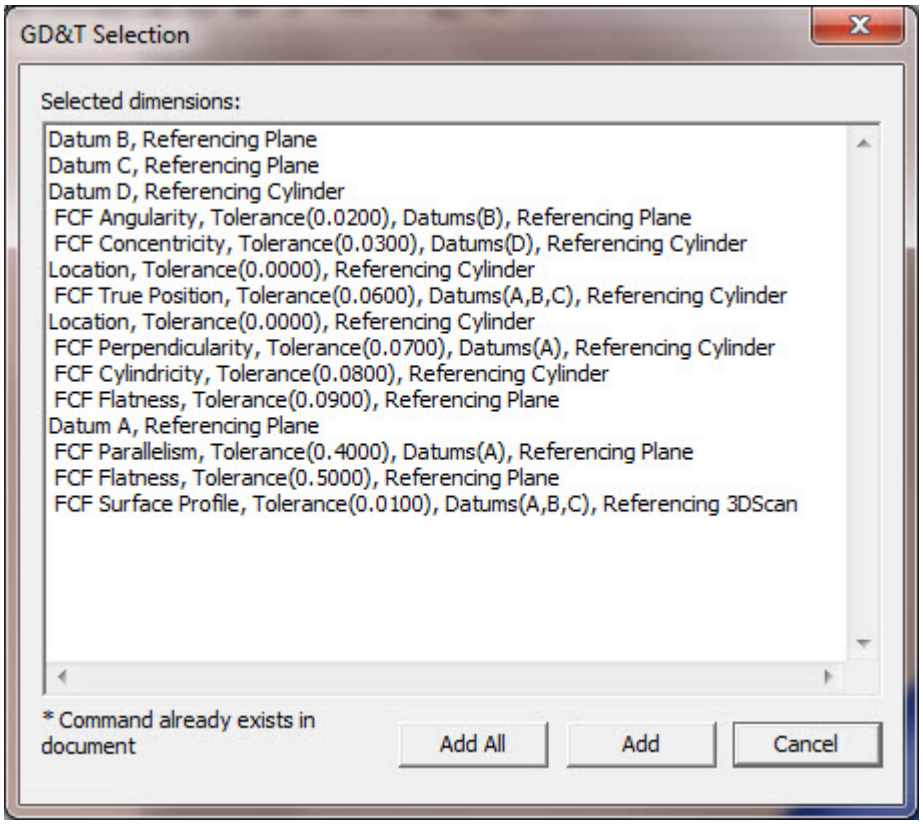
To edit a specific command, you can either double-click on it in the **Plan** list or select it and press F9.

Adding all GD&T components to an Inspection Plan

To select all GD&T components and integrate them into an Inspection Plan that can be used to quickly create and perform a 100% inspection, the **Select All GD&T** feature is used.

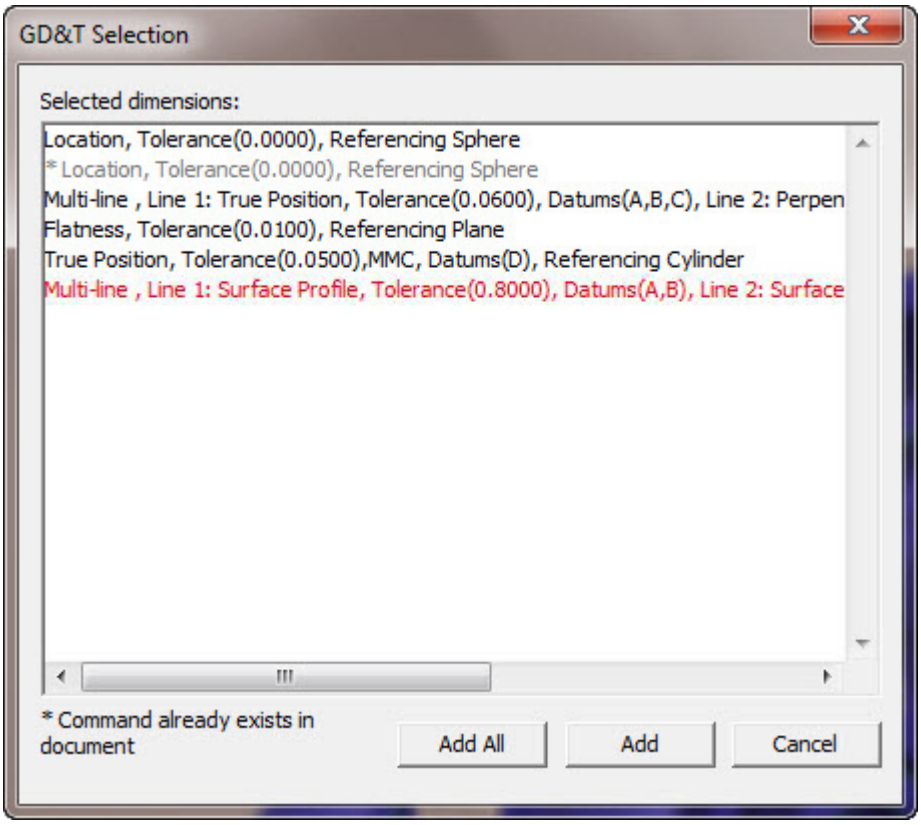
Accessing this feature is done in one of the following ways:

- Select **Insert | Dimension (Plan) | Select All GD&T** from the main menu to display the **GD&T Selection** dialog shown below.



  - Click on the **Add All** button to insert all items into the Inspection Plan.
  - Click on individual items to highlight them, then click the **Add** button to insert only those highlighted items into the Inspection Plan.
  - Click the **Cancel** button to return to Planner without changing the Inspection Plan.

- Select the **Select All GD&T** button from the **Dimensions (Plan)** toolbar to display the GD&T Selection dialog screen described above.
- Press the **<Ctrl>-A** hot key combination from the PC keyboard to display the GD&T Selection dialog screen described above.

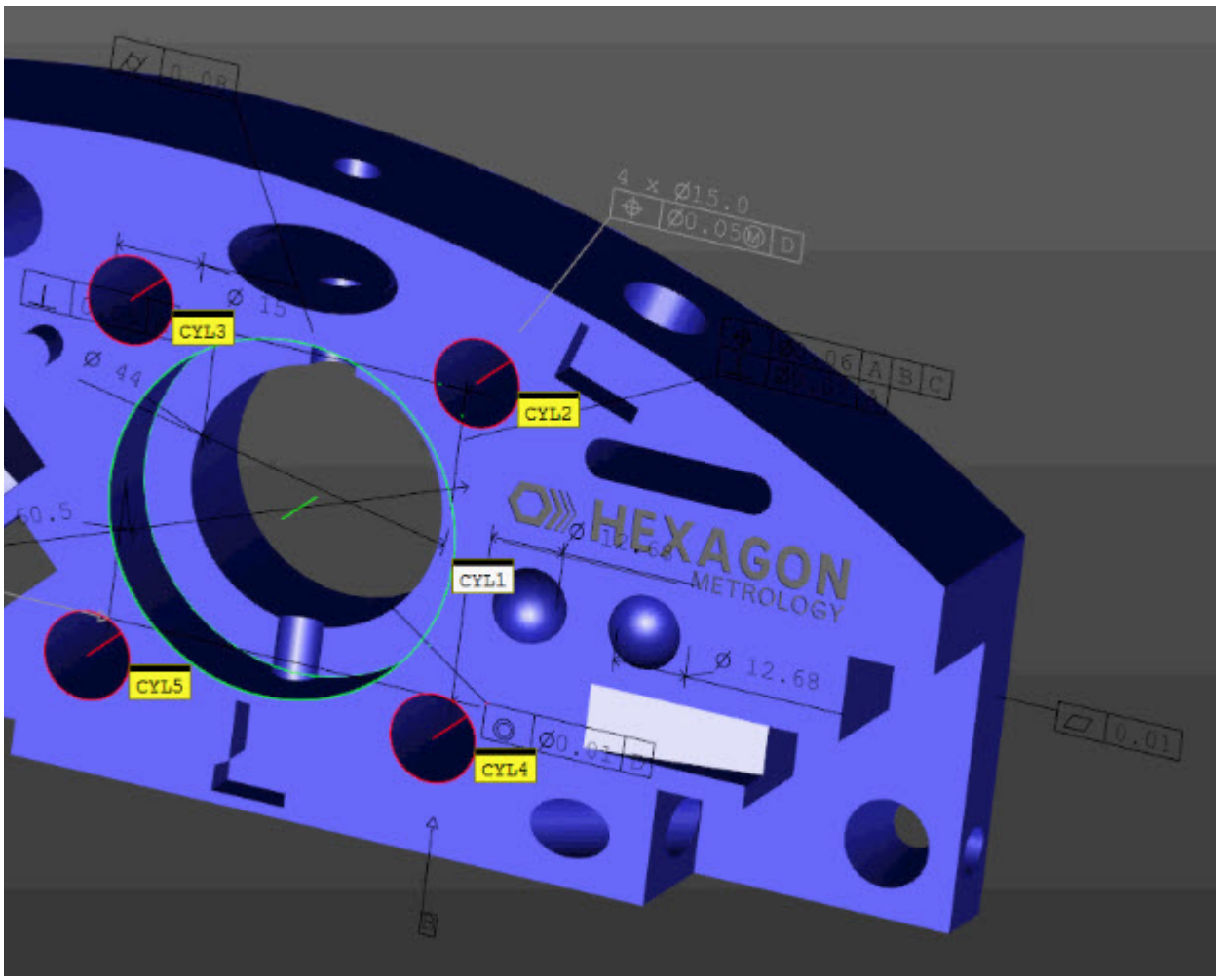


If selected GD&T elements have no geometry features created, they will show up in red. These incomplete GD&T elements cannot be added to the document as is. If you attempt to add any command to the document tagged in this way, a warning message will display stating that the item will not be added.

In addition, GD&T elements that already exist in the document show up as grey in this dialog with an asterisk to the left of each item. If you either click "Add" from that GD&T Selection dialog or do a single GD&T selection in the CAD window, you'll get a warning message saying that it could not create the feature, and then it will open up the appropriate GD&T dialog (FCF dialog, Datum dialog, etc.). At this point, the user can choose an appropriate feature to use, or cancel out of the dialog.

GD&T Callout Multiplier

Some GD&T callouts will have a multiplier signifier which is preceded with a string such as "4 x" or "2 x". This means that it applies to multiple features.



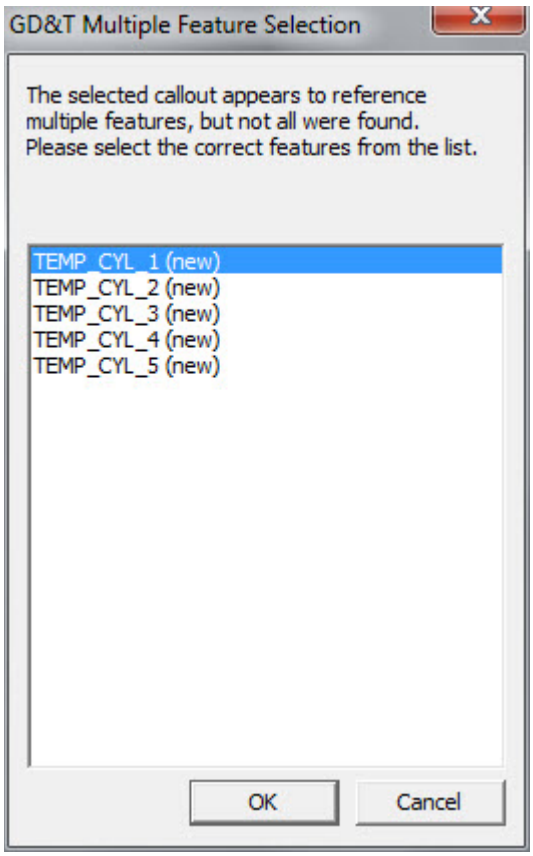
Example of a GD&T Callout with a multiplier signifier

Even though the leader line points to one cylinder, the FCF in the above screen shot applies to all four of the pattern holes (CYL2, CYL3, CYL4 and CYL5).

If the CAD GD&T callout is properly connected to all four holes, the proper four cylinders will be created when this FCF callout is selected.

Often, the GD&T Callouts lack the proper connections to all of the features it is supposed to apply to, and ends up only being connected to the feature that the leader line points to.

If the callout has a multiplier signifier and the GD&T selection is unable to create the proper number of features based on that multiplier signifier, it will bring up the **GD&T Multiple Feature Selection** dialog.



GD&T Multiple Feature Selection dialog box

Note: The **GD&T Multiple Feature Selection** dialog only works with cylinders.

The **GD&T Multiple Feature Selection** dialog displays a list of all of the cylinders in the CAD model that match the diameter of the cylinder that was found. This list is ordered so that the cylinders most likely to be the proper ones are at the top.

When you select a cylinder in the dialog, it will be highlighted in the CAD window. When you have selected the desired cylinders, click the **OK** button. The dialog to create the appropriate GD&T for the selected tolerance will then be displayed normally.

Note: The **GD&T Multiple Feature Selection** dialog will not be displayed if you click the **Add All** button from the **GD&T Selection** dialog. The **GD&T Multiple Feature Selection** dialog is only displayed before the dialog is displayed to create the appropriate GD&T from the selected tolerance.

Updating an Inspection Plan using Change Manager Cad

Note: While the documentation below was written specifically for Inspection Plans, it also applies to updating part program files created from Datalog files as well.

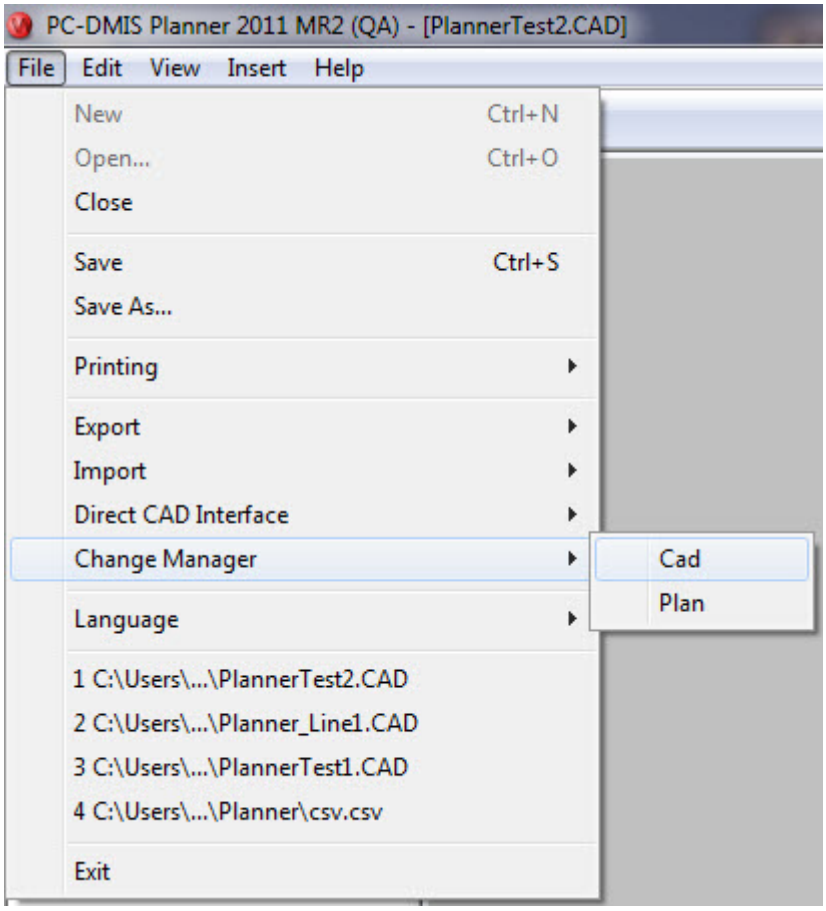
Change Manager Cad

The Change Management capabilities of PC-DMIS allows you to ensure that your part program, created from a previously imported version of your CAD model, is updated with the latest version based on your CAD model.

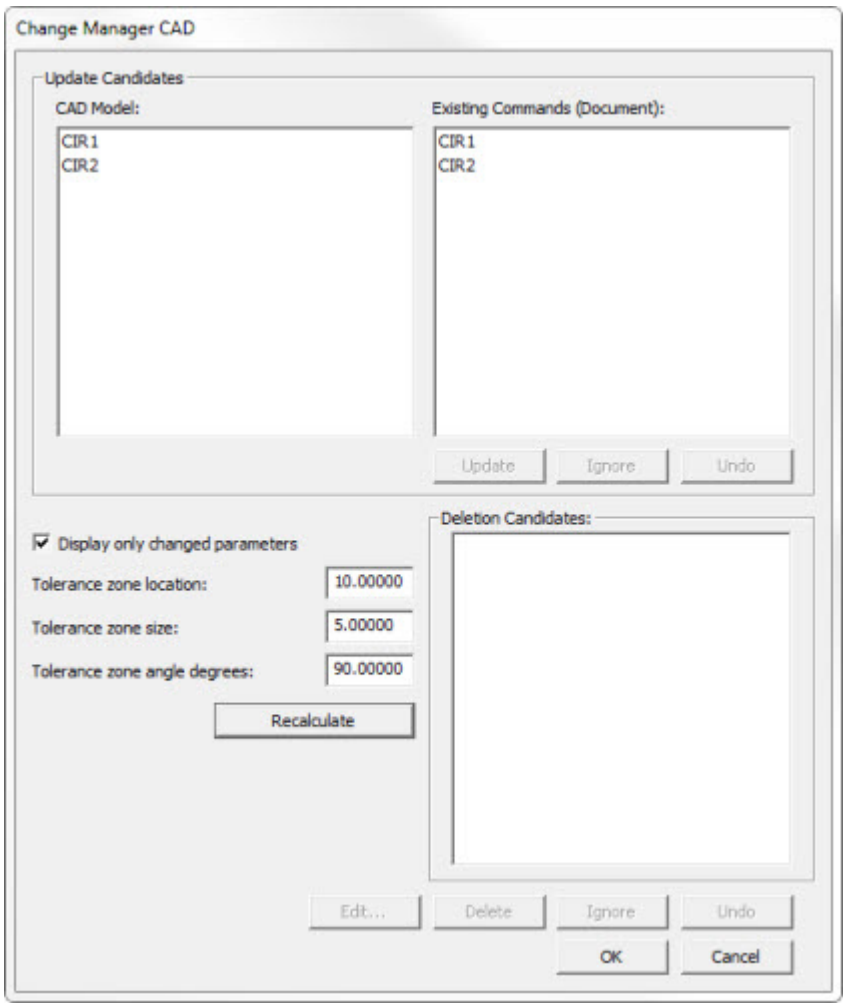
Double-clicking a feature name from any list will expand the display of parameters for that feature. This allows you to compare values before updating your part program.

To update your part program from a CAD model:

1. Select the File | Change Manager | Cad menu option.



2. The **Change Manager Cad** dialog box appears.



3. Use the following controls are available to process the differences in the part program and the imported CAD model.

Update Candidates section:

- **Update:** The selected feature is updated into the **Existing Plan** from the **CAD Model** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Update**.
- **Ignore:** The selected feature is ignored for update into the **Existing Plan** from the **CAD Model** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Ignore**.
- **Undo:** Available only after **Update** or **Ignore** has been processed. Clicking this button will **Undo** the last change made.

Deletion Candidates section:

- **Edit:** The selected feature from the **Deletion Candidates** list is can be edited when you click this option. You can select more than one feature at a time to **Edit**.

- **Delete:** The selected feature from the **Deletion Candidates** list is deleted from the existing plan when you click this option. You can select more than one feature at a time to **Delete**.
- **Ignore:** The selected feature from the **Deletion Candidates** list is ignored (removed from the list), and will not be deleted from the existing plan when you click this option. You can select more than one feature at a time to **Ignore**. Any part program feature removed from the list will remain as is in the part program.
- **Undo:** Available only after **Add**, **Delete** or **Map** has been processed. Clicking this button will **Undo** the last change made.

Miscellaneous sections:

- **Display Only Changed Parameters** check box**:** When you select this option, only the parameters that have changed will be displayed for expanded features in the lists of the **Update Candidate** area.
- **Tolerance Zone:** Type tolerance values by which features will be evaluated. Features that are within tolerance, will be listed in the **CAD Model** and **Existing Plan (Program)** lists. The following feature dimensions are evaluated:
    - **Tolerance zone location:** Provides the tolerance for *location* evaluation.
    - **Tolerance zone size:** Provides the tolerance for *size* evaluation.
    - **Tolerance zone angle degrees:** Provides the tolerance for *angle* evaluation.
    - **Recalculate:** Click on this button to recalculate the Inspection Plan with the changes.
    - **OK:** Accepts all the current changes, writing the changes into the current part program. The dialog box is also closed.
    - **Cancel:** Exits the dialog box, disregarding all the changes made.

Another functionality available in the **Change Manager CAD** dialog box is the ability to **Center** a feature or **Highlight** and **UnHighlight** features that need to be changed or deleted. This helps you to better see where changes in a CAD model have been made or how your Plan needs to be updated to match a new CAD model.

Access this functionality by right-clicking on any feature name in any of the lists in the **Change Manager CAD** dialog box (CAD Model, Existing Commands or Deletion Candidates) to display the options.

Change Management CAD options dialog

The options perform the following functions:

- **Center in CAD** - Clicking this option centers the selected feature location in the CAD Window.
- **Highlight in CAD** - Clicking this option highlights the selected feature in the CAD window. You can only highlight one feature at a time and you cannot highlight any feature in the CAD Model list if the feature does not currently exist in the Plan.
- **UnHighlight in CAD** - Clicking this option will unhighlight the highlighted feature in the CAD window.

Updating an Inspection Plan using Change Manager Plan

Note: While the documentation below was written specifically for Inspection Plans, it also applies to updating part program files created from Datalog files as well.

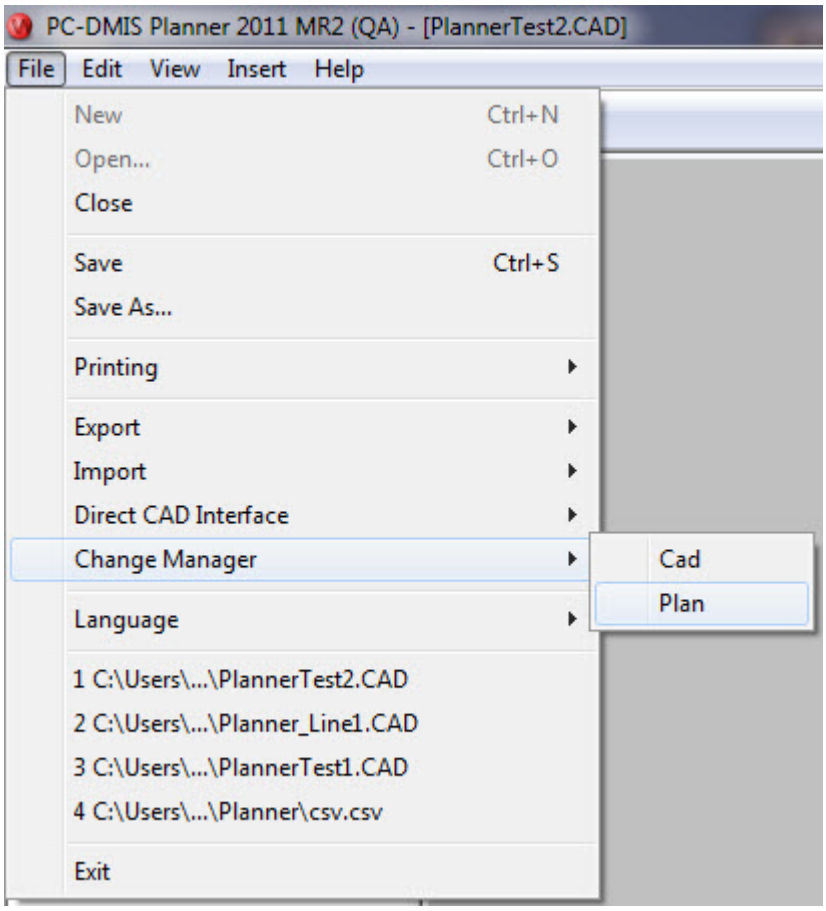
Change Manager Plan

The Change Management capabilities of PC-DMIS allows you to ensure that your part program, created from a previously imported version of your Inspection Plan, is updated with the latest version of your Inspection Plan.

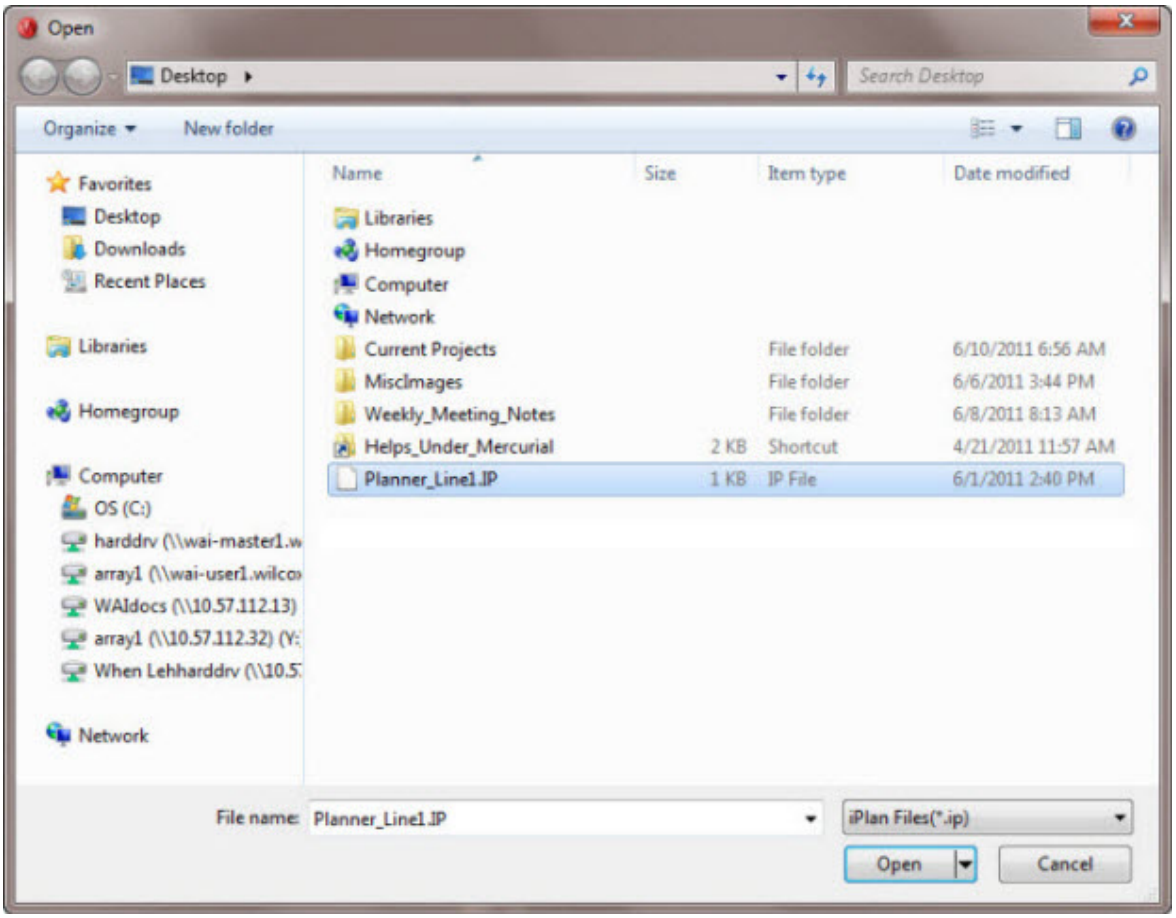
Double-clicking a feature name from any list will expand the display of parameters for that feature. This allows you to compare values before updating your part program.

To update your part program:

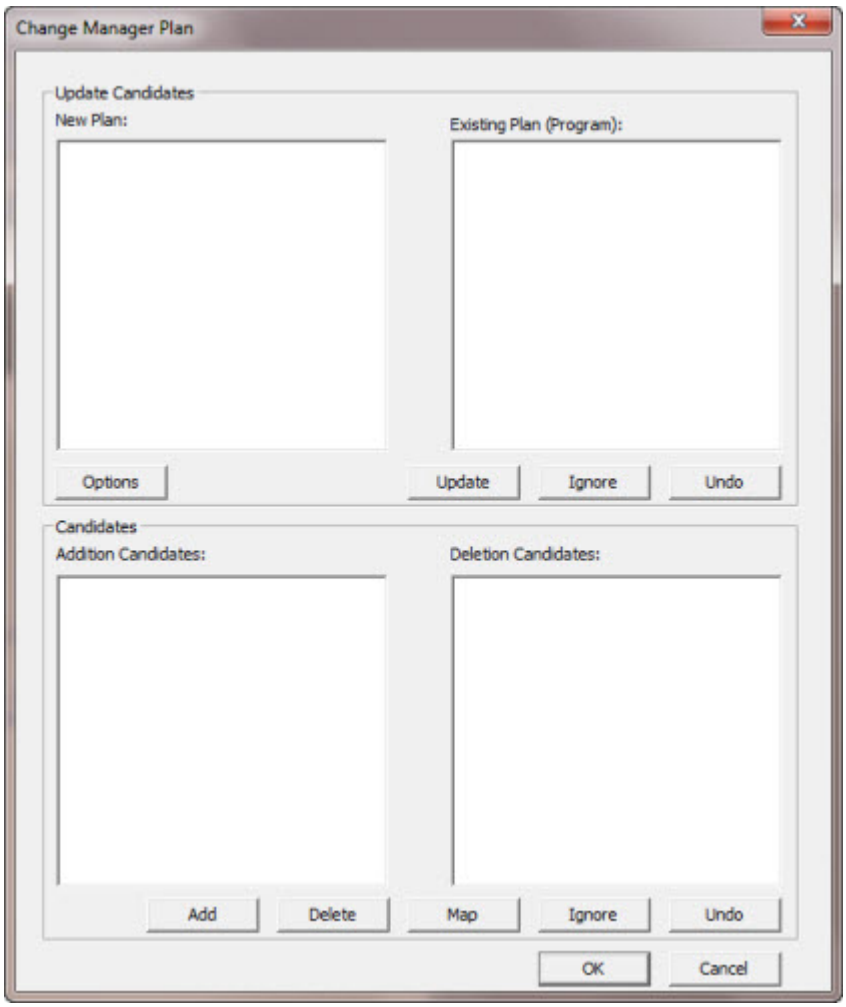
1. Select the File | Change Manager | Plan menu item.



2. The **Change Management Plan Open** dialog box appears. Navigate to the Inspection Plan to update to from the dialog box, left-click on it to highlight it then click the **Open** button to open it.



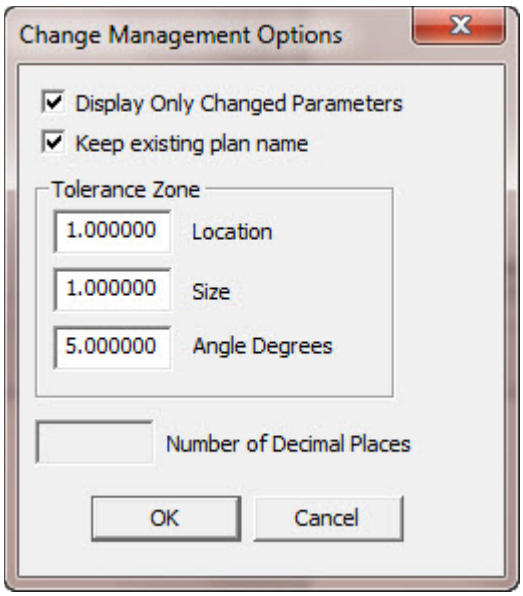
3. The **Change Manager Plan** dialog box is displayed.



The following controls are available to process the differences in the part program and the imported plan.

Update Candidates section:

- **Options**: Displays the Change Management Options dialog box.



- **Display Only Changed Parameters** check box**:** When you select this option, only the parameters that have changed will be displayed for expanded features in the lists of the **Update Candidate** area.
- **Keep existing plan name** check box: When this option is selected, feature names in the existing plan will be retained. Otherwise the feature name would be replaced with the feature name from the imported plan.
- **Tolerance Zone:** Type tolerance values by which features will be evaluated. Features that are within tolerance, will be listed in the **New Plan** and **Existing Plan (Program)** lists. The following feature dimensions are evaluated:
  - **Tolerance zone location:** Provides the tolerance for *location* evaluation.
  - **Tolerance zone size:** Provides the tolerance for *size* evaluation.
  - **Tolerance zone angle degrees:** Provides the tolerance for *angle* evaluation.
- **Number of Decimal Places**: When comparing features with the same name, PC-DMIS evaluates the tolerance within 3 decimal places for millimeter and 4 decimal places for inches.

- **Update:** The selected feature is updated into the **Existing Plan** from the **New Plan** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Update**.
- **Ignore:** The selected feature is ignored for update into the **Existing Plan** from the **New Plan** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Ignore**.
- **Undo:** Available only after **Update** or **Ignore** has been processed. Clicking this button will **Undo** the last change made.

Candidates section:

- **Add**: The selected feature from the **Addition Candidates** list is added to the bottom of the existing plan when you click this option. You can select more than one feature at a time to **Add**.

- **Delete**: The selected feature from the **Deletion Candidates** list is deleted from the existing plan when you click this option. You can select more than one feature at a time to **Delete**.
- **Map**: Available only when one feature is selected from both the **Addition** and **Deletion Candidate** lists. Clicking **Map** will replace the selected **Deletion Candidate** feature with the selected **Addition Candidate** feature. This does not allow multiple selections. You can map both like and unlike feature types (e.g circle to a point, or point to a point).
- **Ignore**: The selected feature from the **Deletion Candidates** list is ignored (removed from the list), and will not be deleted from the existing plan when you click this option. You can select more than one feature at a time to Ignore. Any part program feature removed from the list will remain as is in the part program.
- **Undo**: Available only after **Add**, **Delete** or **Map** has been processed. Clicking this button will undo the last change made.

Miscellaneous

- **OK**: Accepts all the current changes, writing the changes into the current part program. The dialog box is also closed.
- **Cancel**: Exits the dialog box, disregarding all the changes made.

Deleting Inspection Plan Elements

Select any plan or command from the Inspection Plan and remove it by doing one of the following:

- Press the **Delete** key on your keyboard.
- Click **Delete** from the **Main** toolbar.
- Select the **Edit | Delete** menu item.
- Right-click on the item and select the **Delete** option from the shortcut menu.

For each of these options you will be prompted to verify that you really want to delete the item. Click **OK** to delete the item or **Cancel** to keep it. Deleted items cannot be restored and would have to be recreated if needed.

Using the Change Manager

PC-DMIS Planner provides you with the ability to manage changes to CAD model files and update dependent inspection plans by using the **Change Manager** feature for Plan or CAD.

The following options are available for using Change Manager in Planner:

Change Manager CAD

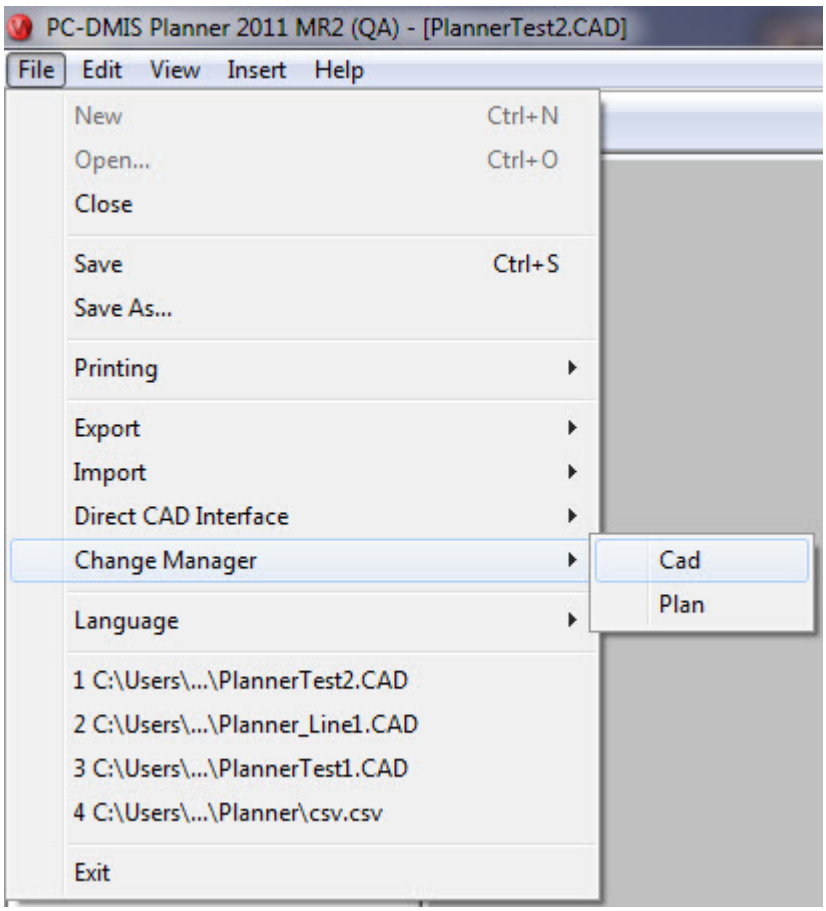
Note: While the documentation below was written specifically for Inspection Plans, it also applies to updating part program files created from Datalog files as well.

The Change Management capabilities of PC-DMIS allows you to ensure that your part program, created from a previously imported version of your CAD model, is updated with the latest version based on your CAD model.

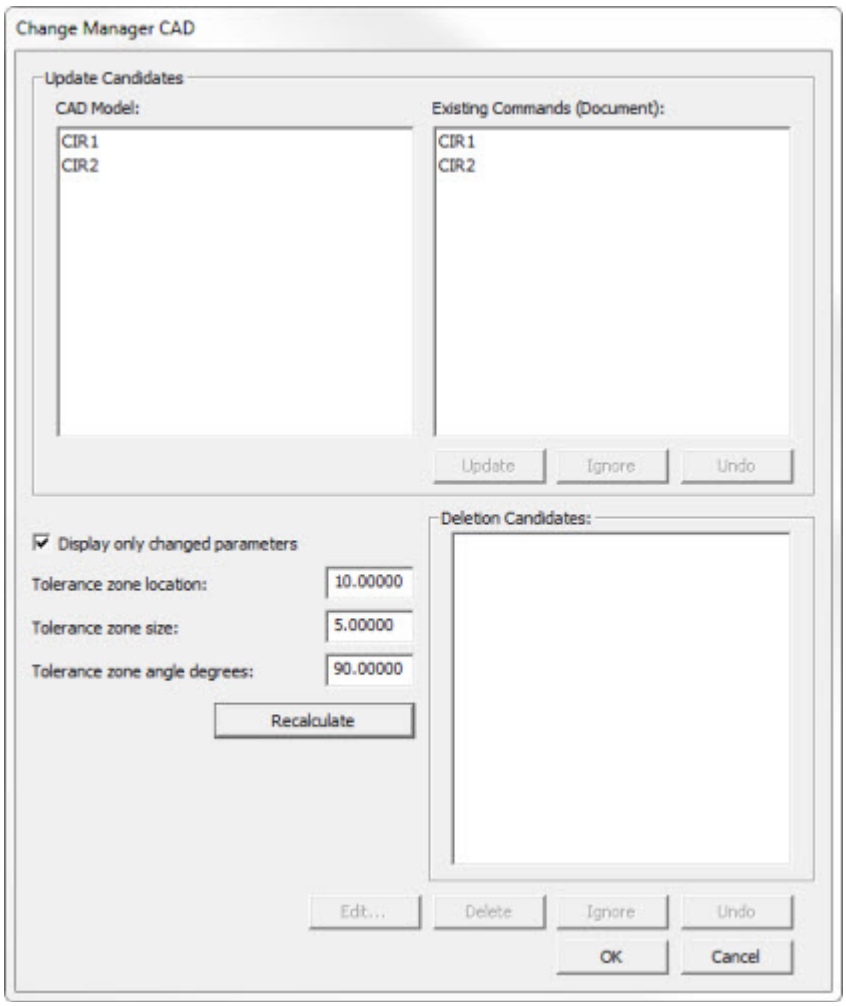
Double-clicking a feature name from any list will expand the display of parameters for that feature. This allows you to compare values before updating your part program.

To update your part program from a CAD model:

1. Select the File | Change Manager | Cad menu option.



2. The **Change Manager CAD** dialog box appears.



Change Manager CAD dialog box

3. Use the following controls are available to process the differences in the part program and the imported CAD model.

Update Candidates section:

- **Update:** The selected feature is updated into the **Existing Plan** from the **CAD Model** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Update**.
- **Ignore:** The selected feature is ignored for update into the **Existing Plan** from the **CAD Model** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Ignore**.
- **Undo:** Available only after **Update** or **Ignore** has been processed. Clicking this button will **Undo** the last change made.

Deletion Candidates section:

- **Edit:** The selected feature from the **Deletion Candidates** list is can be edited when you click this option. You can select more than one feature at a time to **Edit**.

- **Delete:** The selected feature from the **Deletion Candidates** list is deleted from the existing plan when you click this option. You can select more than one feature at a time to **Delete**.
- **Ignore:** The selected feature from the **Deletion Candidates** list is ignored (removed from the list), and will not be deleted from the existing plan when you click this option. You can select more than one feature at a time to **Ignore**. Any part program feature removed from the list will remain as is in the part program.
- **Undo:** Available only after **Add**, **Delete** or **Map** has been processed. Clicking this button will **Undo** the last change made.

Miscellaneous sections:

- **Display Only Changed Parameters** check box**:** When you select this option, only the parameters that have changed will be displayed for expanded features in the lists of the **Update Candidate** area.
- **Tolerance Zone:** Type tolerance values by which features will be evaluated. Features that are within tolerance, will be listed in the **CAD Model** and **Existing Plan (Program)** lists. The following feature dimensions are evaluated:
    - **Tolerance zone location:** Provides the tolerance for *location* evaluation.
    - **Tolerance zone size:** Provides the tolerance for *size* evaluation.
    - **Tolerance zone angle degrees:** Provides the tolerance for *angle* evaluation.
    - **Recalculate:** Click on this button to recalculate the Inspection Plan with the changes.
    - **OK:** Accepts all the current changes, writing the changes into the current part program. The dialog box is also closed.
    - **Cancel:** Exits the dialog box, disregarding all the changes made.

Another functionality available in the **Change Manager CAD** dialog box is the ability to **Center** a feature or **Highlight** and **UnHighlight** features that need to be changed or deleted. This helps you to better see where changes in a CAD model have been made or how your Plan needs to be updated to match a new CAD model.

Access this functionality by right-clicking on any feature name in any of the lists in the **Change Manager CAD** dialog box (CAD Model, Existing Commands or Deletion Candidates) to display the options.

Change Management CAD options dialog

The options perform the following functions:

- **Center in CAD** - Clicking this option centers the selected feature location in the CAD Window.
- **Highlight in CAD** - Clicking this option highlights the selected feature in the CAD window. You can only highlight one feature at a time and you cannot highlight any feature in the CAD Model list if the feature does not currently exist in the Plan.
- **UnHighlight in CAD** - Clicking this option will unhighlight the highlighted feature in the CAD window.

Change Manager Plan

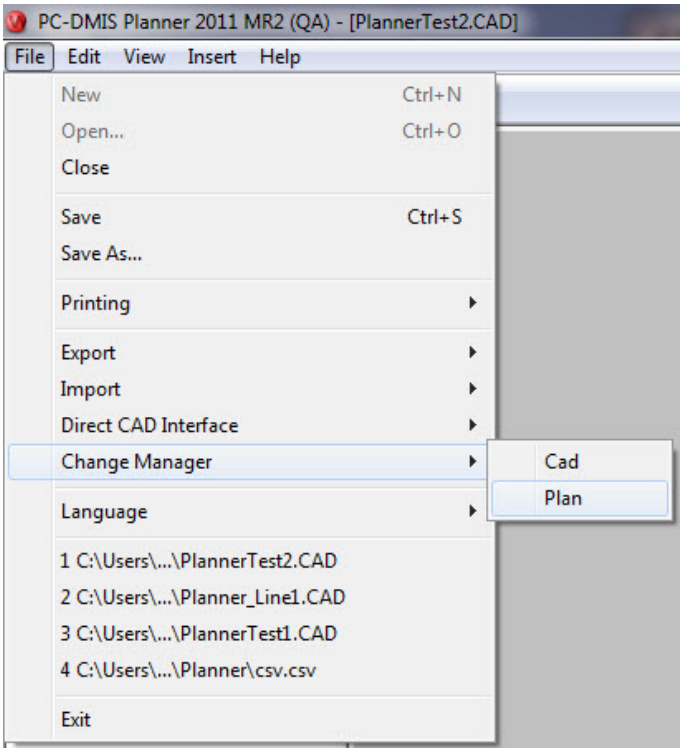
Note: While the documentation below was written specifically for Inspection Plans, it also applies to updating part program files created from Datalog files as well.

The Change Management capabilities of PC-DMIS allows you to ensure that your part program, created from a previously imported version of your Inspection Plan, is updated with the latest version of your Inspection Plan.

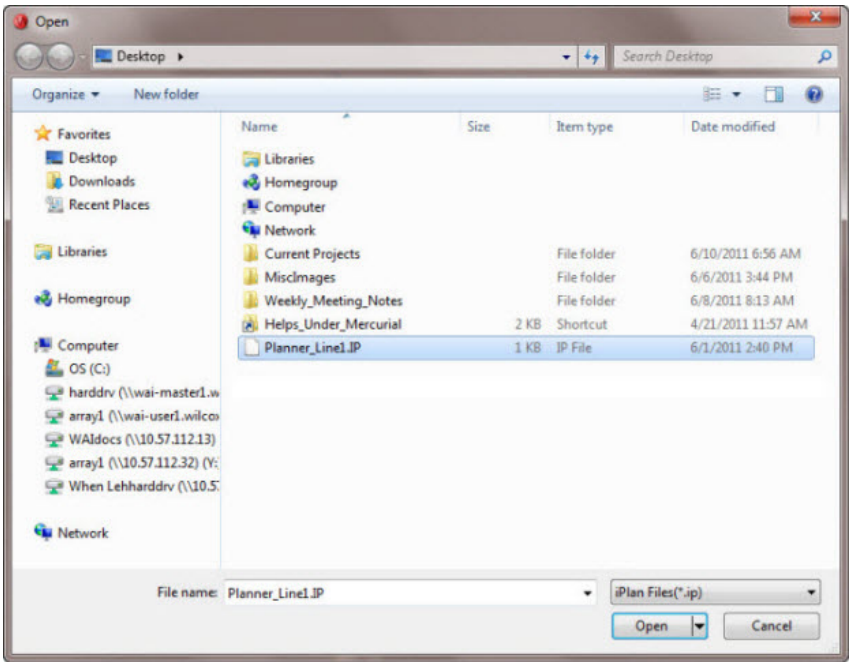
Double-clicking a feature name from any list will expand the display of parameters for that feature. This allows you to compare values before updating your part program.

To update your part program:

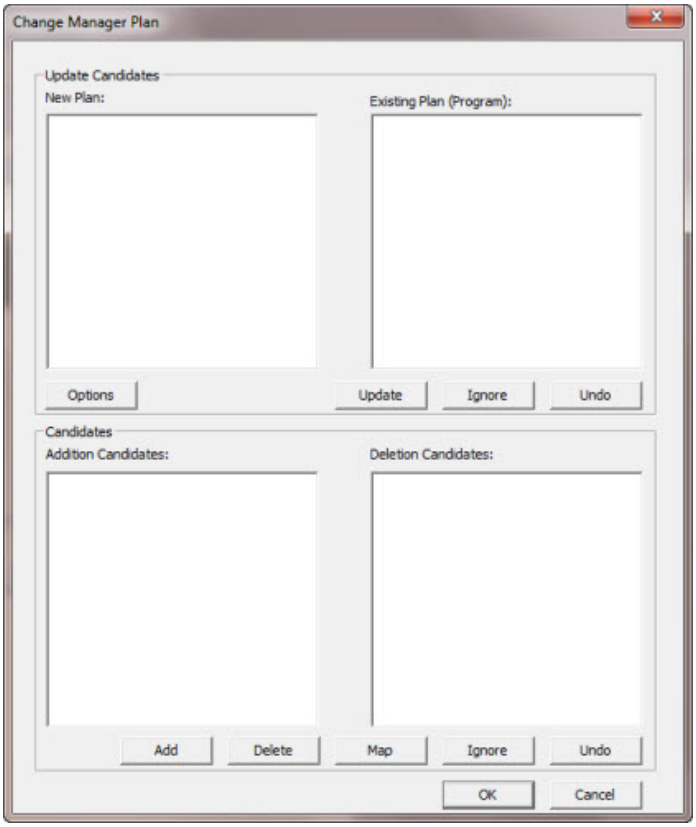
1. Select the File | Change Manager | Plan menu item.



2. The **Change Management Plan Open** dialog box appears. Use this dialog to navigate to the Inspection Plan (.IP) file that requires updates.



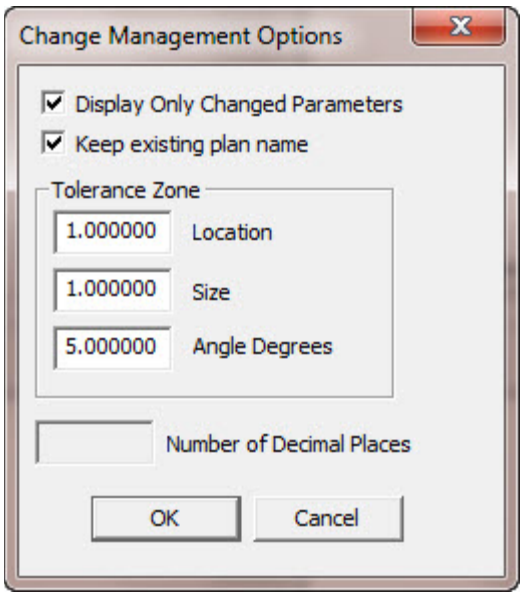
3. Left-click the file to highlight it then click the **Open** button to display the **Change Manager Plan** dialog box.



The following controls are available to process the differences in the part program and the imported plan.

Update Candidates section:

- **Options**: Displays the Change Management Options dialog box.



- **Display Only Changed Parameters** check box**:** When you select this option, only the parameters that have changed will be displayed for expanded features in the lists of the **Update Candidate** area.
- **Keep existing plan name** check box: When this option is selected, feature names in the existing plan will be retained. Otherwise the feature name would be replaced with the feature name from the imported plan.
- **Tolerance Zone:** Type tolerance values by which features will be evaluated. Features that are within tolerance, will be listed in the **New Plan** and **Existing Plan (Program)** lists. The following feature dimensions are evaluated:
  - **Tolerance zone location:** Provides the tolerance for *location* evaluation.
  - **Tolerance zone size:** Provides the tolerance for *size* evaluation.
  - **Tolerance zone angle degrees:** Provides the tolerance for *angle* evaluation.
- **Number of Decimal Places**: When comparing features with the same name, PC-DMIS evaluates the tolerance within 3 decimal places for millimeter and 4 decimal places for inches.

- **Update:** The selected feature is updated into the **Existing Plan** from the **New Plan** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Update**.
- **Ignore:** The selected feature is ignored for update into the **Existing Plan** from the **New Plan** when you click this option. Selecting a feature from one list will highlight the corresponding feature from the other list. You can select more than one feature at a time to **Ignore**.
- **Undo:** Available only after **Update** or **Ignore** has been processed. Clicking this button will **Undo** the last change made.

Candidates section:

- **Add** - The selected feature from the **Addition Candidates** list is added to the bottom of the existing plan when you click this option. You can select more than one feature at a time to **Add**.
- **Delete** - The selected feature from the **Deletion Candidates** list is deleted from the existing plan when you click this option. You can select more than one feature at a time to **Delete**.
- **Map** - Available only when one feature is selected from both the **Addition** and **Deletion Candidate** lists. Clicking **Map** will replace the selected **Deletion Candidate** feature with the selected **Addition Candidate** feature. This does not allow multiple selections. You can map both like and unlike feature types (e.g circle to a point, or point to a point).
- **Ignore -** The selected feature from the **Deletion Candidates** list is ignored (removed from the list), and will not be deleted from the existing plan when you click this option. You can select more than one feature at a time to **Ignore**. Any part program feature removed from the list will remain as is in the part program.
- **Undo -** Available only after **Add**, **Delete** or **Map** has been processed. Clicking this button will **Undo** the last change made.

Miscellaneous

- **OK -** Accepts all the current changes, writing the changes into the current part program. The dialog box is also closed.

- **Cancel** - Exits the dialog box, disregarding all the changes made.

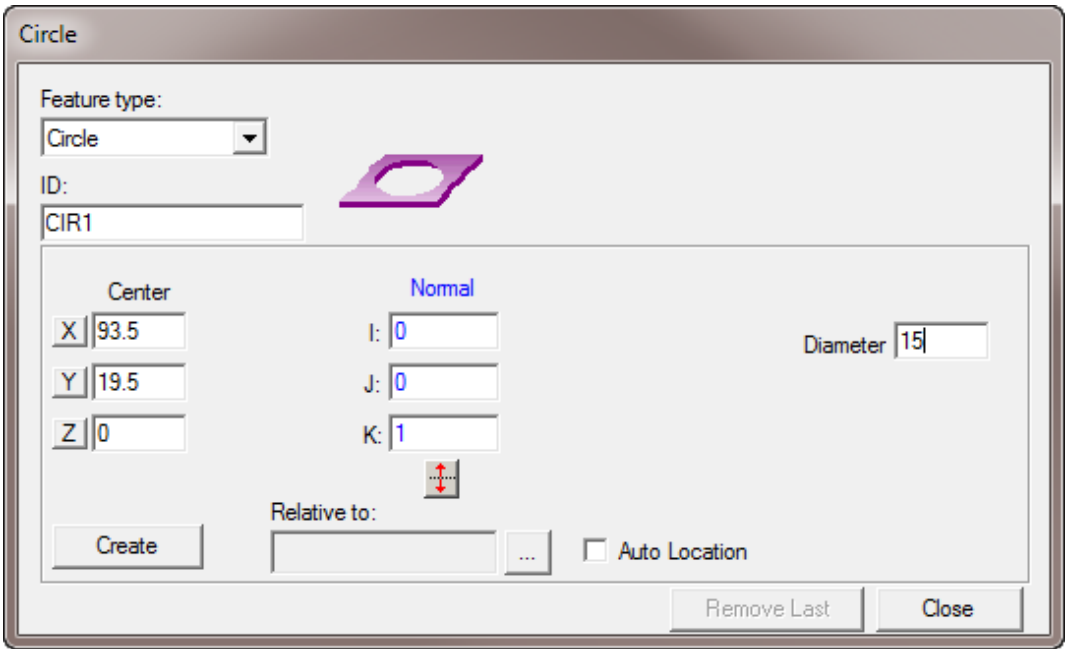
Creating Auto Features

Using the **Auto Features** toolbar or the **Insert | Auto Feature** submenu you can create and add features to your Inspection Plan.

Auto Features toolbar

If a plan does not exist, you can create a feature and PC-DMIS Planner will automatically create a new plan for you. Begin creating features by doing the following:

1. From the **Auto Features** toolbar or the **Insert | Auto Features** menu, select the feature type you want to create (listed below). A dialog box, specific to the selected feature, appears.



Example Circle dialog box

PC-DMIS Planner supports the following features:

Icon	Description	Icon	Description
	Surface Point		Notch Slot
	Edge Point		Flush & Gap
	Angle Point		Polygon
	Corner Point		Cylinder
	Line		Cone
	Plane		Sphere
	Circle		Edge Scan

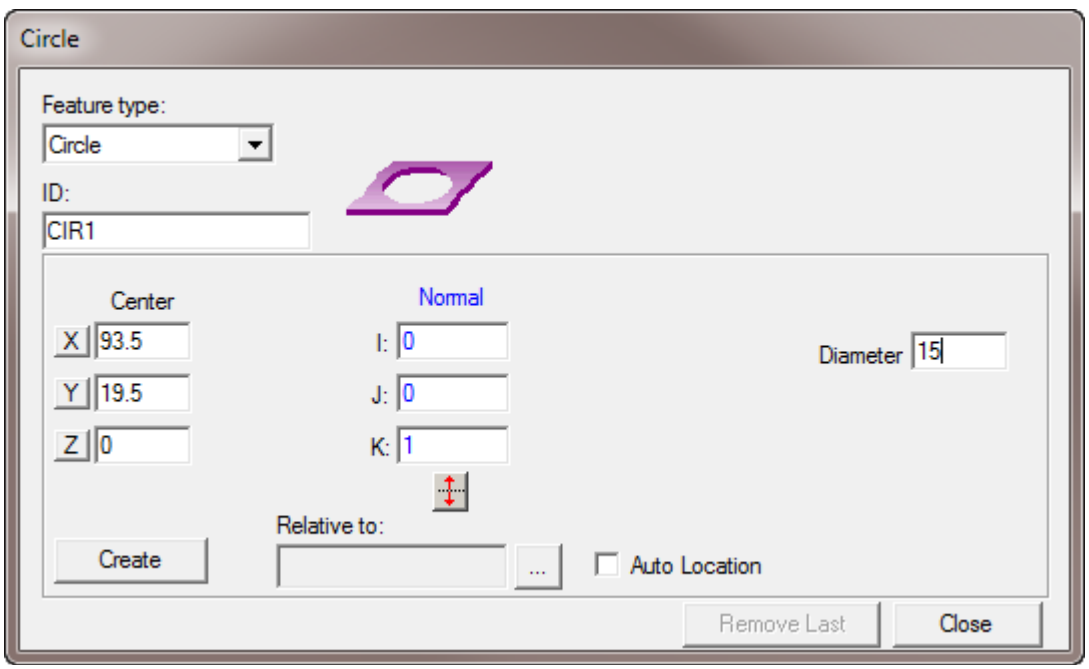
	Ellipse		3D Scan
	Square Slot		Freeform Scan
	Round Slot		

2. Using the mouse, select the face, edge, or vertex from the CAD display that will be used for the feature. The feature is created with the nominal data extracted from the CAD data. You can create multiple features for the given type by selecting from the CAD until you have created the needed features.
3. Edit the values from the "Feature" dialog box as needed. See "The Feature Geometry Dialog Box" topic for more information on each of the available settings. Also, see the "Creating an Edge Scan Feature" and "Creating a 3D Scan Feature" topics for information on creating scan features.
4. Click **Close** when you are done. Click **Remove Last** to remove that last feature that was created via selection if the result is not acceptable. If you want to create a feature by entering in the feature data instead of selecting the feature from the CAD display, type the necessary information into the dialog box fields, and click the **Create** button.

You can also use the **Auto Feature** submenu to create **Perimeter Points** features. For information on this feature type, see the "Creating Perimeter Points" topic.

The Feature Geometry Dialog Box

Depending on the feature that you have selected, the dialog box that appears will display the options needed for that feature.

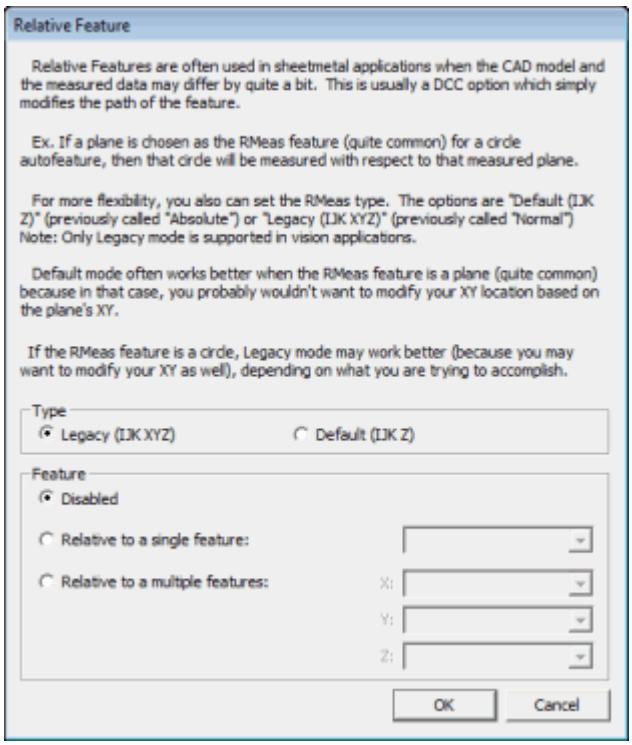


Example Auto Circle dialog box

The available options are:

- **Auto Location:** If checked, the **Location** dialog will automatically be displayed to create a location to dimension the new feature to when a new feature is added to the plan.
- **Feature Type:** Select the feature type from the drop-down list.

- **ID:** Provides the name for the feature that will be created based on the selected feature type. You can type a new ID in this box if desired; however, the ID you choose must be unique. If you type an ID that already exists, the software will display a small **Rename** dialog box telling you that the ID you typed was not unique and asking you to type a new one.
- **XYZ:** These values define the Point, Center, Start or End positions for the Feature.
- **IJK:** These values define the Surface, Edge, Normal and Angle Vectors for the Feature.
-
  The **Find Nearest CAD Element** icon tells PC-DMIS Planner to find the nearest CAD element in the Graphic Display window based on the XYZ location. Only available for Point (Surface, Edge, Angle & Corner), Line and Plane features. If you select one of the **XYZ** axis buttons, only that value will be adjusted to the nearest CAD element.
-
  The **Flip Vector** icon allows you to reverse the direction of the surface vectors. Click on Flip Vector to reverse the displayed values.
- **Diameter:** Diameter for a Circle, Polygon, Cylinder or Sphere feature.
- **Major Diam:** Diameter for the length of an Ellipse feature.
- **Minor Diam:** Diameter for the width of an Ellipse feature.
- **Width:** Width for a Square, Round or Notch Slot feature.
- **Length:** Length for a Square, Round or Notch Slot feature. This also defines the length for a Cylinder or Cone feature.
- **Gap:** Defines the distance of the Gap between the two sides of a Flush and Gap feature.
- **Num Sides:** Determines the number of sides of a Polygon feature.
- **Diameter 1:** Specifies the diameter for a Cone feature at the XYZ location of the cone.
- **Diameter 2:** Specifies the diameter for a Cone feature at the end of the length of the cone.
- **Angle:** The apex angle of a cone feature.
- **Relative to:** This allows you to keep the relative position and orientation between a given feature (or features) and the auto feature being created. Clicking the **...** button will display a **Relative Feature** dialog box where you can define the relative single feature or multiple features (one for each axis) and the type or mode of relative measure (or RMEAS). Choosing the mode is identical to adding an RMEAS/DEFAULT or RMEAS/LEGACY command into a PC-DMIS part program. Each individual feature may have a specific defined mode.



Relative Feature dialog box

Legacy (IJK XYZ)

This RMEAS mode takes into account the orientation of the reference feature.

To understand this, examine an Auto Circle with three sample hits that uses this mode. The inspection plan will tell PC-DMIS to first takes three hits on the surface around the circle and then measure the circle based on the plane's location and the surface normal vector's orientation. Therefore, if the plane is at a 45 degree angle, PC-DMIS will also measure the Auto Circle at a 45 degree angle.

Similarly, if the relative measure feature is rotated from its original orientation, then the associated feature will also be measured with the same rotational offset.

Legacy mode is also slightly smarter than Default mode, because when it looks at the reference feature it only offsets the main feature by valid axes for the reference feature type. For example, for a plane, it doesn't make much sense to shift in all directions but only by the plane's normal vector.

Default (IJK Z)

When an Auto Feature has a relative measure feature associated with it, the inspection plan will cause PC-DMIS to measure the Auto Feature in a location that is adjusted to be on the plane of the RMEAS feature.

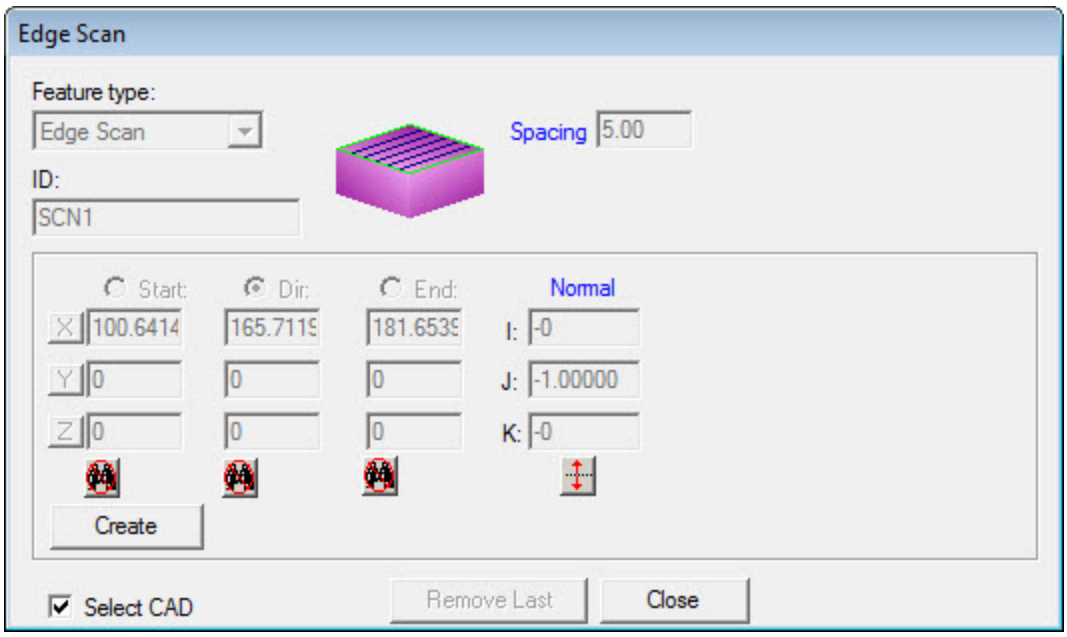
When you click **OK**, the dialog box will display chosen relative feature (or features). RMEAS references are also displayed in the Plan window if you expand that feature.

Creating an Edge Scan Feature

An Edge Scan feature allows you to create a scan feature that will be imported into PC-DMIS as a Perimeter Scan.

To create a Edge Scan Geometry Feature:

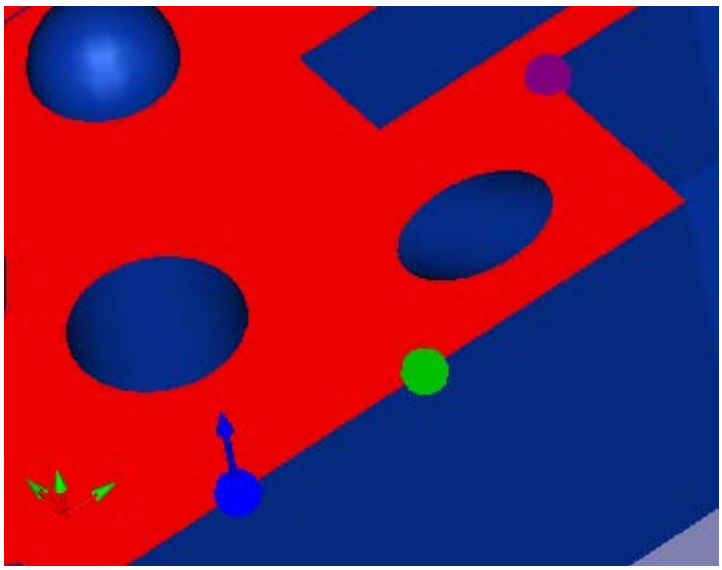
1. Click **Edge Scan** from the **Auto Features** toolbar or select the **Insert | Auto Feature | Scan | Edge Scan** menu item. The **Edge Scan** dialog box appears.



Edge Scan dialog box

2. If needed, provide an ID for the Edge Scan feature in the **ID** box.
3. Specify the **Spacing** value at which points will be added as part of the 2D Scan feature.
4. Select the **Select CAD** check box. When selected, the other parameters in the dialog box become unavailable. Select the 2D CAD surface for which the Edge Scan will be measured by clicking on the CAD. The selected surface is shown in a red highlight color as shown in the example image below.
5. De-select the **Select CAD** check box to enable the **Start**, **Direction (Dir)** and **End** options.
6. Define the **Start**, **Direction (Dir)**, and **End** points along the perimeter of the CAD feature that will be selected. To do this:
    a. Select the radio button next to **Start**. This is selected by default when you first open the **Edge Scan** dialog box.
    b. Click on the CAD near the location where the Edge Scan will begin. The **XYZ** values are derived from the Start point location, a blue start point indicator is displayed on the CAD, and the **Dir** radio button is now selected automatically. The **Normal** vector for the surface that was clicked is also provided.
    c. Click on the CAD near the location on either side of the start point to define the direction in which the scan will proceed. The **XYZ** values are derived from the Direction point location, a green direction point indicator is displayed on the CAD, and the **End** radio button is now selected automatically.
    d. Click on the CAD near the location where the Edge Scan will finish. The **XYZ** values are derived from the End point location, a purple end point indicator is displayed on the CAD, and the **Start** option is now selected again automatically.

Note: The selection of these points will cycle through each of these points until you are satisfied with the Edge Scan definition. You can however select the radio button for the point that you would like to define at any time. You may also click directly on the point indicator and drag it to the desired location.



Example: Selecting Start, Direction, End points and CAD surface

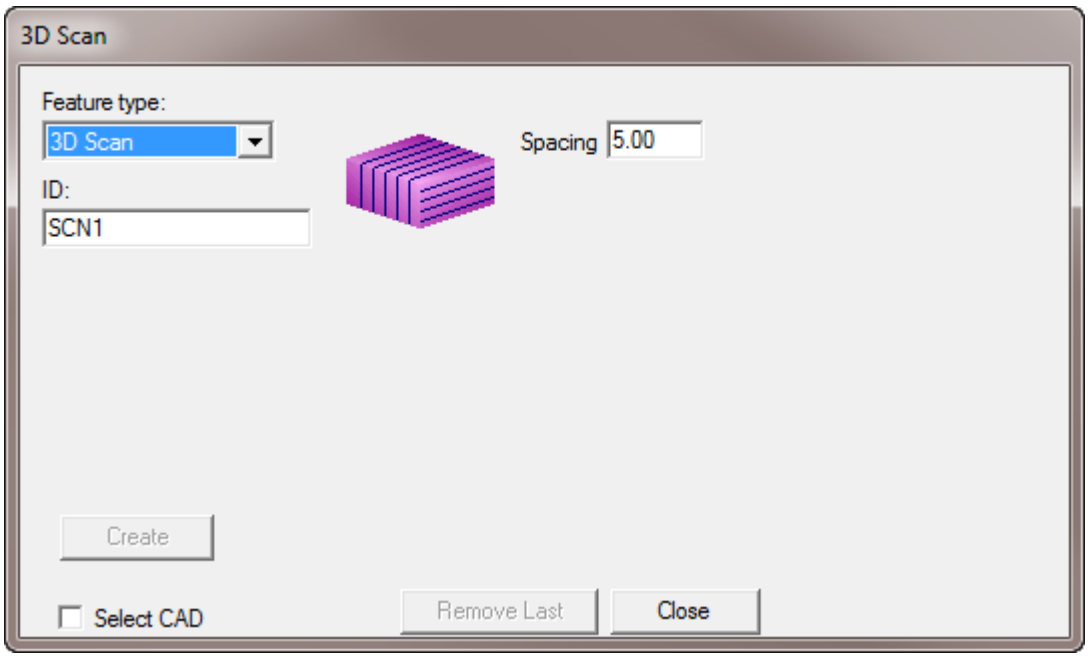
7. Click the **Find Nearest CAD Element** button if necessary to find the nearest CAD element in the Graphic Display window based on the XYZ location and any selected axis (or axes). This is especially useful when you have typed the XYZ values and need to update these values with the closest edge point on the CAD.
8. Click the **Flip Vector** button to flip the Normal vector if necessary.
9. Once the **Start**, **Direction (Dir)** and **End** options are set, the **Create** button becomes enabled as shown in the **Edge Scan** dialog image.
10. Click **Create** to add the Edge Scan feature to the Inspection Plan, or click **Close** to close the dialog box without saving.
11. Click **Remove Last** to remove the Edge Scan feature that was last added to the Inspection Plan.

Creating a 3D Scan Feature

A 3D Scan feature allows you to create a scan feature that will be imported into PC-DMIS as a Patch Scan.

To create a 3D Scan Auto Geometry Feature:

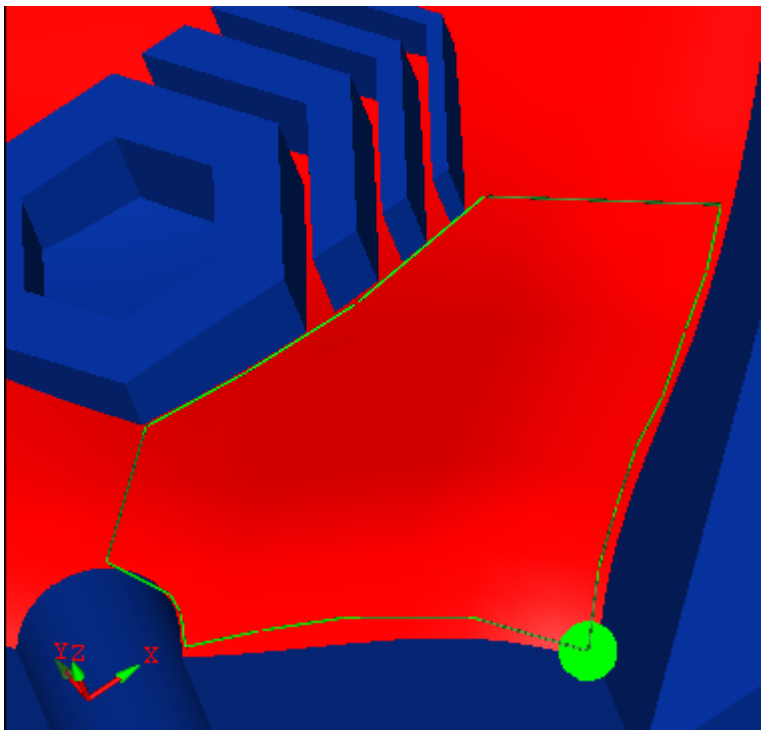
1. Click **3D Scan** from the **Auto Features** toolbar or select the **Insert | Auto Feature | Scan | 3D Scan** menu item. The **3D Scan** dialog box appears.



3D Scan dialog box

2. If needed, provide an ID for the 3D Scan feature in the **ID** box.
3. Specify the **Spacing** value at which points will be added as part of the 3D Scan feature.
4. Click on the CAD to define the start point of the 3D scan boundary. Continue clicking on the CAD surface to define the shape and extent of the 3D Scan boundary. Each click will add a new vertex of two new lines to the end of the last line and back to the start point.

Note: You can adjust the location of each of the vertices of the boundary by selecting the needed green line near the vertex that you want to move (the line will highlight in red) and dragging the vertex to the new location.



Example: Defining the 3D Scan boundary and selecting CAD

5. Select the **Select CAD** option.
6. Select the CAD surface for which the 3D Scan will be measured by clicking on the CAD. The selected surface is shown in a red highlight color.

Note: The resulting Patch Scan in PC-DMIS will create points separated by the spacing value on the selected surface within the defined boundary.

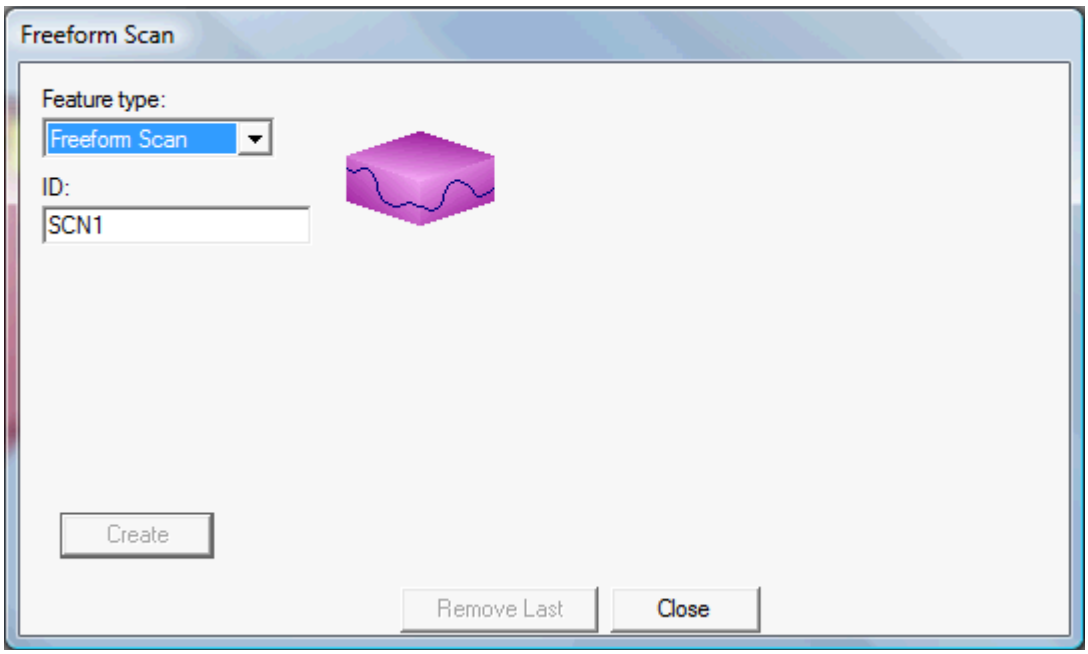
7. Click **Create** to add the 3D Scan feature to the Inspection Plan, or click **Close** to close the dialog box without saving.
8. Click **Remove Last** to remove the 3D Scan feature that was last added to the Inspection Plan.

Creating a Freeform Scan Feature

A Freeform Scan feature allows you to create a scan feature that will be imported into PC-DMIS as a Freeform Scan.

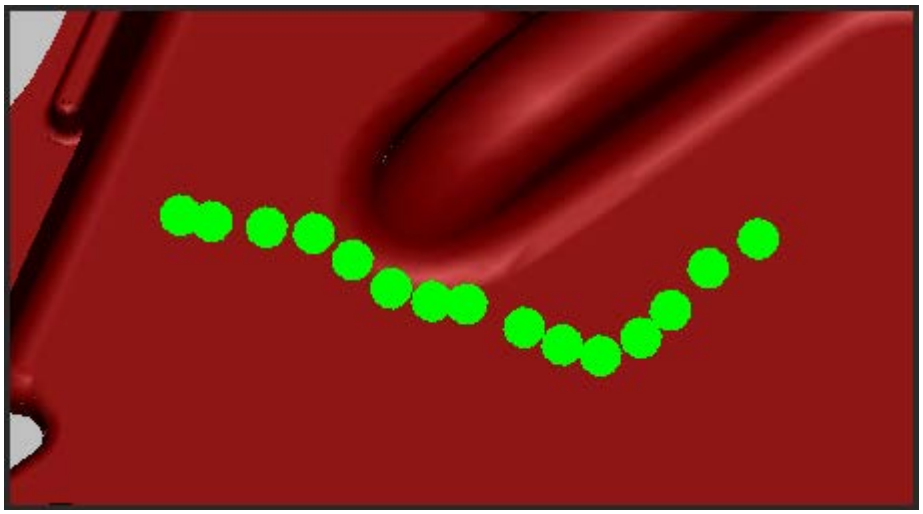
To create a Freeform Scan Auto Geometry Feature:

1. Click Freeform Scan from the Auto Features toolbar or select the Insert | Auto Feature | Scan | Freeform menu item. The Freeform Scan dialog box appears.



Freeform Scan dialog box

2. If needed, provide an ID for the Freeform Scan feature in the **ID** box.
3. Click on the CAD as desired to define the scan points used by the freeform scan. Green dots appear on the CAD model indicating where the scan takes place.



Example: Defining the Freeform Scan by clicking on the CAD

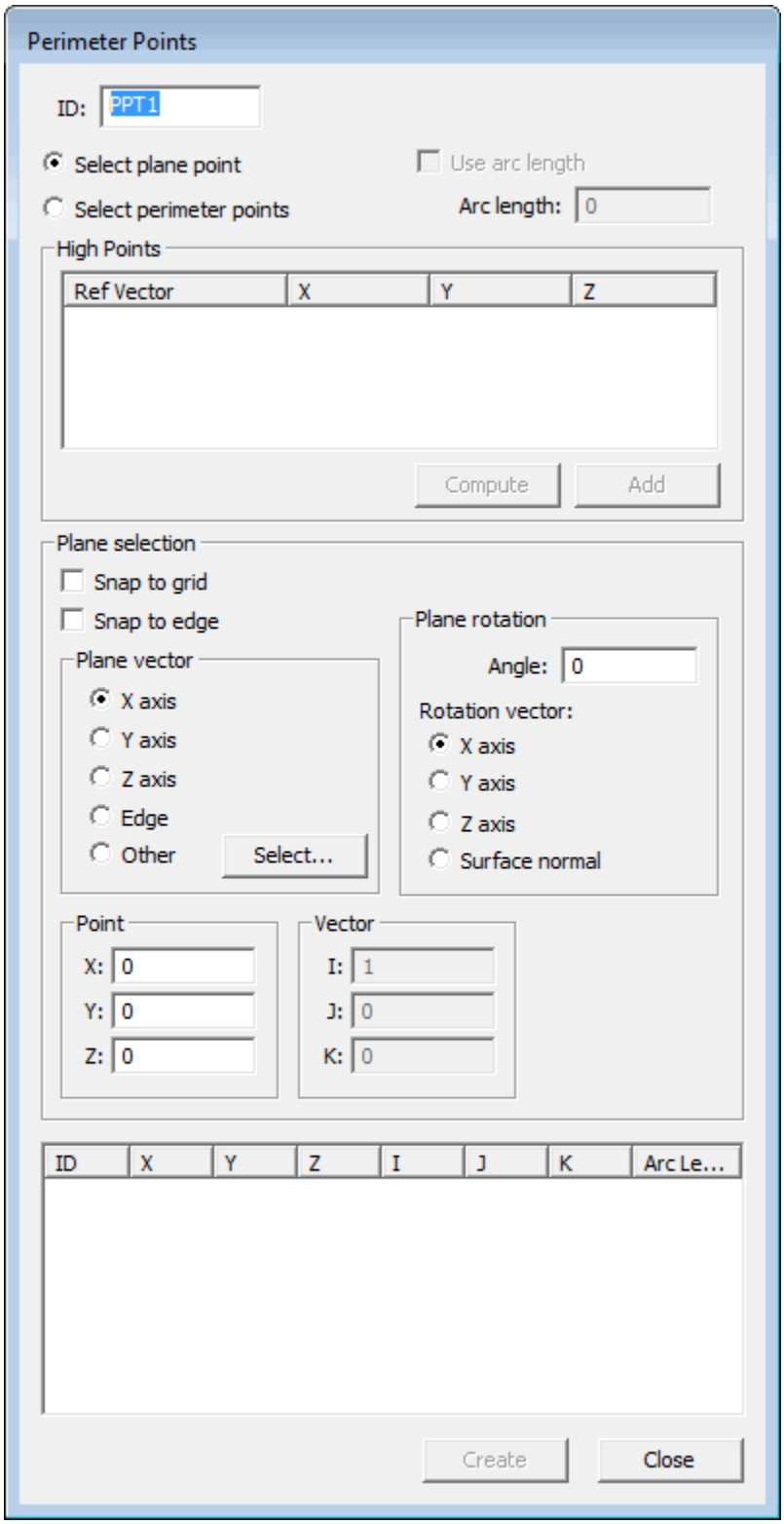
4. Click **Create** to add the Freeform Scan feature to the Inspection Plan, or click **Close** to close the dialog box without saving.

The **Remove Last** button removes the Freeform Scan feature that was last added to the Inspection Plan.

Creating a "Perimeter Points" Feature

The **Perimeter Points** dialog box lets you create and edit Perimeter Points features in the inspection plan. To access this dialog box, select **Insert | Auto Features | Point | Perimeter...**

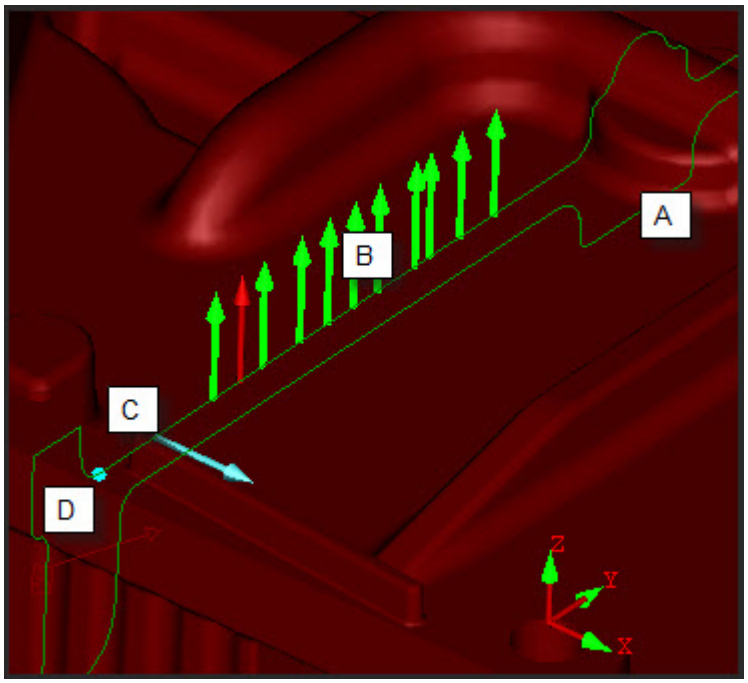
This type of feature contains a collection of points defined on a section cut profile. The points are positioned on the section cut using an arc length. The arc length measures the distance along the section cut profile from a surface edge. When imported into PC-DMIS, these points are converted into vector points.



Perimeter Points dialog box

Similar to section cuts, this dialog box lets you specify a cut plane that intersects with the CAD model. Along the intersection line, you can define where points are created. The Perimeter Points feature is then created from these points.

For example, consider this sample Perimeter Points feature:



A Sample Perimeter Points Feature in the Graphic Display window

A - The green polyline in this screen capture indicates the cut plane's location. It goes completely around the part.
B - The green arrows are perimeter points selected by clicking on the polyline. A red arrow indicates that the point is currently selected from the list of perimeter points.
C - The blue arrow that points out of the cut plane indicates the cut plane's vector.
D - A blue point appears when you select a single point from the list of points. It indicates the surface edge that the selected point is being measured from.

This may also display as a cross-hairs, depending on the DrawPointsAsSpheres registry entry located in the OpenGLSettings section of the PC-DMIS Settings Editor.

Creating a Perimeter Points Feature:

1. Define where the plane cuts the cad model.

    - Ensure that the **Select plane point** option is selected.
    - Click on the part model where you want the plane to be, or, if you want more control, type the exact point location in the **XYZ** boxes in the **Point** area. You can also use the snap option buttons to snap to the nearest 3D grid line or on the nearest surface edge.
    - In the High Points area, click the **Compute** button to show the high points in the list. For each reference vector, only the closest high point to the plane point will be shown. Determine which of the computed high points is relevant for the program, and then check the high points to be added. Click the **Add** button to incorporate them into the program.
2. Once you define the plane's location, a colored outline of the plane as it cuts through the CAD model appears (see A in the above image). Multiple colors sometimes appear along this cut plane make to make it easier to distinguish between multiple section cut profiles on the plane.

3. Choose the cut plane's direction by selecting or defining its vector. The colored cut plane outline changes to match the defined direction. The blue arrow pointing out from the cut plane shows its direction (See C in the above image).
   - Use the **Plane vector** area to have the direction match one of the major axes (**XYZ axis** option buttons), an axis that matches the edge closest to the plane point (**Edge**), or a custom vector based on mouse clicks (**Select** button), or a typed in vector (**Other**).
4. If desired, define the plane's rotation. You can use the **Plane rotation** area to define an angle and a vector about which to rotate. The colored cut plane outline changes to match the defined rotation.
5. Add perimeter points.
   - First, click the **Select perimeter points** option button. This tells Planner to interpret mouse clicks on the section cut profile to define perimeter points instead of defining the plane point.
   - Next, along the colored polyline for the cut plane, click on the line with the mouse. Planner will display a green arrow on the cut plane's polyline indicating the perimeter point's location and vector (see B above). The point is added to the **Perimeter Points** dialog box. If you try to create a perimeter point at the exact location for one that already exists in the list, Planner will simply highlight the existing point from the list. This helps you prevent creating duplicating points. This is especially useful when creating points with an arc length.
   - Continue clicking until you have the points you want.

Note: Be careful about modifying the plane's vector or rotation after you've added points; doing so, even after you've created the feature, will potentially delete some or all of the points. After the plane parameters are modified, a point will remain only if it lies on the same surface. All other points will be deleted.

6. Adjust perimeters points using the **Perimeter Points** list. To work with multiple points at once, you can either hold down the SHIFT key while clicking within the list to select a contiguous range of points. Or you can hold down the Ctrl key to select multiple points that are not in a contiguous range. Selected points will be highlighted in red in the Graphic Display window (see B above).
   - To move a point to a new location, click on the point in the list and then click at a different location on the cut plane polyline. To deselect the point after moving it, hold down Ctrl and select it again from the list.
   - To delete one or more points, select the point or points, right-click on the list, and choose **Delete**.
   - To flip the vector for one or more points, select the point or points, right-click on the point list, and choose **Flip Vector**.
7. Click **Create**. Planner inserts the feature into the Plan list as a with a "PPT" prefix.

Dialog Box Options

Select plane point option

This option is the first of two perimeter point selection modes that determine what happens when you click on the CAD model. This tells Planner to create a point on the CAD model which will become the cut plane point. The plane point's rotation and vector are controlled by the options in the **Plane selection** area.

Select perimeter points option

This option is the second of two perimeter point selection modes that determine what happens when you click on the CAD model. This tells Planner to create or edit perimeter points where you click along the cut plane's polyline (or section cut profile). If a single point is selected in the point list, then selecting a point on the section cut profile will update the selected point. Otherwise, selecting on the section cut profile will create a new point at the end of the list.

High Points area
This area provides a way to compute the highest points in the same section cut plane as the perimeter points and then add those to the program.

Compute - The **Compute** button remains disabled until you click on the model to define the section cut. Once a section cut is defined, this button computes the highest points and adds them into the **High Points** area. Check boxes appear next to each point in the list. These check boxes toggle the displayed equivalent point in the Graphic Display window.

Add - Adds any check marked points from the **High Points** area to the Perimeter Points list.

Use Arc Length check box
If marked, and you are selecting perimeter points, Planner will create points along the cut plane's polyline at the distance defined in the **Arc length** box from the nearest surface edge. The surface edge used is the one closest to the location clicked on the CAD model. **Arc Length** is grayed out if the check box is not marked.

Snap to edge and **Snap to grid** check boxes

> **Snap to edge** - This check box determines whether or not Planner snaps the plane point to the nearest surface edge or surface boundary.
>
> **Snap to grid** - This check box determines whether or not Planner snaps the plane point to the nearest grid intersection. You can use the snap to grid functionality even when the 3D grid is not showing. See the "Setting up the View" topic to enable the 3D Grid.
>
> If you select both **Snap to edge** and **Snap to grid**, Planner will snap the point to the nearest grid line that intersects a surface edge or boundary.

Plane vector area
This area allows the user to specify the plane vector. This can be one of the axes, the tangent vector of the nearest surface edge to the plane point, or a user-defined vector.

X axis - This option sets the cut plane normal to the X axis vector (1,0,0).

Y axis - This option sets the cut plane normal to the Y axis vector (0,1,0).

Z axis - This option sets the cut plane normal to the Z axis vector (0,0,1).

Edge - This option sets the cut plane normal to the nearest surface boundary tangent vector. Whenever you select the plane point, the plane normal will be updated to the nearest surface boundary tangent vector.

Other - This option lets you define the cut plane normal values manually. Once selected, you can type the IJK values in the **Plane Vector** area.

Select - This option displays a **Select Points** dialog box that lets you select geometry in the Graphic Display window to define the cut plane's vector. The selected geometry can be two points or a linear edge. This dialog box is already documented in the "Transforming a CAD Model" topic.

Plane rotation area
This area lets you rotate the plane vector about the specified rotation axis by a specified angle (in degrees). The rotation axis can be one of the axes or the surface normal at the plane point.

Angle - You can specify any angle value from -180 to 180.

X,**Y**, or **Z** axis - Rotates the plane about the specified axis by the defined angle.

Surface Normal - Rotates the plane about the plane point's surface normal vector by the defined angle.

Point area
This area shows the XYZ values of the plane point. You can manually modify the values by typing new values in the **X**, **Y**, and **Z** boxes. Note that if the point you specify does not lie on a CAD surface, the actual point that is used will be projected onto the CAD model.

Vector area
This area shows the IJK values of the plane normal vector. You can manually modify these values by clicking **Other** and typing new values in the **I**, **J**, and **K** boxes.

Perimeter Points List
This list shows all the points added to the Perimeter Points feature. The columns display its ID, location (XYZ columns), vector (IJK columns) and the Arc Length. The arc length is the distance from the surface edge that the point is being measured from. This distance is measured along the section cut profile. You can use this list to work with existing points.

You can edit the ID and Arc Length values. To do this, click once to select a point. Click again on either that point's ID or Arc Length values. Type a new value.

You can use SHIFT and Ctrl keys to work with multiple points. Once you have selected points, right-click on the selection and a small popup menu appears with these options:

Flip Vector - This flips the vector of all the selected points.

Delete - This deletes all of the selected points.

Create (or **OK**) button
The **Create** button creates the Perimeter Points feature and adds it to the Plan. The dialog box remains open after clicking **Create**, allowing you to create multiple Perimeter Points features.

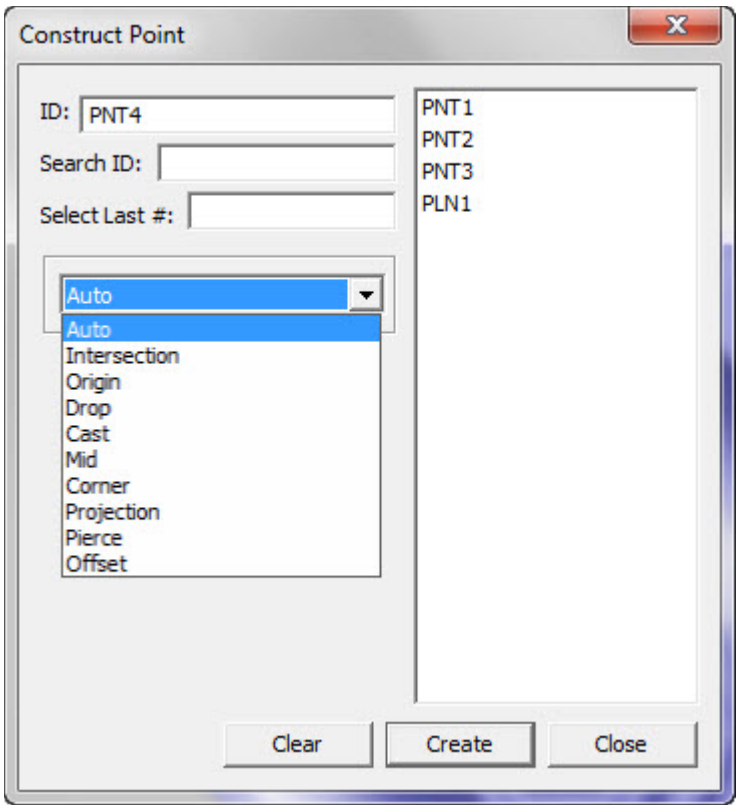
If you are editing an existing Perimeter Points feature, this button changes its text to read **OK**. Clicking **OK** applies changes made to the feature and closes the dialog box.
Close (or **Cancel**) button
This button closes the **Section Cut** dialog box.
If you are editing an existing Perimeter Points feature, this button changes its text to read **Cancel**. Clicking **Cancel** closes the dialog box discarding any changes made.

Constructing Features

Using the **Constructed Features** toolbar or the **Insert | Constructed Feature** menu you can create constructed features for your Inspection Plan.

If a plan does not exist, you can create a constructed feature and PC-DMIS Planner will automatically create a new plan for you. Begin constructing features by doing the following:

1. From the **Constructed Features** toolbar or the **Insert | Constructed Feature** menu, select the feature type you want to create (listed below). A dialog box, specific to the constructed feature you selected, appears.



Example of Constructed Feature dialog box - Constructed Point

PC-DMIS Planner supports the following constructed features:

Icon	Description	Icon	Description
	Constructed Point		Constructed Slot
	Constructed Line		Constructed Generic Feature - See "Creating a Generic Feature"
	Constructed Plane		Constructed Cylinder
	Constructed Feature Set		Constructed Cone
	Constructed Circle		Constructed Sphere
	Constructed Ellipse		

2. If needed, provide an **ID** for the constructed feature.
3. Select the features that will be used to create the constructed feature by:
   - Selecting the features by clicking the feature name from the list.
   - Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.

- Selecting the last of a certain number of features by typing the needed number in the **Select Last #** box. For example, if you wanted to select the last four features that were created, simply type 4 and press the TAB key. PC-DMIS Planner will highlight the features selected in the **Feature** list box.
- Click **Clear** to clear the previously selected features.

4. Select the construction operation to perform from the drop-down listing. See "Constructed Feature Operations" for more information on the available operations.

5. Click **Create** to perform the operation and add the constructed feature to the Inspection Plan, or click **Close** to close the dialog box without saving.

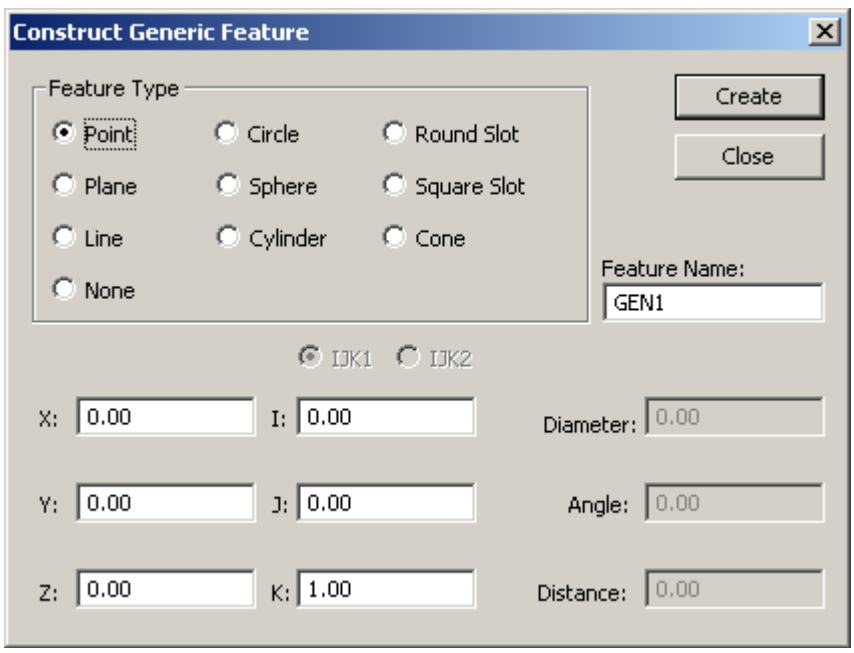
Note: When you create a **Constructed Feature Set** a point is created at the average of all the input centroids with the specified ID.

Creating a Generic Feature

The process for creating a constructed Generic feature is a little different from the other constructed features. It does not perform operations on other existing features, rather it uses input values to create a specified feature type.

To create a Constructed Generic Feature:

1. Click Constructed Generic Feature from the Constructed Features toolbar or select the Insert | Constructed Feature | Generic menu item. The Construct Generic Feature dialog box appears.



Construct Generic Feature dialog box

2. Select the **Feature Type** that you would like to create.

3. If needed, provide an ID for the Generic feature in the **Feature Name** box.

4. Specify the **XYZ** values to define the centroid of the feature.

5. Specify the **IJK** values to define the surface vector. For **Round** and **Square Slots**, Use the **IJK1** and **IJK2** values and then define the associated IJK values. **IJK1** is used for the surface vector of the slot, while **IJK2** is used for the angle vector of the long axis of the slot.

6. For Circle, Cones, Cylinders & None Features, define the **Diameter**.
7. Provide the **Angle** of the vertex of a Cone feature.
8. Specify the length of a Line, height of a Cone or Cylinder, or the length of the long axis of a Round or Square Slot feature via the **Distance** value box.
9. Click **Create** to add the Generic constructed feature to the Inspection Plan, or click **Close** to close the dialog box without saving.

Constructed Feature Operations

The list below shows available construction operations. It discusses possible feature types under each operation section.

- **2D** or **3D Line** - Creates a constructed line feature that is oriented relative to 2 or 3 axes.
- **At Origin** - Constructs a point at the alignment origin. No input features are needed.
- **Auto -** PC-DMIS Planner determines the best operation for the constructed feature based on the selected features. This operation is not available for Constructed Slots, Sets or Generic features.
- **Best Fit** - Constructs a Best Fit feature from the inputs. Using the **Best Fit** operation, you can create a *line, plane, circle, ellipse, slot, cylinder, cone,* or *sphere.* The following table lists how many inputs you will need to construct your feature.

Best Fit Feature To Construct	Number of Input Features Needed
Line	2
Plane	3
Circle	3
Ellipse	1
Slot	1
Cylinder	6
Cone	6
Sphere	5

- **BF** (Best Fit) **Recomp** - Constructs Best Fit with Recomp feature using inputs. At least *one input must be a point*. Using the **Best Fit with Recomp** operation you can create a *line, plane, circle, ellipse, slot, cylinder, cone,* or *sphere.* The number of inputs you will need to construct your feature is the same as the inputs for Best Fit operations (see table above).

Best Fit and **Best Fit with Recomp** cylinders can be constructed from two circles. The input circles must be of type *Constructed Best Fit (BF)*, *Best Fit Recompensate (BFRE)*, or *Measured circles*. The total amount of hits in each circle must be at least three.

- **Cast** - Constructs the specified feature type at centroid of the input feature. Only one input feature of any type is needed. Using the Cast operation you can create a *point, line, plane, circle, ellipse, cylinder, cone, or sphere.*
- **Corner** - Constructs a corner point based on the input features.
- **Drop -** Constructs a point feature when first feature is dropped onto the second line feature. Two input features are needed. The 1st feature *can be any type*. The 2nd feature must be a *cone*, *cylinder*, *line*, or *slot*.

- **High Point** - Creates a constructed plane based on at least three input features. From the input features, PC-DMIS Planner will construct a Best Fit plane and then rotate the plane so that the highest three points reside within the plane surface.

- **Intersection** - Constructs the specified feature intersection of the 2 input features. Using the **Intersect** operation, you can create a *point, line,* or *circle.*

    - **Intersect Point** - Constructs a point at the intersection of the linear attributes of 2 features. The input types for either input feature should be a *circle, cone, cylinder, line* or *slot*.
    - **Intersect Line -** Constructs a line at the intersection of two planes. The input features must be *planes*.
    - **Intersect Circle -** Constructs a circle at the intersection of a circle, cone, cylinder, or sphere feature with a plane, cone or cylinder feature.
- **Mid** - Constructs a point, line, or plane midway between the two input features..

    - **Midpoint or Midplane** - Constructs the point or plane feature midway between the centroids of *any input feature*.
    - **Midline -** Constructs the line midway between *line*, *cone*, *cylinder*, *slot*, or *plane* input features**.**
- **Offset** - Constructs the specified feature based on the centroids of the input features and moved by the offset distance specified. The offset distance is given in terms of XYZ values. Using the **Offset** operation, you can create a point, line, or plane. The following table lists how many inputs you will need to construct your feature.

Offset Feature To Construct	Number of Input Features Needed
Point	1
Line	2
Plane	3

- **Parallel** - Constructs a line or plane feature parallel to first input feature and passing through the second input feature.

- **Perpendicular** - Constructs a line or plane feature perpendicular to first input feature and passing through second input feature.

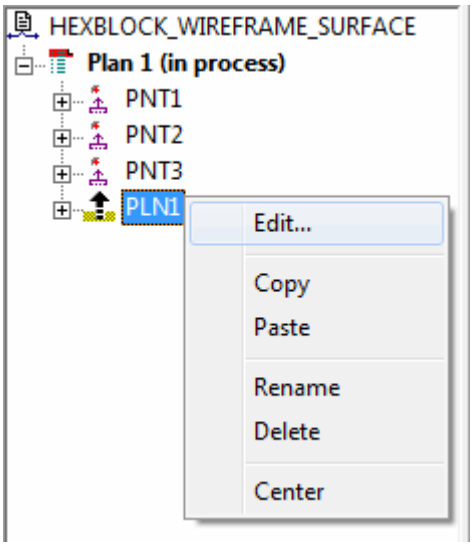
- **Pierce** - Constructs a point feature where the 1st input feature pierces the 2nd input feature. The order that the input features are selected is important. The 1st feature can be a *cone*, *cylinder*, *line*, *slot*, *circle* or *ellipse*. The 2nd feature can be a *cone*, *cylinder*, *plane*, *sphere*, *circle* or *ellipse*. If the 1st feature is a line, than the direction is important.

- **Projection -** Constructs a point, line, plane, circle, cylinder, cone, or sphere onto the selected plane, or the current work plane, when no plane is specified.

- **Reverse** - Constructs a line, plane, circle, cylinder, cone or sphere with the same centroid and the same feature type as the input feature with the vector reversed.

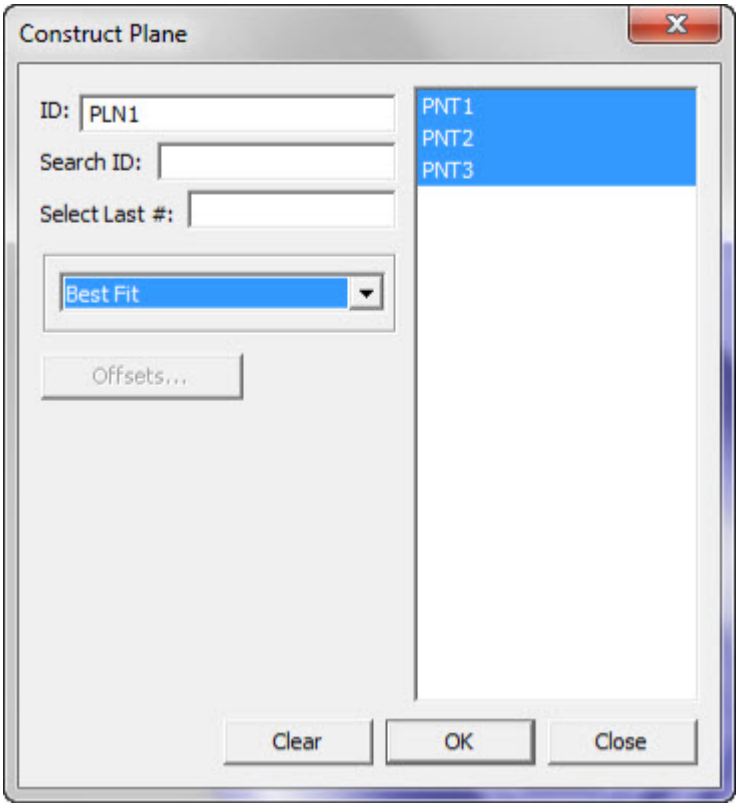
Editing a Constructed Feature

Editing an existing constructed feature is possible by right-clicking on any of the constructed features listed in the Plan Window and select the **Edit** option.



Constructed Feature menu

This will bring up the **Construct Feature** dialog for editing the selected constructed feature as shown below. Alternatively, pressing the keyboard **F9** key will also bring up the appropriate **Construct Feature** dialog.



Edit a Constructed Feature dialog

Available options:

- **Clear** - Click on the **Clear** button to reset all options then make any updates as needed.
- **OK** - Update the constructed feature as necessary then click on the **OK** button to save the changes.
- **Close** - Click on the **Close** button to exit this dialog without saving any changes made.

Adding Dimensions to Features

PC-DMIS Planner allows you create both basic and Feature Control Frame (FCF) dimensions for any feature in the Inspection Plan. The available dimension types are:

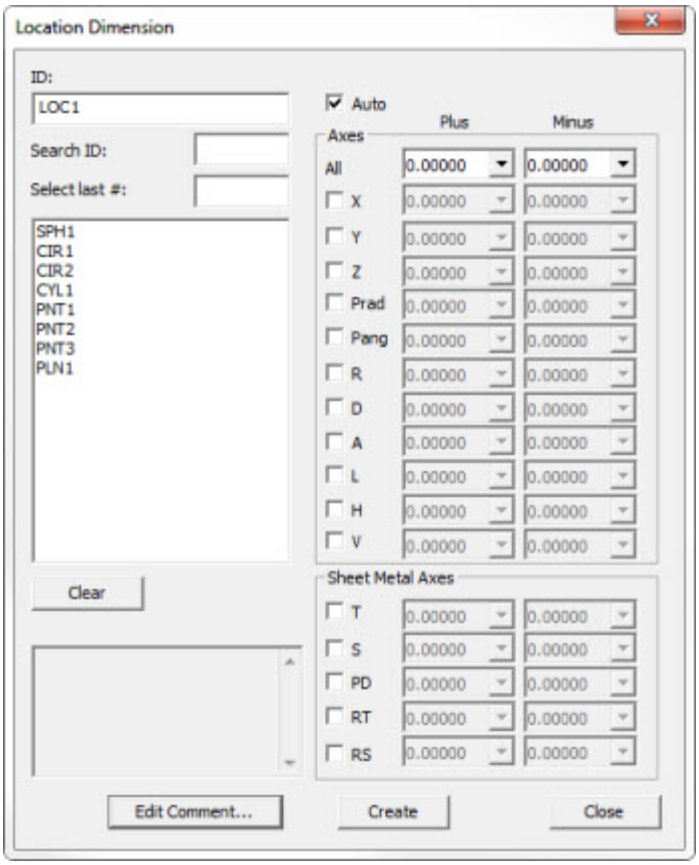
Icon	Description
	Datum
	Location
	True Position
	Distance
	Angle Between
	Concentricity
	Coaxiality
	Circularity
	Cylindricity
	Straightness
	Flatness
	Perpendicularity
	Parallelism
	Total Runout
	Circular Runout
	Surface Profile
	Line Profile
	Angularity
	Symmetry

The Location, Distance, and Angle dimensions do not use Feature Control Frames (FCF) to display their information. These are Basic dimensions. All other dimensions use FCFs.

Creating a Location Dimension

To create a Location Dimension:

1. Click Location (Plan) from the Dimensions (Plan) toolbar or select the Insert | Dimension (Plan) | Location menu item. This opens the Location Dimension dialog box.



Location Dimension dialog box

2. If needed, type an ID for this location dimension in the **ID** box.

3. Select the features for which location dimensions will be created by:

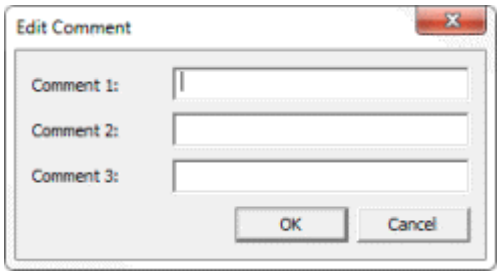
   - Selecting the features by clicking the feature name from the list.
   - Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.
   - Selecting the last of a certain number of features by typing the needed number in the **Select Last #** box. For example, if you wanted to select the last four features that were created, simply type 4 and press the TAB key. PC-DMIS will highlight the features selected in the **Feature** list box.
   - Click **Clear** to clear the previously selected features.

4. Select the appropriate **Axes** and **Sheet Metal Axes** for the selected feature(s). If you select **Auto**, when you import your inspection plan into PC-DMIS, PC-DMIS will automatically select the default **Axes** and **Sheet Metal Axes**. For example, if, inside PC-DMIS Planner, you selected a point feature from the list and chose **Auto**, when you import the inspection plan into PC-DMIS, PC-DMIS would automatically select the **X**, **Y**, **Z**, and **T** (vector) axes and apply the default tolerances as though **Axes** had been selected inside PC-DMIS.

5. Use the **Plus** and **Minus** tolerance fields to the right of each axis to provide the tolerance values for each axis selected.

6. If the same tolerances are wanted for more than one axis, select "ALL" from the **Axes** drop-down box and type the **Plus** and **Minus** tolerance values. These values are applied to all of the Axis types.

7. To add a comment to the location dimension commands, click **Edit Comment** to open the **Edit Comments** dialog box.

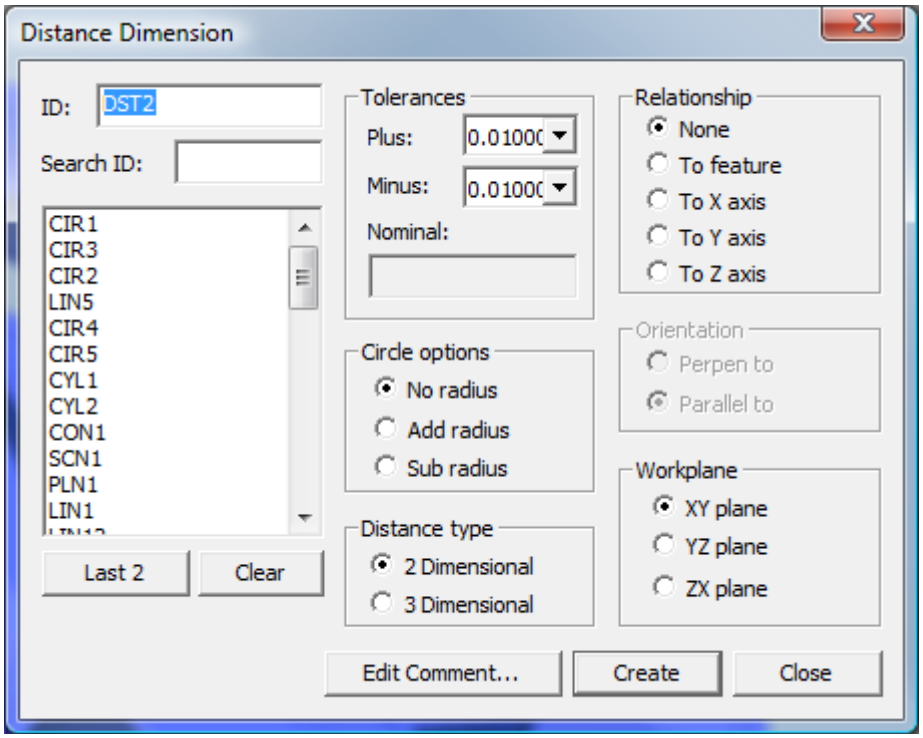


8. Click **Create** to add location dimension(s) to the inspection plan, or click **Close** to close the dialog box without saving.

Creating a Distance Dimension

To create a Distance Dimension between two features:

1. Select Distance (Plan) from the Dimensions (Plan) toolbar or select the Insert | Dimension (Plan) | Distance menu item. This opens the Distance Dimension dialog box.



Distance Dimension dialog box

2. If needed, type an ID for this distance dimension in the **ID** box.

3. Select 2 features for which a distance dimension will be created by:

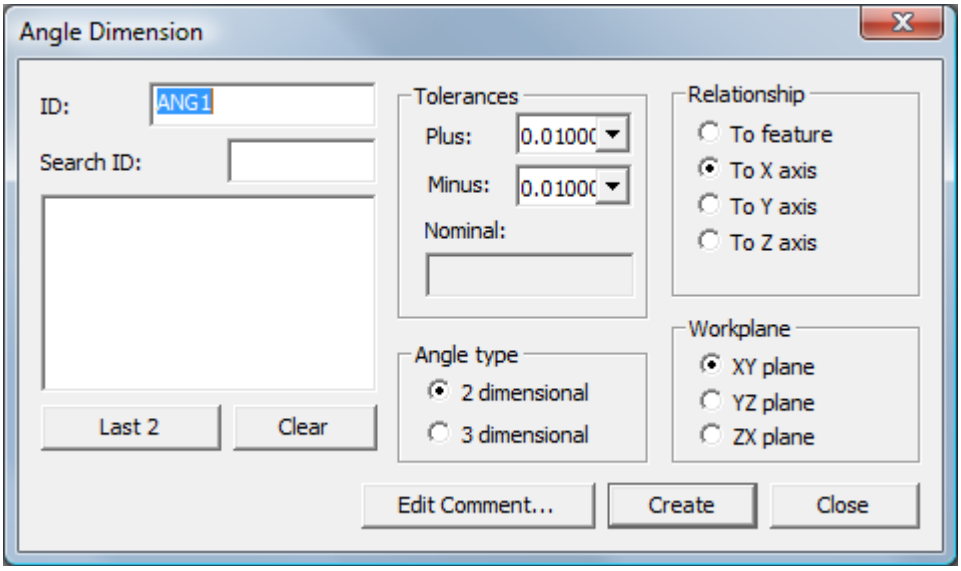
   - Selecting the 2 features by clicking the feature names from the list.
   - Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.
   - Click **Last 2** to select the last two features from the list.
   - Click **Clear** to clear the previously selected features.

4. Provide the **Plus** and **Minus Tolerance** values for the distance dimension(s). PC-DMIS calculates the **Nominal** distance between two features. The nominal value provides distance by which the tolerance values will be evaluated. You may also apply a previously used tolerance value by selecting it from the drop-down list. The last 10 tolerance values are stored.
5. Select the **Circle Options** that will be used when measuring from circular features. The default option (**No Radius**) will measure from the centroid of the feature. **Add Radius** will add the radius distance to the centroid measured distance. **Sub Radius** will subtract the radius distance from the centroid measured distance.
6. Specify the **Distance Type**. The **2 Dimensional** option will measure in a two-dimensional plane relative to the specified feature vector or XYZ axis. For **2 Dimensional** options you must also specify the **Orientation** and **Relationship** options. **3 Dimensional** option will measure point to point.
7. Choose the **Relationship** option for a **2 Dimensional** distance type to which the distance will be relative. The **To Feature** option uses the feature vector for the relationship.
8. Select an **Orientation** option for a **2 Dimensional** distance where a **Relationship** has been selected. You can measure the distance **Perpen** (Perpendicular) **To** or **Parallel To** the feature or axis specified by **Relationship** option.
9. To add a comment to the distance dimension commands, click **Edit Comment** to open the **Edit Comments** dialog box.
10. Click **Create** to add the distance dimension to the Inspection Plan, or click **Close** to close the dialog box without saving.

Creating an Angle Dimension

To create an Angle Dimension between two features:

1. Select **Angle (Plan)** from the **Dimensions (Plan)** toolbar or select the **Insert | Dimension (Plan) | Angle** menu item. This opens the **Angle Dimension** dialog box.



Angle Dimension dialog box

2. If needed, type an ID for this angle dimension in the **ID** box.

3. Select 2 features for which angle dimensions will be created by:
   - Selecting the 2 features by clicking the feature names from the list.
   - Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.
   - Click **Last 2** to select the last two features from the list.
   - Click **Clear** to clear the previously selected features.
4. Provide the **Plus** and **Minus Tolerance** values for the angle dimension(s). Planner calculates the **Nominal** angle between two features. The nominal value provides distance by which the tolerance values will be evaluated. You may also apply a previously used tolerance value by selecting it from the drop-down list. The last 10 tolerance values are stored.
5. Choose the **Relationship** option for a **2 Dimensional** angle type to which the angle will be relative. The **To Feature** option uses the feature vector for the relationship.
6. Specify the **Distance Type**. The **2 Dimensional** option will measure in a two-dimensional plane relative to the specified feature vector or XYZ axis. For **2 Dimensional** options you must also specify the **Orientation** and **Relationship** options. **3 Dimensional** option will measure the best fit angle created between the two features.
7. To add a comment to the angle dimension commands, click **Edit Comment** to open the **Edit Comments** dialog box.
8. Click **Create** to add the angle dimension to the Inspection Plan, or click **Close** to close the dialog box without saving.

Creating Feature Control Frame Dimensions

A Feature Control Frame (FCF) is a rectangular graphic that represents specified dimension information for one or more features. It usually appears on a blueprint or cad file to define tolerance specifications for specific dimension types. The ASME Y14.5 – 1994 Geometric Dimensioning and Tolerancing describes appropriate use of Feature Control Frames (FCFs).

These FCF dimensions include:

- **Form Dimensions:** Straightness, Flatness, Circularity (Roundness), Cylindricity
- **Orientation Dimensions:** Parallelism, Perpendicularity, Angularity
- **Location Dimensions:** Position, Concentricity, Coaxiality, Symmetry
- **Profile Dimensions:** Surface Profile, Line Profile
- **Runout Dimensions:** Circular Runout, Total Runout

To Create a Feature Control Frame Dimension:

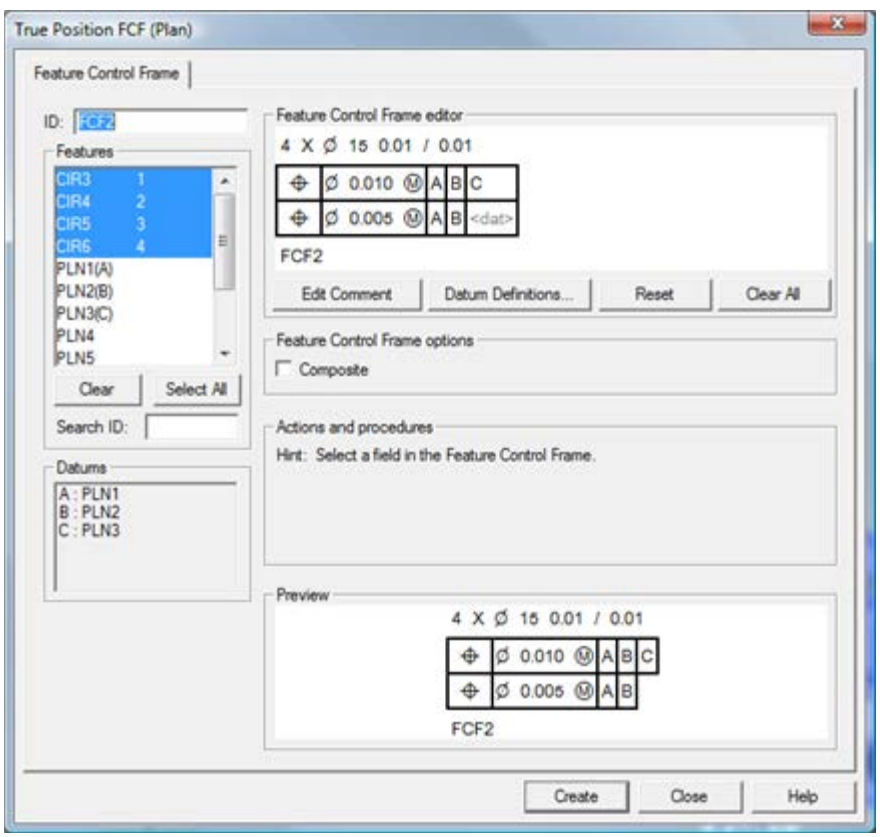
1. Click the desired icon from the **Dimensions (Plan)** toolbar, or select the menu item from one of the **Insert | Dimension (Plan)** submenus. The **FCF (Plan)** dialog for that dimension appears.
2. Use the **Feature Control Frame** tab to select the features to dimension and to construct your FCF. See "FCF (Plan) Dialog Box - Feature Control Frame Tab" for information on the available options in this tab.
3. Click **Create**. PC-DMIS inserts a FCF command into your inspection plan.

Feature Control Frame Creation Conditions

In order for Planner to properly create a FCF dimension, it must meet the appropriate conditions for the type of dimension you want to create. Consult the following table as needed:

Dimension Type	Conditions to Meet
Form dimensions:	No datums. No material conditions. No projected zones.
Profile dimensions:	May or may not have datums. No material conditions. No projected zones.
Orientation dimensions:	Must have datums. May have material conditions. May have projected zones.
Runout dimensions:	Must have datums. No material conditions. No projected zones.
Position dimensions:	May have datums. May have material conditions. May have projected zones.
Other location dimensions (these include symmetry concentricity, coaxiality):	Must have datums. No material conditions. No projected zones.

FCF (Plan) Dialog Box - Feature Control Frame Tab



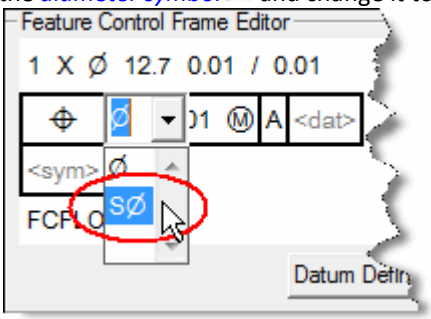
FCF (Plan) - Feature Control Frame tab

The **Feature Control Frame** tab helps you construct a Feature Control Frame (FCF). It provides you with the tools to define datum features, select the features used in the FCF dimension, an editor to define the specific symbols, tolerances, and datums used in your FCF, and a preview area to see the current state of the FCF as you build it. For information on the various items in this tab, see the list of controls below:

ID - This box displays the name for the FCF. You can edit it if you want something different.
Features - This list displays available features for the particular FCF type. Some features may exist in the part program, but are not available to the feature control frame. For example, a plane feature is not available to Circularity. Planner updates this list after you select the first feature. This ensures that when you create a pattern of features for the FCF, those features are compatible features.
Datums - The Datums shows all the datums in the Plan. It lists datum features above the current cursor position in the Plan window. It also lists datums it will create to satisfy this FCF.
Leader Lines - This list shows the same features that you selected from the **Features** list. Each feature has a check box associated with it. When you select a check box, Planner draws a leader line in the Graphic Display window from the FCF to that feature. By default, Planner initially displays all possible leader lines, but you can clear check boxes to turn off leader lines as well.
Feature Control Frame Editor - This area lets you apply changes to the FCF. You can select fields with the mouse or by pressing TAB and then ENTER to place the field into edit mode. If the field is editable, it will either display a drop-down list of available options, or it will display a box into which you can type text. When finished editing a field, press ENTER, TAB, or click on a different field to exit the edit mode. Pressing SHIFT + TAB will move backwards to the previous field.
When a field is empty a short description is displayed with brackets around it for identification. These descriptions correspond to these fields:

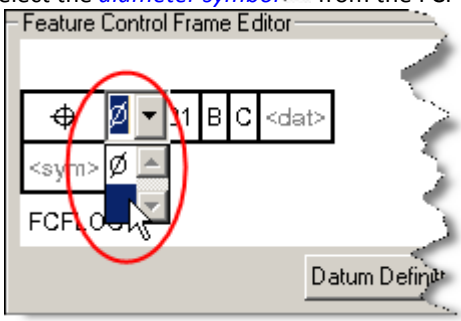
<MC> - Material condition
<D> - Diameter
<Dim> - Dimension or feature control frame type
<PZ> - Projected zone
<num> - Number of features
<nom> - Nominal of feature size
<+tol> - Plus tolerance
<-tol> - Minus tolerance
<tol> - Tolerance
<dat> - Datum
<sym> - Dimension symbol
<add notes here> - Note field in the first line
<add optional design notes> - Optional design notes in the last line

Spherical Zone - You can use a spherical tolerance zone if you have a True Position FCF for a point or sphere feature. To enable a spherical tolerance zone, select the feature to dimension, then in the FCF editor, select the *diameter symbol* Ø and change it to the spherical zone symbol as *shown here*:

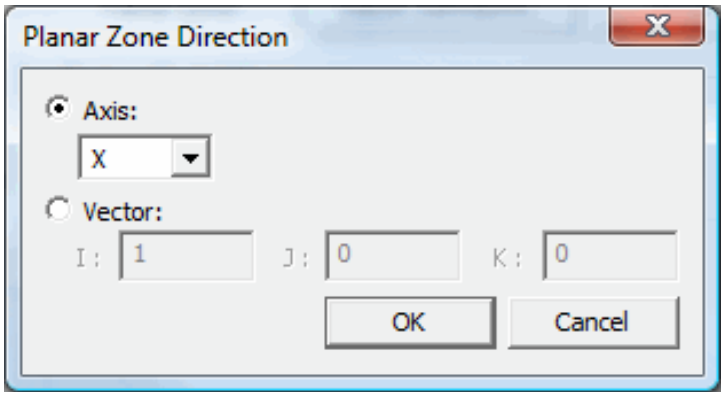


Planar Zone - This button appears in the **Feature Control Frame Editor** area of this tab if you are defining a True Position FCF dimension or one of the orientation FCF dimensions (Perpendicularity, Parallelism, or

Angularity) and the tolerance zone is set to a planar zone. To enable a planar zone and display this button, select the *diameter symbol* Ø from the FCF editor and set it to blank as *shown here*:



Once the **Planar Zone** button appears, you can click it to open the **Planar Zone Direction** *dialog box*.



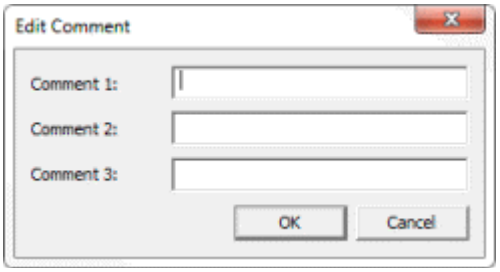
Planar Zone Direction dialog box

The **Planar Zone Direction** dialog box lets you specify the direction vector for the planar zone in one of two ways:

- Choose an axis in the current active alignment. Select the **Axis** option. Then from the **Axis** list, select an axis from the list: **X**, **Y**, or **Z**.
- Input the IJK of the direction vector directly. Select the **Vector** option. Then in the **I**, **J**, and **K** boxes, type the direction vector.

This direction vector will be given in the current active alignment.

Edit Comment - Click this button to display the *Edit Comment* dialog to insert up to three comments to the Feature Control Frame command that will be displayed in the Edit Window.



Edit Comment dialog box

Reset - Click this button to reset all numerical values in the **Feature Control Frame Editor** display to their default values.

Clear All - Click this button to clear the contents of the **Feature Control Frame Editor** display and returns it to the default view.

Datum Definitions - This button provides you with easy access to the **Datum Definitions** dialog box, allowing you to define datums for your current FCF dimension.

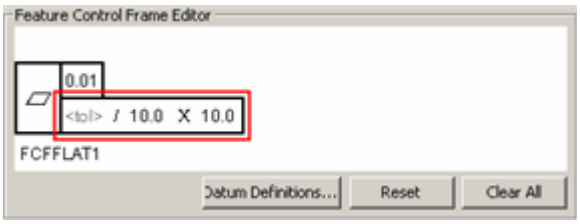
Feature Control Frame Options - The options in this area change based on the type of FCF dimension you are creating.

Per Unit check box

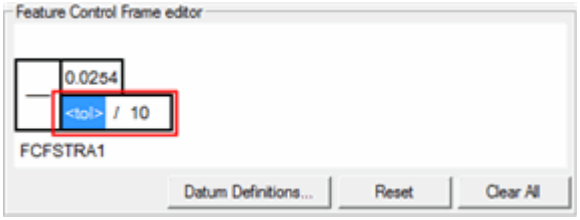
The **Per Unit** check box appears for flatness and straightness dimensions. It lets you specify a flatness per unit area or a straightness per unit length:

Per Unit check box

When you select the **Per Unit** check box, PC-DMIS changes the **Feature Control Frame Editor** area, adding an additional line where you can enter a tolerance value as well as the unit area or unit length values:

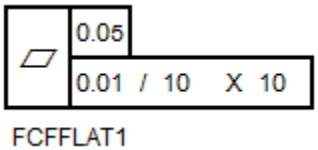


Feature Control Frame Editor (per unit flatness)



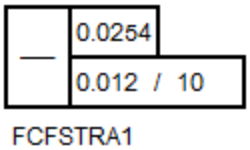
Feature Control Frame Editor (per unit straightness)

PC-DMIS now automatically uses a step increment that ensures that the "per unit window" will include maximum overlap as it "slides" over all the point data.
Once you enter in the desired information and create the Feature Control Frame, PC-DMIS also applies the flatness per unit area or straightness per unit length in the Feature Control Frame:
For example, a flatness FCF with an overall flatness tolerance of .05 and a per unit tolerance of .01 per unit area of 10 x 10 would look like this:



Example of a Finished Per Unit Flatness FCF

For example, a straightness FCF with an overall straightness tolerance of .0254 with a per unit tolerance of .012 / per unit length of 10 would look like this:



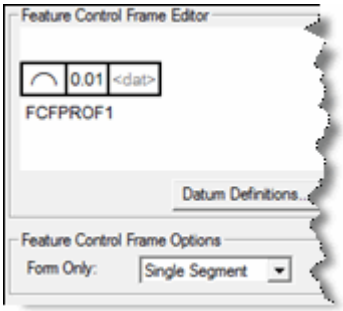
Example of a Finished Per Unit Straightness FCF

Form Only / Form and Location options

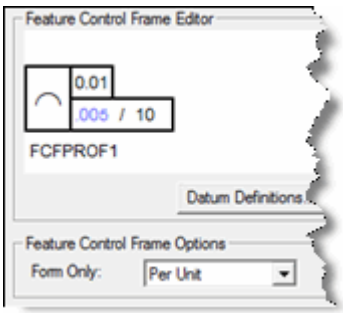
Form Only Profile: A **Form Only** list appears for profile FCF dimensions if you do not specify a datum for the dimension. Whenever this list appears it signifies that the dimension is a "form only" profile dimension. A form only profile dimension determines how close the profile dimension matches the same form as its nominal. Form

only ignores whether or not the location of the profile is outside of acceptable tolerances. For a Form Only dimension, pay attention to the measured values in the generated dimension.
PC-DMIS 2009 and later support the Form Only Profile of a Line when the considered feature is a constructed curve based on a linear scan. The theoretical points are projected onto the theoretical curve to determine what theoretical points and vectors to use in the profile calculation. Curves can also be used in the FCF per unit length profile of a Line.
The **Form Only** list contains these two items:

- **Single Segment** - This is a single segment overall "form only" profile (as compared to a composite or two segment profile).

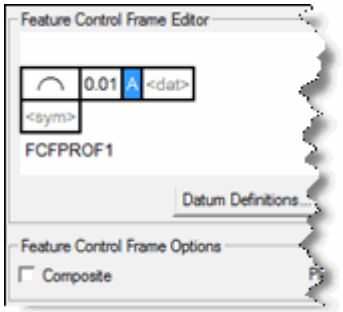


- **Per Unit** - This behaves the same as the **Per Unit** check box for flatness except that for straightness and line profile, the "Per Unit" distance is a linear distance along a line (for straightness) or along a line, scan, curve, circle, or ellipse (for line profile). This item only applies to line profile, not surface profile.

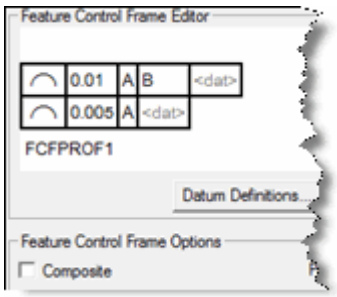


Form and Location Line Profile: If you do choose a datum then a "form and location" profile is implied, and the **Form Only** list is replaced by the **Composite** check box, allowing you to determine whether or not the dimension is a composite line profile dimension. See "Composite check box" below. Form and location dimensions let you determine if a particular profile matches the same form and the same location as its nominal. This differs from the **Form Only** option, because the location of the profile must also be within acceptable tolerances defined by the **Plus** and **Minus** tolerances. For a form and location dimension, pay attention to Max / Min values in the generated dimension.

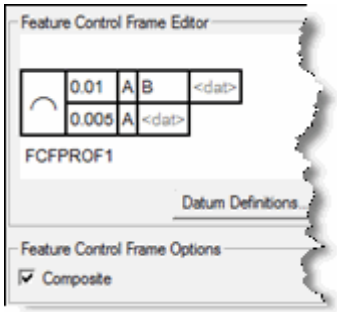
- A single segment "form and location" line profile:



- A double segment "form and location" line profile:



- A composite "form and location" line profile:



Composite check box

The **Composite** check box controls whether or not the primary dimension row and secondary dimension row should use the same symbol in a single merged cell that spans both rows. When you select this check box, the symbol displays like this:

4 X Ø 0.375 0.01 / 0.01

⌖	Ø 0.01	Ⓜ	A	B	Ⓜ	C
	Ø 0.005	Ⓜ	A	B	Ⓜ	

FCF6

If you clear the check box, the primary and secondary rows display with separate symbols, even if the symbol represents the same thing.

4 X Ø 0.375 0.01 / 0.01

⌖	Ø 0.01	Ⓜ	A	B	Ⓜ	C
⌖	Ø 0.005	Ⓜ	A	B	Ⓜ	

FCF6

The ASME Y14.5 – 1994 Geometric Dimensioning and Tolerancing describes the differences in interpreting these conditions.

Actions and Procedures - This section displays hints and instructions to assist you in building valid FCFs.

Preview - This section displays a preview of the FCF with the current settings. It will not include any of the empty fields or empty descriptions that appear in the **Feature Control Frame Editor** section, for example fields that have brackets, such as
"<dat>".

Notes on the Feature Control Frame Editor

In the **Feature Control Frame Editor** area be aware that some of the icon fields, if they are not supported, will not appear for certain dimensions. For example, if you have a Circularity dimension, Planner displays the tolerance field, but hides any datum or modifier fields.

Form dimensions (Circularity or Roundness, Cylindricity, Flatness, Straightness, Profile with Form Only) - These don't support datums or modifiers, so these fields will not appear. Profile in some cases will fall in this category when checking for "Form Only". But "Form and Location" for Profile dimensions allows the display of modifiers.
Orientation dimensions (Parallelism, Perpendicularity, Angularity) - These allow datums and modifiers.
Location dimensions (True Position, Concentricity, Coaxiality, Symmetry, Profile with Form and Location) - These allow datums and modifiers.
Runout dimensions - These must have datums.

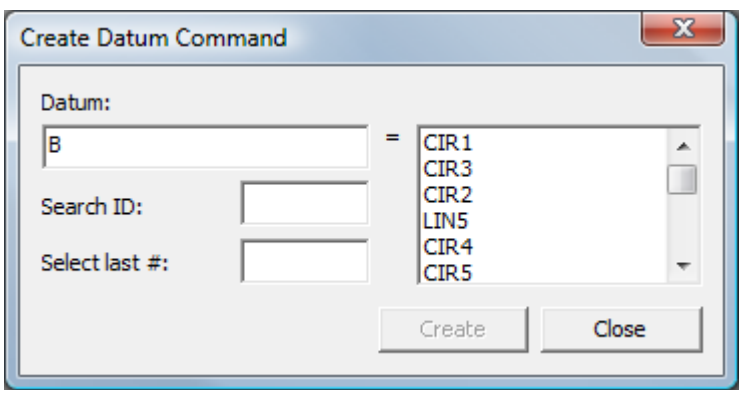
Also, only these dimensions can have multiple lines in the editor:

- True Position
- Parallelism
- Perpendicularity

Creating Datums

A datum is a theoretically perfect plane, point, or axis reference feature. A dimensional measurement is made from a datum. Datums are used by Feature Control Frames (FCFs) when displaying GD&T information. In order to define the datums used in FCFs, you should first create the features in your plan that will become the datums. These features can come from auto features or constructed features.

1. First, create the features that will be datum features. These can be auto features or constructed features.
2. Click the **Datum (Plan)** icon from the **Dimensions (Plan)** toolbar. The **Create Datum Command** dialog box appears.



Create Datum Command dialog box

3. Type the datum's ID in the **Datum** box. Planner will default to the next available datum letter.
4. Select the feature to use as the datum from the list of features. Planner highlights the selected feature in the Graphic Display window.

Note: When selecting features use the following dialog box options:

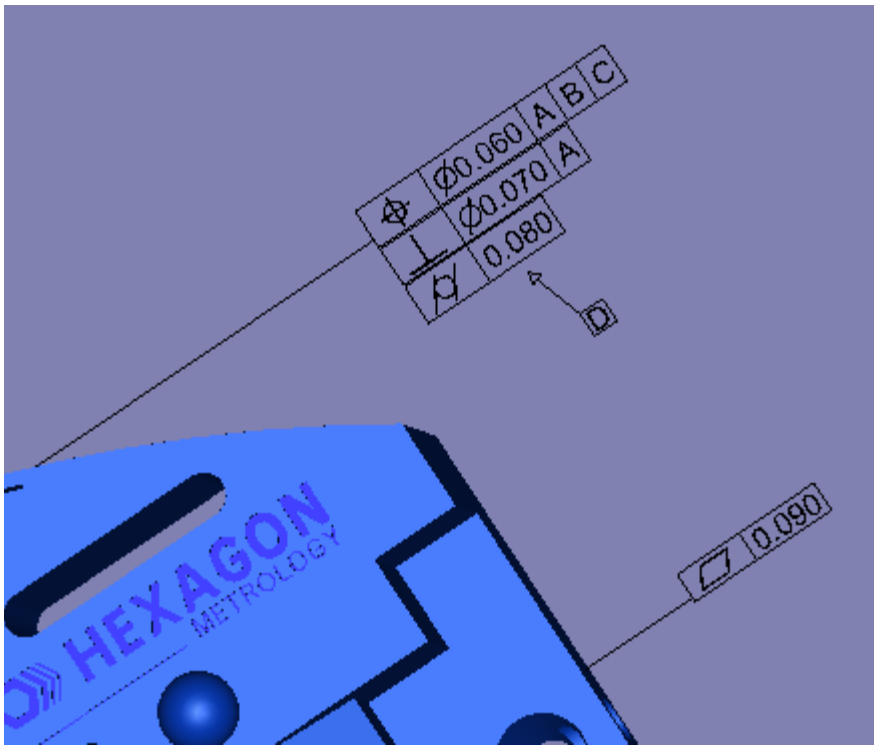
● Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.

● Selecting the last of a certain number of features by typing the needed number in the **Select Last #** box. For example, if you wanted to select the last four features that were created, simply type 4 and press the TAB key. PC-DMIS will highlight the features selected in the **Feature** list box.

● Click **Clear** to clear the previously selected features.

5. Click **Create**. Planner inserts the datum into your inspection plan.

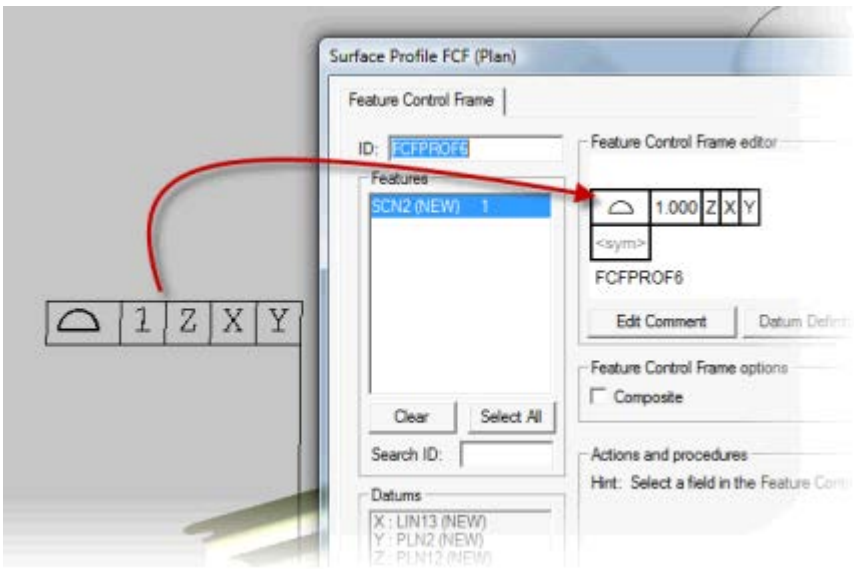
Adding Features from GD&T Callouts



CAD imported with GD&T Callouts

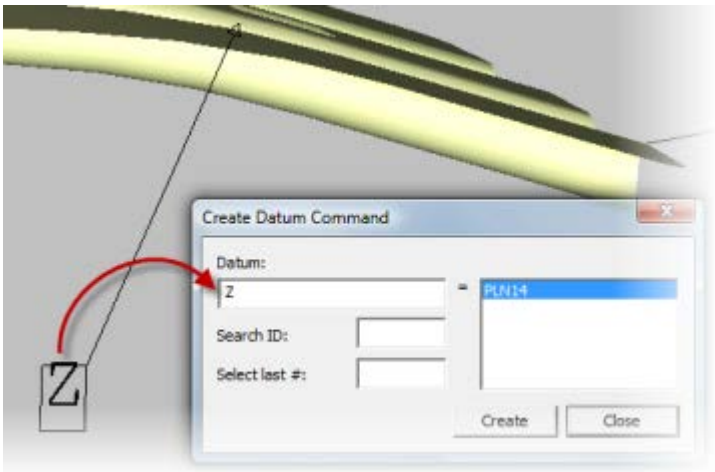
If your CAD data was imported with GD&T callouts, you can quickly add features to your Inspection Plan from these callouts by doing the following:

1. Click on the callout in the **Graphic Display** window.
   - If the callout is a Feature Control Frame, this opens a **FCF (Plan)** dialog box for that FCF callout. Planner automatically creates the Feature Control Frame inside the **Feature Control Frame editor**.



Sample Showing a GD&T FCF Callout created as a FCF

- If the callout is a datum, this opens the **Create Datum Command** dialog box with the datum inside.



Sample Showing a GD&T Datum Callout created as a Datum

When you click on a datum callout in the CAD window, Planner checks to see if there are any datum targets associated with the datum. If there are, it uses the datum targets to create the datum feature.

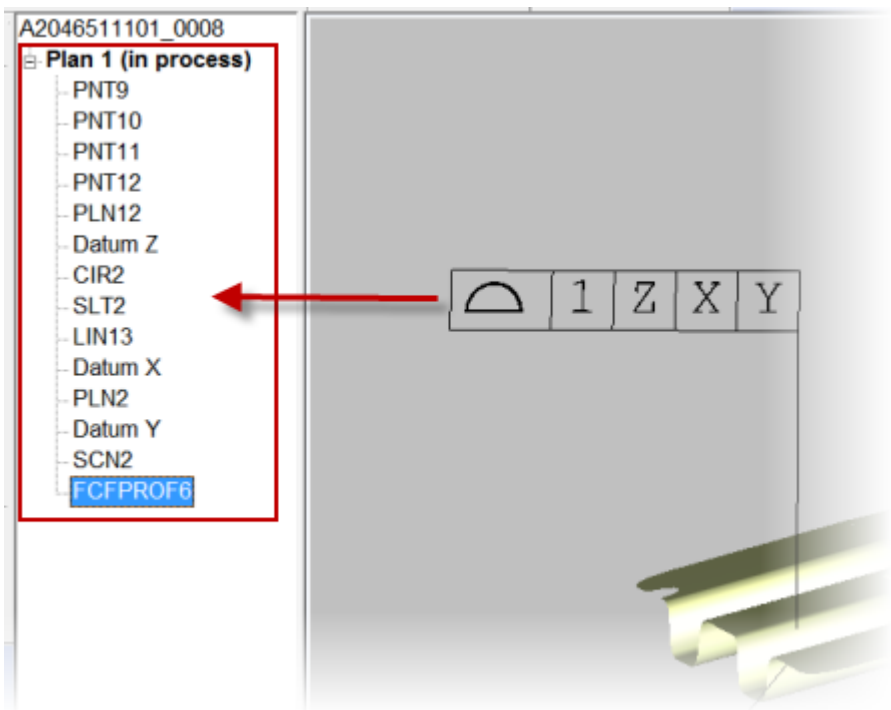
- If there is just one target, it uses the feature from that target for the datum.
- If there are two targets, it creates a constructed offset line.
- If there are three targets, it creates a constructed offset plane. If a constructed plane is created, and the datum callout is attached to a plane, Planner calculates the offsets so that the offset plane is coplanar with the datum plane.

When you click on the callout to add a datum, Planner highlights the associated surface or feature in the Graphic Display window. If the feature doesn't exist, Planner adds the feature into the **Create Datum Command** dialog box. This feature will get created and added to the plan when you click **Create** at the end of this procedure.

2. Review the features, Feature Control Frame, and any datums commands that will be added. While Planner attempts to create the appropriate feature or features, it may not always succeed. When this happens, you will need to manually create any missing features.
3. Make any adjustments as needed in either the **FCF** dialog box or the **Create Datum Command** dialog box.
4. Click **OK** or **Create**. All associated commands to generate that Feature Control Frame or datum are added to the Inspection Plan
5. Repeat steps 1 through 3 for all the needed callouts.

Note: You cannot select GD&T callouts while in "Text Box Mode" since this mode allows you to reposition GD&T labels in the Graphic Display window.

Below is an example of the Graphic Display window and the commands added to the Inspection Plan.

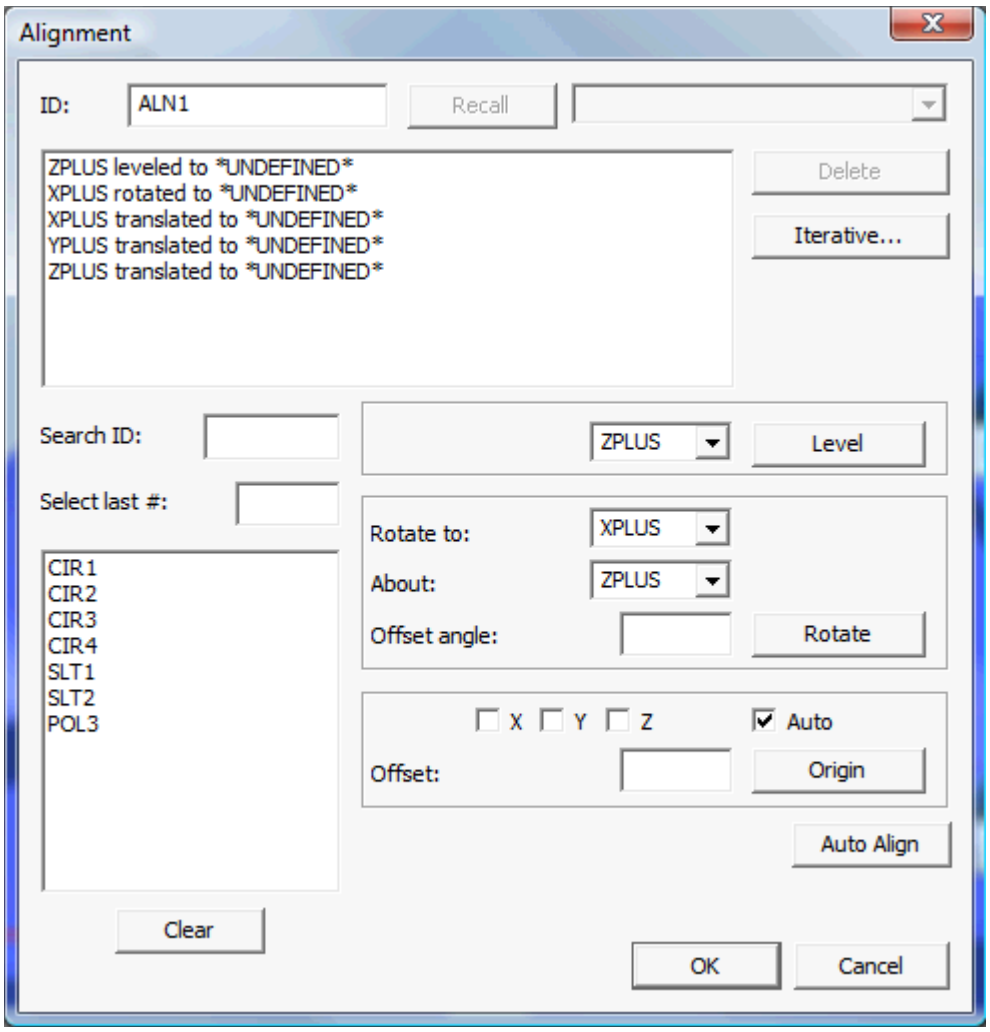


Plan Created from a GD&T Callout

Creating Alignments

Once you have created features, you can create or modify alignments. The procedure below explains how to create a standard 3-2-1 alignment. See the "Creating an Iterative Alignment" topic for Iterative Alignment information.

1. Click **Alignment** from the **Plans** toolbar or select the **Insert | Alignment** menu item. This opens the **Alignment** dialog box.



Alignment dialog box

2. If needed, type an ID for the alignment in the **ID** box.
3. Select the feature and the axis (from the drop-down box next to **Level**) to which the alignment will be leveled. Click **Level** to define the leveling component to the **Active Alignment** list.
4. Select the feature, **Rotate To** axis and **About** axis (from the drop-down boxes next to **Rotate**) to which the alignment will be rotated about. Use the **Offset Angle** value to adjust the rotational offset angle from the nominal rotation to the specified feature. Click **Rotate** to define the rotating component to the **Active Alignment** list.
5. Select the feature to which the selected origin axis/axes of the alignment will be established. Choose one or more **XYZ** axes or the **Auto** option. Click **Origin** to define the translated component(s) to the **Active Alignment** list. The **Auto** will choose the axes to move based on the feature type, the orientation of that feature, and the workplane.

Note: When selecting features use the following dialog box options:

- Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.

● Selecting the last of a certain number of features by typing the needed number in the **Select Last #** box. For example, if you wanted to select the last four features that were created, simply type 4 and press the TAB key. PC-DMIS will highlight the features selected in the **Feature** list box.

● Click **Clear** to clear the previously selected features.

6. Click **OK** to add the alignment to the Inspection Plan, or click **Cancel** to close the dialog box without saving.

Using Auto Align

If desired, you can instead select the above three features and click the **Auto Align** button to have Planner automatically level to the first feature, rotate about the second, and translate to the third. The order in which the three features are selected is *extremely* important.

The first of the three features will be used to establish the orientation of the normal axis to the current working plane.
The current working plane will then be rotated to the second feature.
Lastly, the part origin will be set according to the third feature type.

For example, if you select a:

Plane, Line and a Point with a ZPLUS workplane:
The plane will establish the ZPLUS orientation of the part. The measured line becomes the XPLUS major axis. The origin is set with the Z zero equal to that of the plane. The Y is equal to that of the line and the X is equal to that of the point.
Plane, Line and a Measured Circle with a YMINUS workplane:
The plane will establish the YMINUS orientation of the part. The measured line becomes the ZMINUS major axis. The origin is set with the Y zero equal to that of the plane. The X and Z zeros (origins) are equal to the center of the circle.

If Planner is unable to create a new alignment based on the three selected features, a pop-window will appear with an error message.

Important: In an Auto Alignment the origin of the alignment is based on the third feature used.

● If it's a point, Planner sets the axes origin. If not, Planner checks the feature's default axes.

● If it's a line, Planner returns X, Y, and Z and sets the origin on the X, Y, and Z.

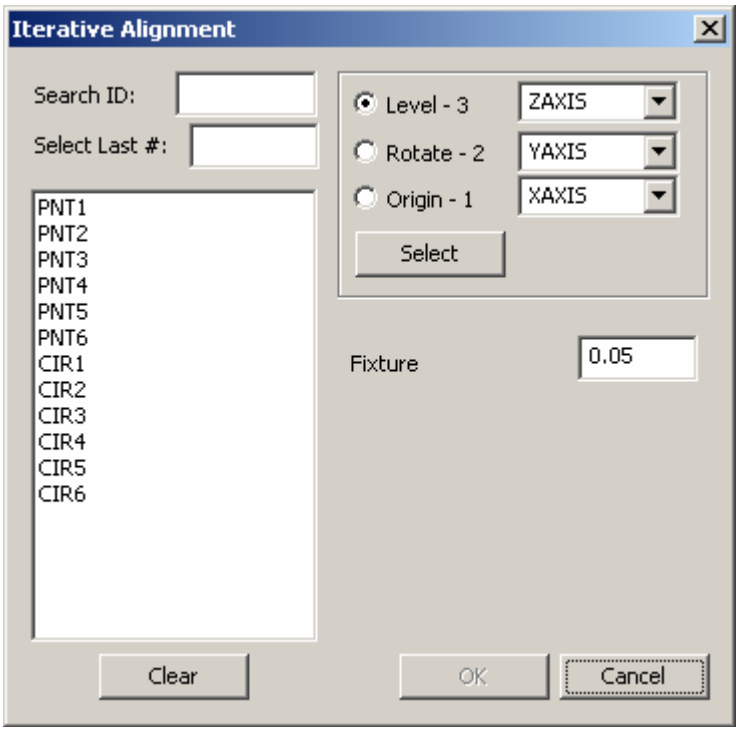
● If it's a Circle, Planner sets all three axes.

Creating an Iterative Alignment

Aside from standard alignments, capability exists to create Iterative (3-2-1)Alignments using PC-DMIS Planner.

To Create an Iterative Alignment:

1. Click **Alignment** from the **Tools** toolbar or select the **Tools | Alignment** menu item. This opens the **Alignment** dialog box.
2. If needed, type an ID for the alignment in the **ID** box.
3. Click **Iterative** to open the **Iterative Alignment** dialog box.



Iterative Alignment dialog box

4. Select the proper axis from the **Level - 3** combo box.
5. Select the three (or more) features from the feature list on the left for the level.

Note: When selecting features use the following dialog box options:

● Typing the needed feature name in the **Search ID** box and the feature will be highlighted in the Feature list box when you press the TAB key.

● Selecting the last of a certain number of features by typing the needed number in the **Select Last #** box. For example, if you wanted to select the last four features that were created, simply type 4 and press the TAB key. PC-DMIS will highlight the features selected in the **Feature** list box.

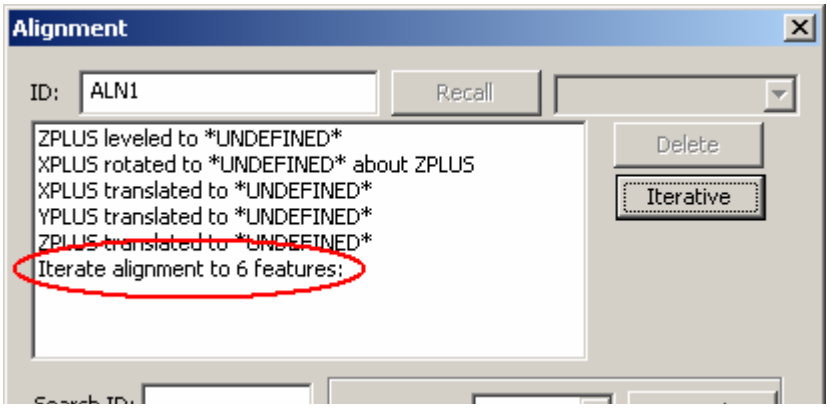
● Click **Clear** to clear the previously selected features.

6. Click **Select** or the **Rotate - 2** radio button.
7. Select the proper axis from the **Rotate - 2** combo box.
8. Select the two (or more) features from the feature list on the left for rotation.
9. Click **Select** or the **Origin - 1** radio button.
10. Select the proper axis from the **Origin - 1** combo box.
11. Select one feature from the feature list on the left for the origin.
12. Type in the proper value for the **Fixture** tolerance. This value allows you to provide a fitting tolerance value against which PC-DMIS compares feature elements making up the iterative alignment to their theoretical values.

Note: If after fitting the measurement values to the theoretical values, one or more of the input features have an error along their assigned datum axis that exceeds this tolerance value, PC-DMIS displays an error message. The message shows the errors along each of the datums. You will have the choice to accept the datum as it is and continue with the rest of the part program, or to cancel the part program execution.

PC-DMIS can only use the **Fixture** tolerance value if you used more than the minimum number of points needed to create the feature. For example, if you are measuring a plane, the minimum number of points needed for that plane is usually three points. However, if you wanted to use the fixture tolerance value, you would need to measure at least four points. If you use only three points then only one solution exists and PC-DMIS cannot adjust or re-iterate.

13. Click **OK**. The iterative alignment will get added to the list of alignments in the main **Alignment** dialog box:



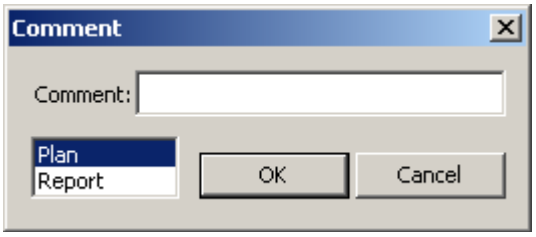
Iterative Alignment added to list of alignments

14. Click **OK** from the main **Alignment** dialog box will add this alignment to the Inspection Plan.

Inserting Comments

Comments come in two types: Plan or Report. This allows you to insert a specific type of comment into your Inspection Plan, providing programmer notes or notes that will show up on your Inspection Plan reports.

1. Click **Comment** from the **Plans** toolbar or select the **Insert | Comment** menu item. This opens the **Comment** dialog box.

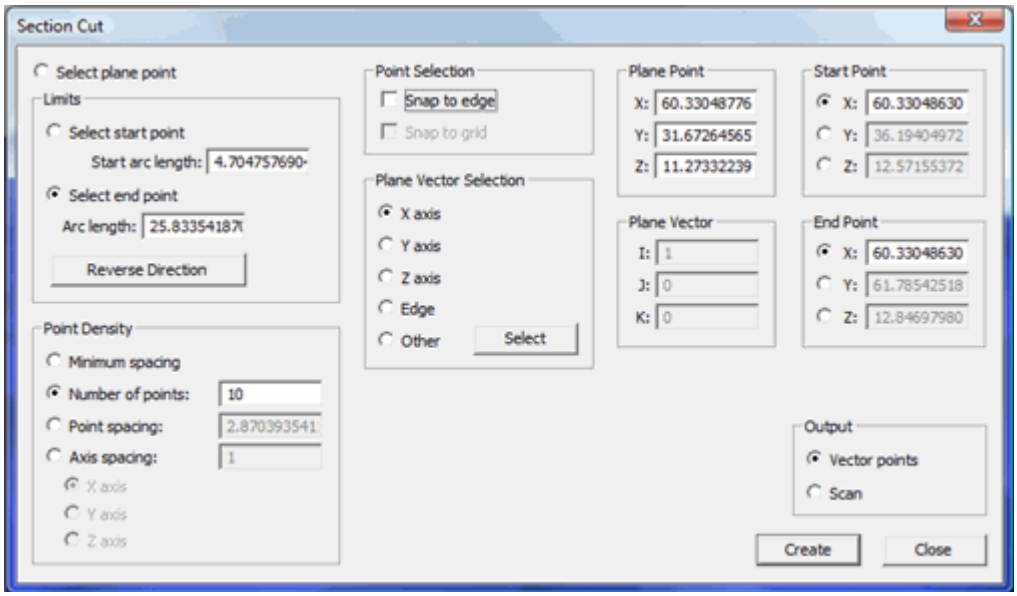


Comment dialog box

2. Type the needed comment in the **Comment** field.
3. Select either **Plan** or **Report** to determine when the Comment is displayed.
   - **Plan** comments are documentation comments in the plan.
   - **Report** comments are added to the report.
4. Click **OK** to add the comment to the Inspection Plan, or click **Cancel** to close the dialog box without saving.

Working with Section Cuts

The **Insert | Section Cut** menu item displays the **Section Cut** dialog box.

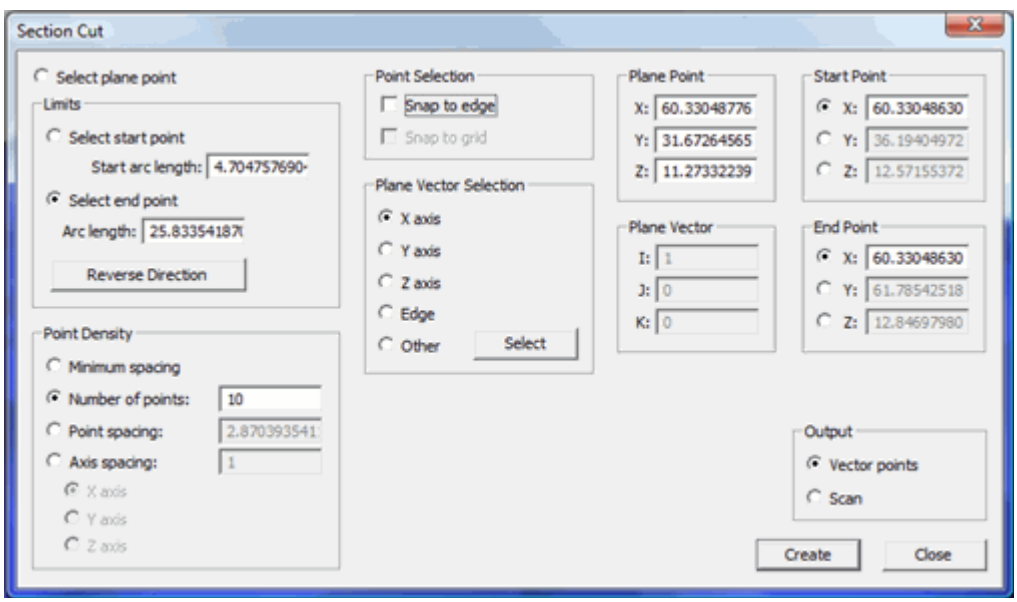


Section Cut dialog box

Use this dialog box to specify a cut plane that intersects with the CAD model. Along the intersection line, you can define a start and end point between which points are created. From these points you can choose to create vector point features or a Freeform scan.

Note: This process does not visually cut the CAD model in any way like the clipping plane functionality does; instead, it acts as a tool to help you create point features or a Freeform scan along the intersection line of the cut plane and the CAD model.

Description of the Section Cut Dialog Box

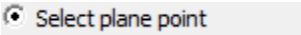


Section Cut dialog box

Note: For detailed information about creating a section cut, see "Creating a Section Cut".

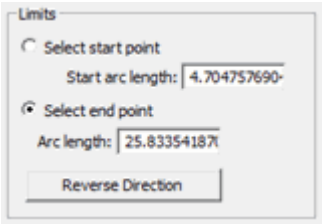
The following options are available:

Select plane point option



Select a point on the CAD model, which will become the cut plane point.

Limits area



Specify the start and end points along the intersection. You can either select the points in the Graphic Display window, or specify an arc length to precisely position the start and end point.

Select start point - Select the start point of the section cut by selecting the start point in the Graphic Display window. Select the point on the black intersection line. A red point will appear on the screen indicating the location of the start point.

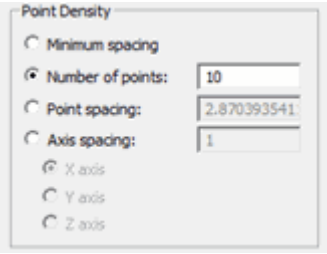
Start arc length - Use this box to precisely position the start point with respect to the cut plane point. Type the arc length between the projection of the cut plane point onto the section cut and the start point. Note that you can also define a negative number.

Select end point - Specify the end point on the section cut by selecting the end point in the Graphic Display window. Select the point on the black intersection line. A magenta point will appear on the screen indicating the location of the end point.

Arc length - Use this box to precisely position the end point. The value you type is the arc length between the start and end points. Note that you can also define a negative number.

Reverse Direction - Click this button to swap the direction that the arc lengths are measured from the plane point.

Point Density area



Use this area to control the point spacing and number of points computed between the start and end points.

Minimum Spacing - This option uses a minimum number of points based on the curvature of the surfaces along the section cut. If the surfaces are flat, then only two points will be created at the start and end points. If the surfaces are curved, then more points will be created. The number of points created on curved surfaces depends on the value set in the tessellation multiplier defined in the **OpenGL Options** dialog box. See the "OpenGL Tab" topic.

Number of points - Type the number of points you want created. Planner evenly distributes the points between the start and end points.

Point spacing - Specify the arc length between each point.

Axis spacing - This option limits the creation of the points along only the selected axis. Once you select this option the **X axis**, **Y axis**, and **Z axis** options become enabled. The box next to this option lets you define the spacing between points along that selected axis. For example, if you selected the X axis, then the points would be spaced along the X axis according to the value you specified.

Point Selection area

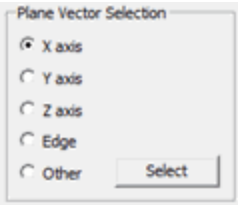
Use this area to specify the snap options for the plane, start, and end points.

Snap to edge - This check box determines whether or not Planner snaps the point to the nearest surface edge or surface boundary.

Snap to grid - This check box determines whether or not Planner snaps the point to the nearest grid intersection. You can use the snap to grid functionality even when the 3D grid is not showing. See the "Setting Up the View" topic to enable the 3D Grid.

If you select both **Snap to edge** and **Snap to grid**, Planner will snap the point to the nearest grid line that intersects a surface edge or boundary.

Plane Vector Selection area



Use this area to specify the cut plane normal vector.

X axis - Sets the cut plane normal to the X axis vector (1,0,0).

Y axis - Sets the cut plane normal to the Y axis vector (0,1,0).

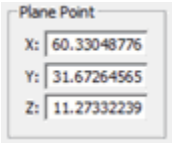
Z axis - Sets the cut plane normal to the Z axis vector (0,0,1).

Edge - Sets the cut plane normal to the nearest surface boundary tangent vector. Whenever you select the plane point, the plane normal will be updated to the nearest surface boundary tangent vector.

Other - Defines the cut plane normal values manually. Once selected, you can either type the IJK values in the **Plane Vector** area or you can click the **Select** button to select a feature on the CAD model to use as the normal vector.

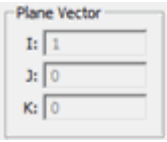
Select - Displays the **Select Points** dialog box, which you can use to select a feature to use as the cut plane normal vector. This dialog box is documented in the "Transforming a CAD Model" topic.

Plane Point area



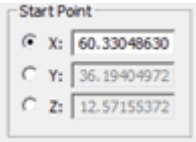
This area shows the XYZ values of the plane point. You can manually modify the values by typing new values in the **X**, **Y**, and **Z** boxes. Note that if the point you specify does not lie on a CAD surface, the actual point that is used will be projected onto the CAD model.

Plane Vector area



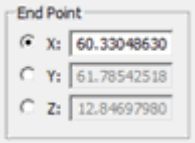
This area shows the IJK values of the plane normal vector. You can manually modify these values by typing new values in the **I**, **J**, and **K** boxes.

Start Point area



This area displays the XYZ values of the start point. You can also use this area to define or adjust the value of the selected axis. The other two axis values will be computed from the intersection line.

End Point area



This area displays the XYZ values of the end point. You can also use this area to define or adjust the value of the selected axis. The other two axis values will be computed from the intersection line.

Output area

Use this area to determine the type of feature or features created from the section cut. Planner creates the output feature or features only after you click the **Create** button.

Vector points - This option specifies that vector points should be created.

Scan - This option specifies that a Freeform scan should be created from the points.

Create button

Creates the specified feature or features from the section cut. The type of features depends on the option selected in the **Output** area.

Close button

Closes the **Section Cut** dialog box.

Creating a Section Cut

To create a section cut, you need to define these pieces of information:

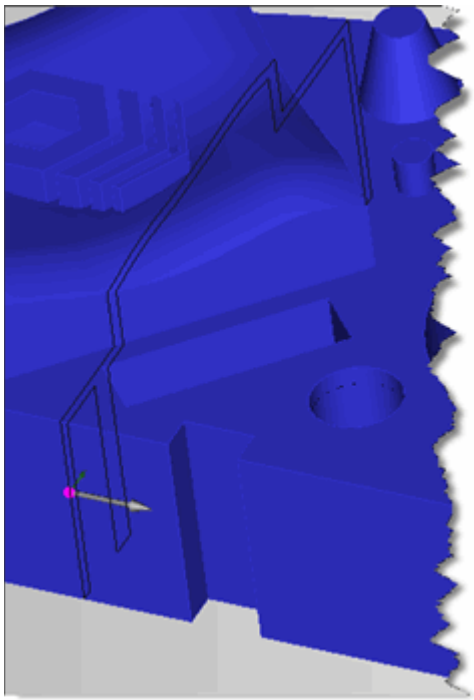
- A cut plane
- A start point on the section cut
- An end point on the section cut

Step 1: Define the Cut Plane

To define the cut plane, specify a point on the plane. You can do this in two ways:

- You can select the **Select plane point** option. Then click a point on the CAD model.
- You can manually type the XYZ values in the **Plane Point** area.

Once defined, Planner draws a gray arrow indicating the plane point and the direction of the cut plane normal. Additionally, Planner draws a polyline (or one or more connected lines) on the CAD model, representing the intersection of the plane (called the "cut plane") with the surfaces in the entire CAD model. Multiple section cuts are drawn as different colored polylines to show when very small surface gaps are present. Since you haven't defined the start and end points yet, the red and magenta dots, representing the start and end points respectively, will initially appear on the CAD model at the plane point's location:



A sample Plane Point (indicated by the gray arrow) and Cut Plane (indicated by the black lines) drawn atop the CAD model

Note: If the plane intersects the model in more than one location, Planner draws all the intersections.

Once you define the cut plane point, you can optionally specify the cut plane's normal vector. By default, the normal vector will be (1,0,0). You can modify this normal vector by selecting an option in the **Plane Vector Selection** area, thereby shifting the normal along one of the selected axes or you can define your own custom vector.

Step 2: Define the Start and End Points along the Section Cut

Now that you have the cut plane defined, you need to define a start and end point along the section cut. You can do this using any combination of these different methods, depending on your preference to define the start and end points:

Method 1: Clicking on the CAD

1. Choose the **Select start point** option and then click a point on one of the black lines making up the section cut. This defines the distance away from the **Plane Point** along the section cut and places that distance in the **Start arc length** box. Planner places the XZY values for the selected point into the **Start Point** area.
2. Choose the **Select end point** option and then click another point on the same section cut. This defines the length of the arc between the start and end point. Planner places the XYZ values for the selected point into the **End Point** area.

Method 2: Typing Arc Values

1. Define the start point by specifying the distance away from the **Plane Point** by typing the value in the **Start arc length** box.
2. Define the end point by specifying the arc's length. Do this by typing the value in the **Arc length** box.

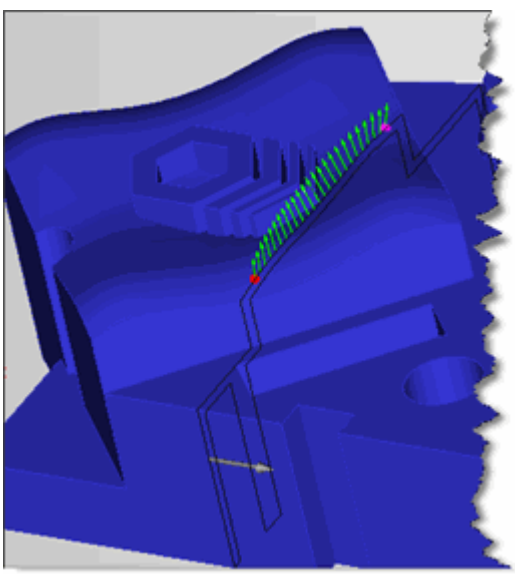
Method 3: Typing XYZ Values

Define the start and end points by typing the XYZ value in the **Start Point** and **End Point** areas.

Important: The start and end points must be on the same section cut. For example, if a gap between two surfaces breaks your section cut into multiple cuts, the start and end points must be defined on only one cut. If you try to select the start and end points across different section cuts, the first selected point will be removed and you will need to select it again.

A red dot appears on the CAD model representing the start point and a magenta dot appears representing the end point. Additionally, Planner draws green arrows along the section to show where the points of the section cut will be created. If the surface is curved, then several arrows will be drawnIf the surface is flat, then these green arrows will only be drawn at the start and end point (because the **Point Density** area has **Minimum Spacing** selected by default).

You can modify options in the **Point Density** area to control the number of points between the two points:



A Sample Section Cut Showing 25 Equally Spaced Points Between the Start Point (Red Dot) and the End Point (Magenta Dot)

Step 3: Define the Output and Create

1. Select the desired output format in the **Output** area. The output can either be in individual Auto Vector Points or a Free Form scan containing the points.
2. Modify any other controls as needed. These allow you to customize the parameters affecting the plane, start and end points, point spacing, and feature type created.
3. Click **Create** button to create the output features or scan.

Planner creates the specified feature or features in the part program.

Fixing the Direction of Normals Along the Section Cut

The green arrows represent the surface normal vectors at the points. The section cut algorithm is designed so that the surface normal vectors along the section cut will not flip as they transition across multiple surfaces. However, these vectors may all be pointing in the wrong direction (inside the part). If these arrows are pointing in the wrong direction, use the **Edit CAD Vectors** dialog to fix the surface normals.

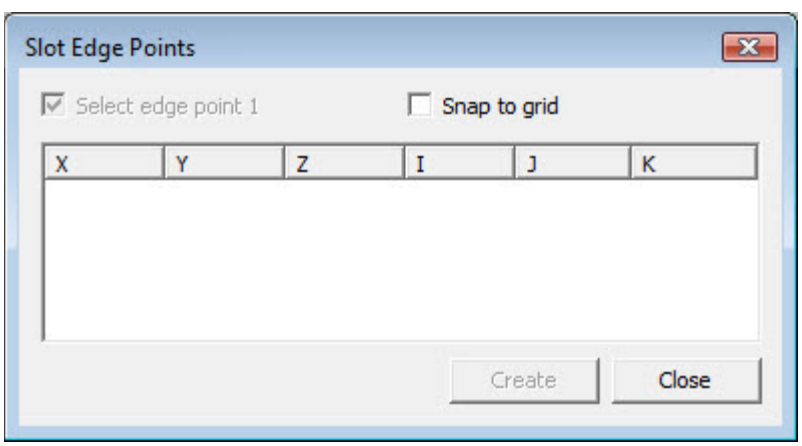
To do this:

1. Select Edit | CAD | CAD Vectors.
2. Clear the **Show vectors** check box (so drawing is faster) and then box-select the entire CAD model in the Graphic Display window.
3. Click the Fix surface vectors button.
4. Click the **Close** button.

If the part is a well defined manifold solid, then all the normals will point outside the part. Once you have fixed the surface vectors, create the section cut scan and the normals should now point in the same direction. See "Editing CAD Vectors".

Creating Slot Edge Points

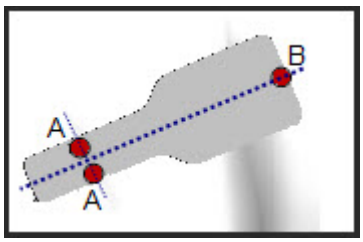
The Insert | Slot Edge Points menu item displays the Slot Edge Points dialog box.



Slot Edge Points dialog box

This dialog box lets you create edge points on section cuts that bisect supported features. Supported features include slots, circles, and ellipses.

This provides you with a way to create edge points along the slot at precise locations. A typical area where this method of point creation may come in useful includes creating three edge points along a "key-hole" shaped slot, like this:



Key-hole slot requiring two points (A) on the smaller side of the slot directly across from each other and one point (B) on the larger surface along the section cut between the other two points (A)

Dialog Box Items

The dialog box contains these items:

Select edge point 1 - This lets you define the first edge point. This check box is initially marked and deactivated when you first access the dialog box. After you click once on the CAD model, however, the check box clears and becomes active so that you can choose a different start location if needed.

Points list - The list contains the slot edge points as you click to define them in the Graphic Display window. The columns display each point's location (XYZ columns) and vector (IJK columns). You can use this list to delete existing points or flip their vectors. Use SHIFT and Ctrl keys while clicking with the mouse to select multiple points. Once you have selected the desired points, right-click on the selection and a small popup menu appears with these options:

Flip Vector - This flips the vector of all the selected points.

Delete - This deletes all of the selected points.

Snap to grid - This check box snaps the slot edge point to the closest 3D grid line. You can display the 3D grid using the **View Setup** dialog box. However, note that Planner still snaps the point to the closest grid line even if the grid is invisible.

Create - This button creates any slot edge points you selected from the CAD model and adds them to the inspection plan. It becomes activated the moment you click on the CAD to select your first edge point.

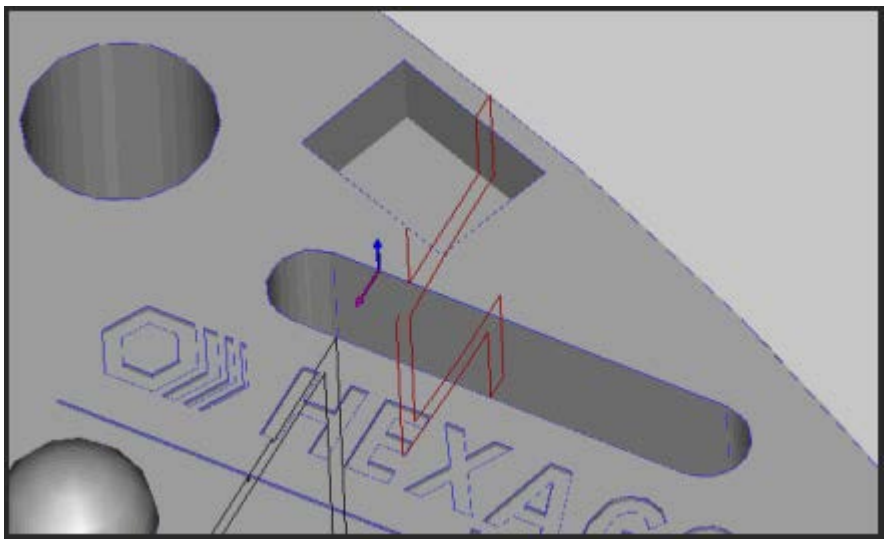
Close - Closes the dialog box without adding any point features into the inspection plan.

Key-Hole Slot Example

To create the slot edge points similar to those shown in the key-hole slot above, follow this procedure:

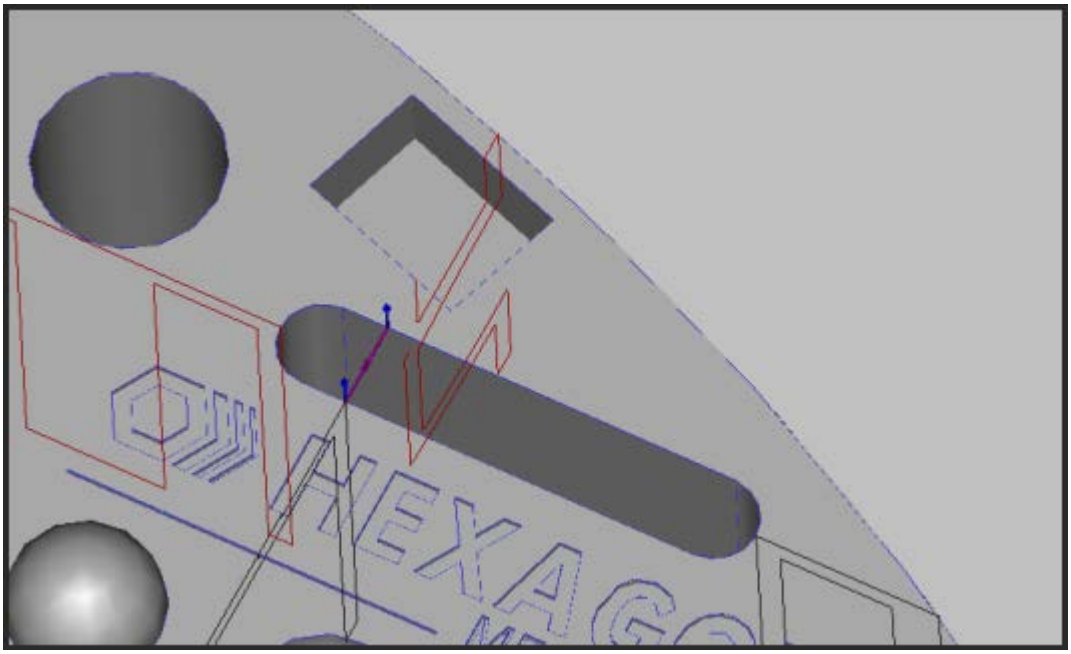
1. Near one of the sides of the slot, click on the top surface of the part in the Graphic Display view. Do not click directly on the edge. Planner draws the first edge point on the edge nearest your mouse click on the CAD model and shows the point in the **Slot Edge Points** dialog box. Any time you click to add a point to the CAD model, that point is added into the dialog box. You can use the dialog box to delete selected points or flip their vectors.

   The point's two colored arrows (blue and purple) indicate the surface vectors making up the edge point. A section cut perpendicular to the side on which you just clicked also appears on the screen. The section cut will be drawn in two colors to show the portion of the section containing the first point and where you should click for the second point.



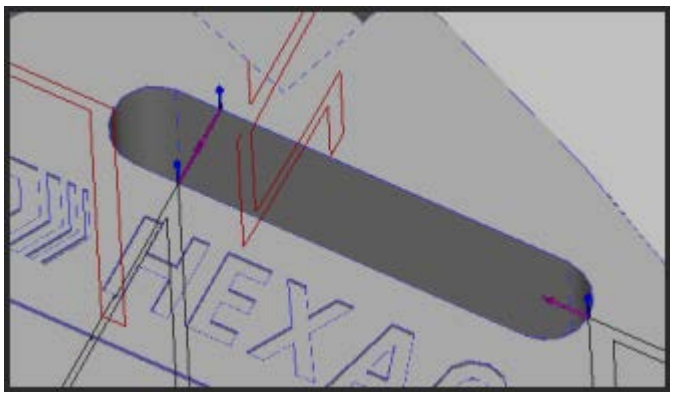
Example of the first point and the initial section cut

2. Click a point on the surface along the section cut on the other side of the slot. The second point appears drawn on the CAD model, and a second section cut, perpendicular to the first section cut, appears on the screen.



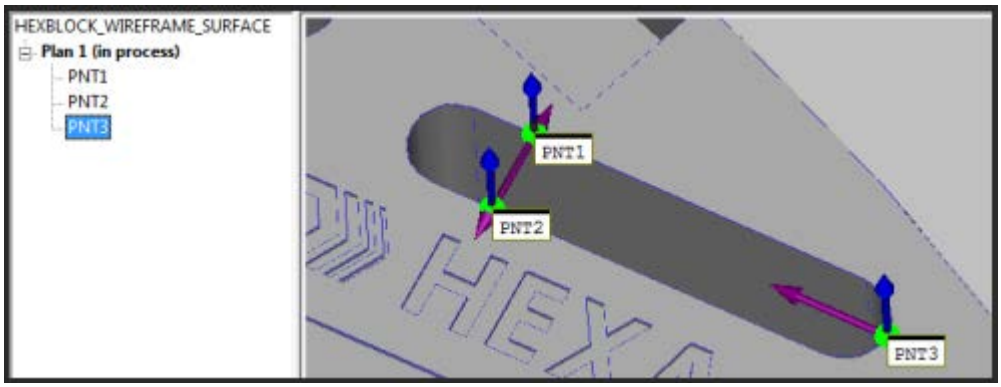
Example of the second point and the second section cut, perpendicular to the first

3. Along the second section cut, click on a surface near a slot edge. A third point appears on the CAD model on the edge.



Example of the third point

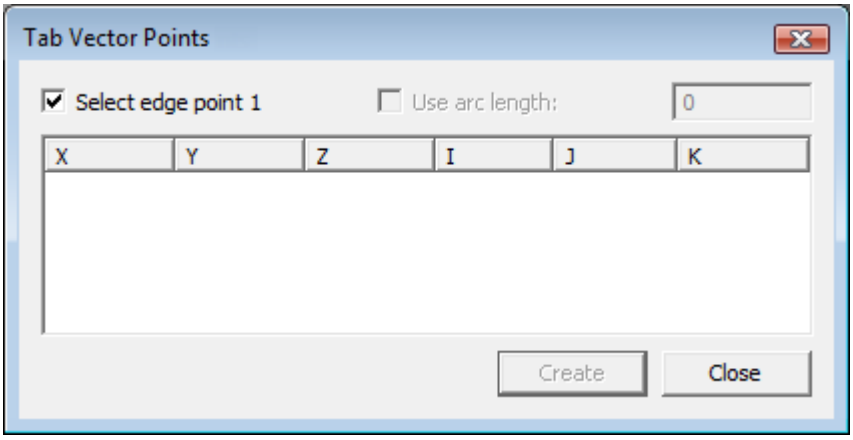
4. Click **Create**. Planner adds the points into the inspection plan.



Example of the three created points added to the plan

Creating Tab Vector Points

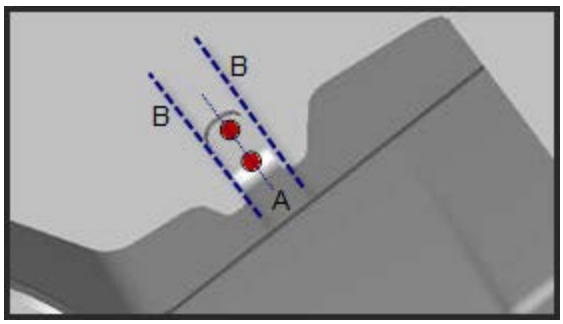
The Insert | Tab Vector Points menu item displays the Tab Vector Points dialog box.



Tab Vector Points dialog box

Similar to the Slot Edge Points functionality, this dialog box lets you create vector points along a section cut polyline that bisects a tab feature.

This provides you with a way to create vector points precisely along the middle line of a tab feature like this:



Two points on the middle line (A) between CAD boundaries (B)

Dialog Box Items

The dialog box contains these items:

Select edge point 1 - This lets you define the first edge point. This check box is initially marked and deactivated when you first access the dialog box. After you click once on the CAD model, however, the check box clears and becomes active so that you can choose a different start location if needed.

Use arc length - If marked and you click on the CAD model, Planner will create a point along the cut plane's polyline at the defined distance from the nearest surface edge. The surface edge used is the one closest to the location clicked on the CAD model.

Points list - The list contains the tab vector points as you click to define them in the Graphic Display window. The columns display each point's location (XYZ columns) and vector (IJK columns). You can use this list to delete existing points or flip their vectors. Use SHIFT and Ctrl keys while clicking with the mouse to select multiple points. Once you have selected the desired points, right-click on the selection and a small popup menu appears with these options:

Flip Vector - This flips the vector of all the selected points.

Delete - This deletes all of the selected points.

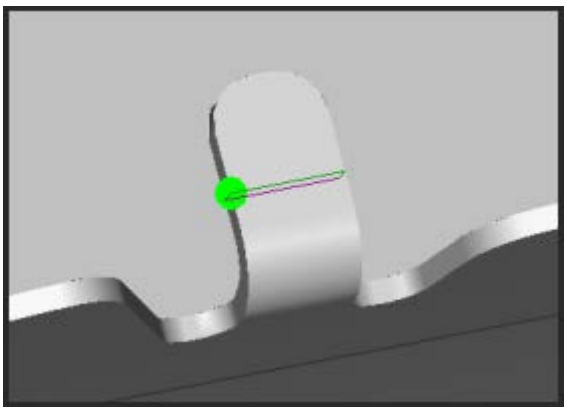
Create - This button creates any tab vector points you selected from the CAD model and adds them to the inspection plan. It becomes activated the moment you click on the CAD to select your first edge point.

Close - Closes the dialog box without adding any point features into the inspection plan.

Example of Creating Two Tab Vector Points

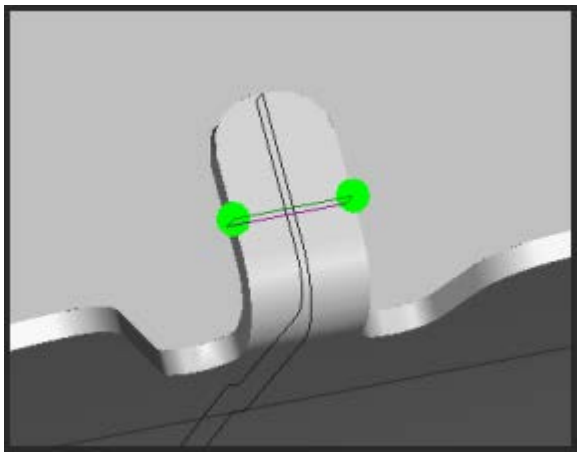
To create vector points on a tab similar to those shown in the image above, follow this procedure:

1. Click on one side of the tab, near the edge of the surface. A point appears on the surface edge. The model also displays a section cut through this point, perpendicular to the surface edge.



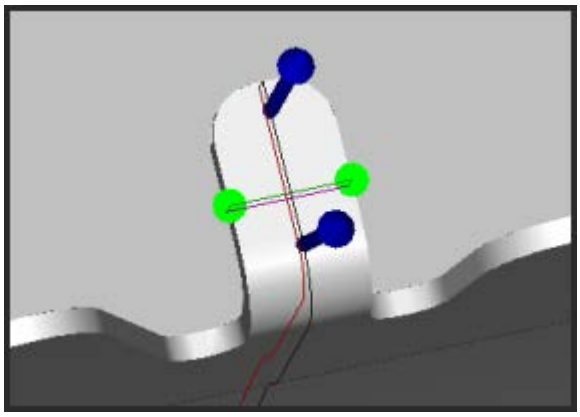
Example of the first point and the initial section cut

2. Click on this section cut near the opposite side of the tab. A second point appears. Planner uses an imaginary midpoint between these two points to generate a second section cut passing through the midpoint, perpendicular to the first section cut.



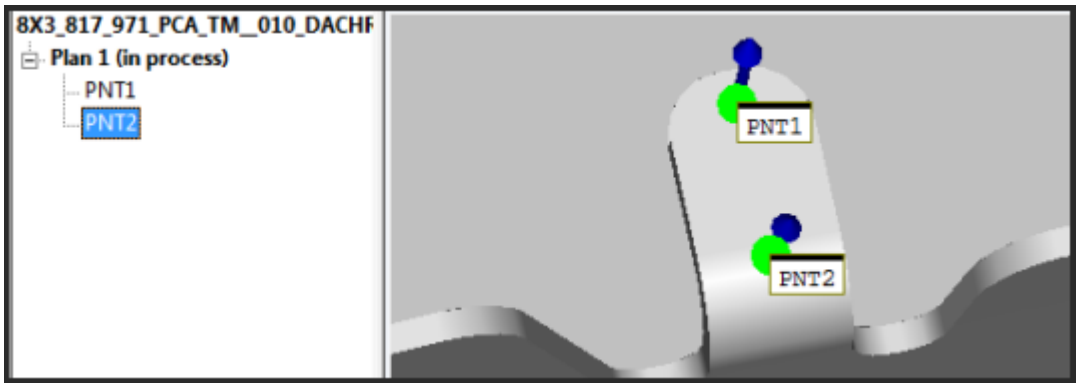
Example of the second point and the second section cut, perpendicular to the first

3. You can now click anywhere on the second section cut. Each click draws an arrow on the section cut at the clicked location. These arrows represents where vector point features will be created when you click **Create**. Alternatively, you can specify an arc length distance. Similar to arc length used in the Perimeter Points functionality, the arc length adds a point at the specified distance from the nearest surface edge. Any vector point along this second section cut also appears inside the **Tab Vector Points** dialog box. You can use the dialog box to delete selected points or flip their vectors.



Example of the two points created along the second section cut

4. Click **Create** to add the vector point features to the current plan. The dialog box remains open for additional tab vector points.

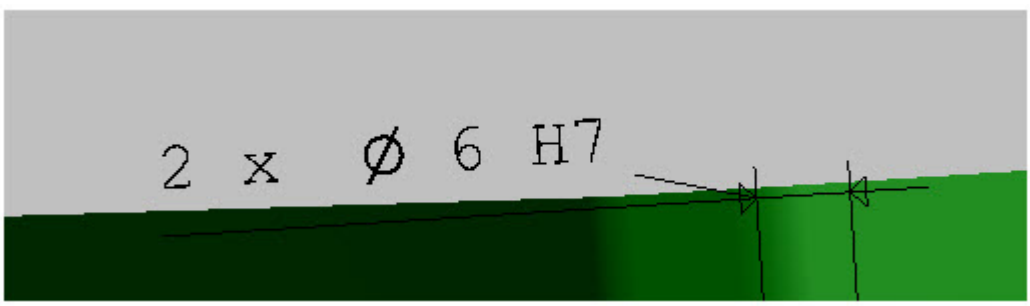


Example of the two created points added to the plan

ISO Dimension Tolerances

Note: This only applies to location dimensions created by a GD&T Selection.

ISO tolerances will have an alphanumeric label instead of normal plus and minus tolerance values. The labels will be displayed as "H7", "h7" or "G9" for example as shown below.



Example of ISO Tolerances in a PC-DMIS graphic

The following is based on ISO Standard 268-1, ISO system of limits and fits.

The letter ("H", "h", "g", etc.) indicates the tolerance class. It is case-sensitive: capitol letters indicate holes or other internal shapes, while lower-case letters indicate stems or other external shapes.

The number indicates the grade of the tolerance.

For the ISO Tolerances to work in Planner (or PC-DMIS), the tolerance values must be set up in a CSV (Comma-Separated Value) file.

The following two lines show a part of a sample CSV file and corresponds to the first half of the '7' column in Table 6 of ISO 286-2: 1988 (E) which defines the values for an H7 tolerance:

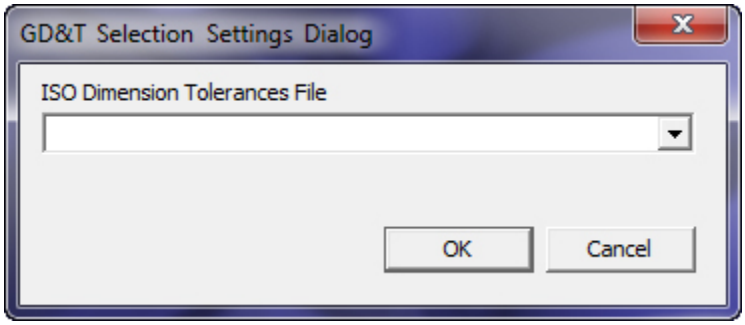
```
H7, 0, 3, 6, 10, 18, 30, 50, 80, 120, 180, 250, 315, 400, 500
0.010, 0, 0.012, 0, 0.015, 0, 0.018, 0, 0.021, 0, 0.025, 0, 0.030, 0, 0.035, 0, 0.040, 0,
0.046, 0, 0.052, 0, 0.057, 0, 0.063, 0
```

The initial entry on the first line, **H7**, defines the type of tolerance these two lines will apply to. The values in these two lines will only be applied to tolerances that have **H7** in them.

The rest of that line defines size categories. The first size category will be for dimensions greater than 0 and less than or equal to 3. The second is for greater than 3 and less than or equal to 6. The third is for larger than 6 and less than or equal to 10, etc.

The next line gives the tolerance values that match up to the size categories in line 1. The plus/minus tolerances for the first size category is +0.010/-0. The second is +0.012/-0. The third is +0.015/-0, etc.

For PC-DMIS or PC-DMIS Planner, to use a tolerance CSV file, you need to select that CSV file from the **GD&T Selection Settings** dialog. Use the **ISO Dimensions Tolerance File** combo box from the **GD&T Selection Settings Dialog** to choose the CSV file that you wish to use as shown below.



GD&T Selection Settings dialog

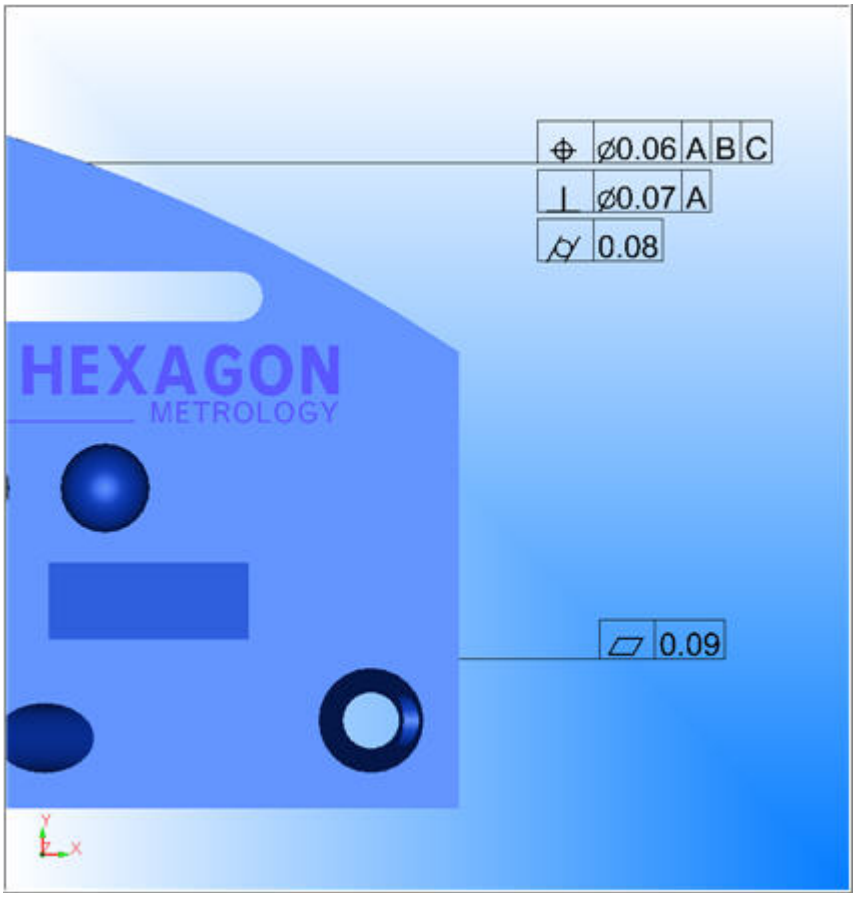
Access this dialog by selecting **Edit | Preferences | CAD GD&T Selection** menu in PC-DMIS, or through the **Edit | Setup | GD&T Selection** menu in Planner.

For more detailed information on ISO Dimension and Tolerance standards, visit the ISO web site.

Adding GD&T CAD Callouts

Using PC-DMIS Planner, you can insert GD&T CAD callouts and store them as CAD elements inside your .cad file. When the part model is imported into PC-DMIS, these callouts will appear as CAD elements in the Graphic Display window. After they are created, they function as if they were brought into PC-DMIS Planner with the model.

Note that the **CAD++** and **GD&T Selection** options in your portlock must be enabled to use this functionality.

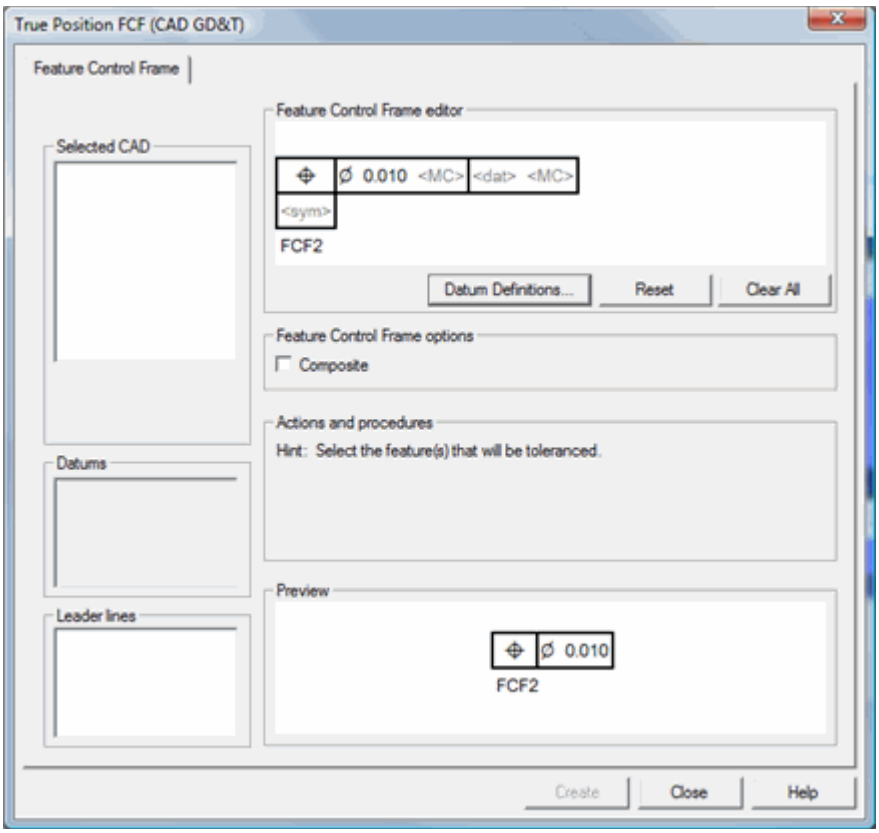


A .Cad File with GD&T Callouts

Note: If you re-import your part's original CAD model file, any existing GD&T CAD callouts that you've added in the Graphic Display window will be removed.

To Add GD&T CAD Callouts to your .Cad File:

1. Use either the **Dimensions (CAD GD&T)** toolbar or the **Insert | Dimension (CAD GD&T)** submenu, and select a GD&T Feature Control Frame (FCF) item of your choice. The **FCF (CAD GD&T)** dialog box appears.



FCF (CAD GD&T) dialog box

2. Use the **FCF (CAD GD&T)** dialog box to create the GD&T callout.

   - You can use the information presented in the "FCF (Plan) Dialog Box - Feature Control Frame Tab" topic for information on how to use this dialog to create the callout.
   - While that topic specifically describes how to use that dialog box to insert FCF dimensions into your inspection plan, the overall operation is similar enough for creating the GD&T callouts.
   - Bear in mind that when adding GD&T callouts, unlike the **FCF (Plan)** dialog box where you select a feature from the list to dimension, here you instead simply click a CAD feature on your part model with your mouse to associate the callout to that feature.

Moving the CAD GD&T Callouts

To reposition a CAD GD&T callout, place PC-DMIS into Text Box mode , and then click and drag the callout to a new location.

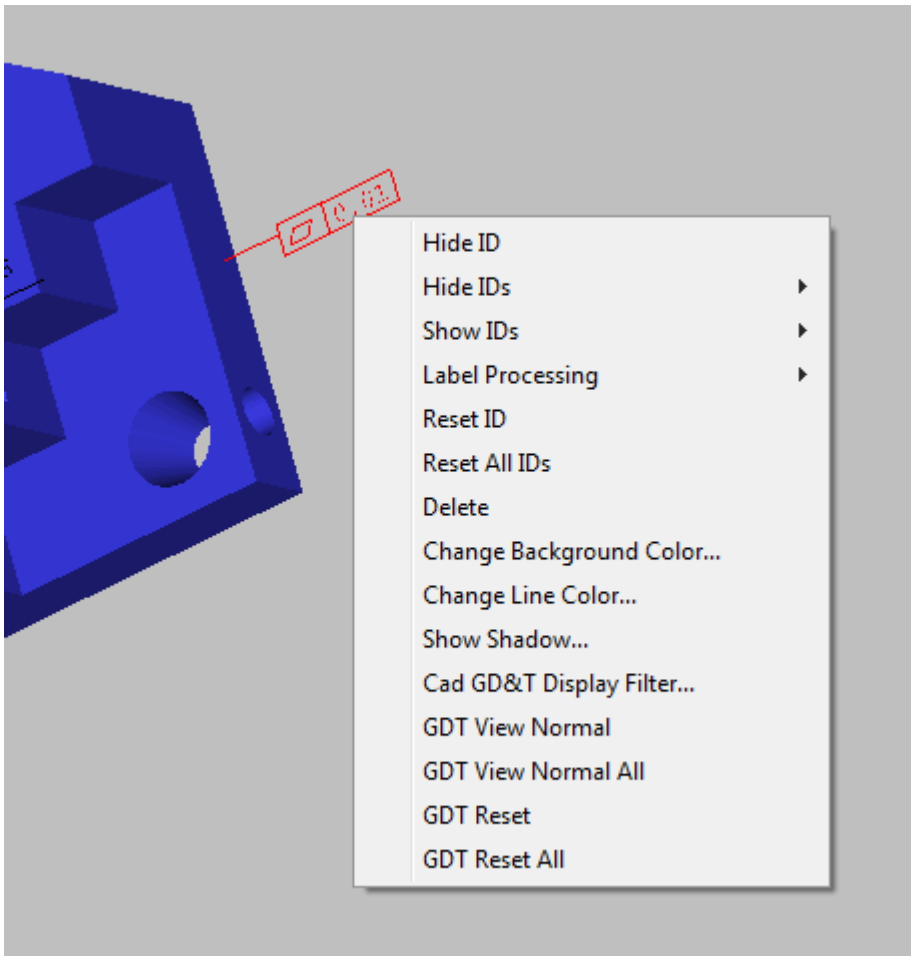
View Options for CAD GD&T Callouts

CAD GD&T callouts often become difficult to read when the image is rotated in the Graphic Display Window. The callout view can be updated to be displayed normal to the screen.

Change the Graphics Mode to **Text Box mode** by selecting the **Text Box Mode** button from the Graphics Mode toolbar, or select **Edit | Graphics Modes | Text Box** from the main menu.

Right-click on a specific GD&T callout to display an options menu (shown below) to change or reset the current view of the callout or all the callouts displayed in the graphics window. If the view of the callout has not been normalized to the screen yet, only the **GDT View Normal** or **GDT View Normal All** options are available. The **GDT View Normal** option affects only the label that was clicked on while the **GDT View Normal All** option affects all

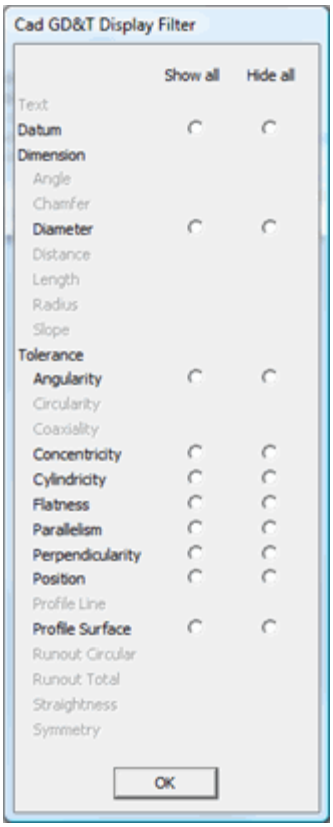
GD&T labels. If the view of the callout has been normalized to the screen, the **GDT Reset** and **GDT Reset All** options are available to switch back to the original view. The **GDT Reset** option affects only the label clicked on while the **GDT Reset All** option affects all GD&T labels.



GD&T Callout menu

Filtering the CAD GD&T Callouts

Enter Text Box mode , right-click on a GD&T callout, and select **Cad GD&T Display Filter** from the short-cut menu. The **Cad GD&T Display Filter** dialog box will appear, showing a list of CAD GD&T information and two columns of option buttons (**Show all** and **Hide all**) to toggle the visibility state of the various GD&T callout elements.

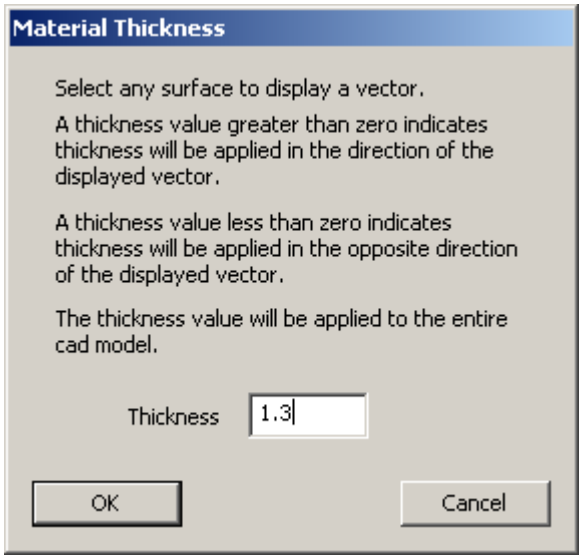


*Selecting an option button will either show or hide that item immediately. Clicking **OK** will close the dialog box, storing the current filtered state until accessed and modified later.*

Defining Material Thickness

For sheet metal parts it may be necessary to define the material thickness. To define material thickness for all features:

1. Click Thickness from the **Plans** toolbar. The **Material Thickness** dialog box appears.



2. Select any surface to display the surface vector.
3. Specify the needed **Thickness** according to the guidelines provided in the **Material Thickness** dialog box.
4. Click **OK** to apply the specified thickness to the entire CAD model. Click **Cancel** to exit without making any changes.

Viewing Toolbars and the Status Bar

You can customize the toolbars that are displayed by toggling them on or off by selecting the toolbar name from the **View | Toolbars** menu. Menu items with a check mark are displayed, otherwise they are not displayed. You can also close undocked toolbars by clicking the **X** in the corner of the toolbar.

Toolbars can be moved and docked to desired locations on the top, bottom, and right of the Graphic Display Window or on the left side of the Plan window. Undocked (floating) toolbars can also be moved to any position over the Graphic Display window and the Plan window.

Select the **View | Status Bar** menu item to toggle the display of the Status Bar, which displays along the bottom of the PC-DMIS Planner application window.

Constructed Features Toolbar

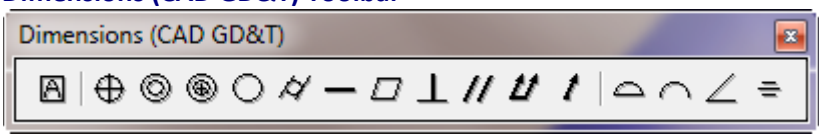
Constructed Features toolbar

With the **Constructed Features** toolbar use icons to quickly access the feature construction options available from the **Insert | Constructed Feature** submenu.

Icon	Description
	Point
	Line
	Plane
	Circle
	Ellipse
	Slot
	Cylinder
	Cone
	Sphere
	Feature Set
	Generic

For more information, see the "Constructing Features" topic.

Dimensions (CAD GD&T) Toolbar

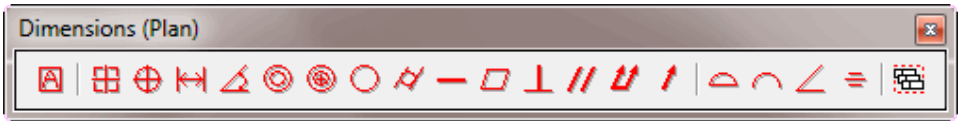


Dimensions (CAD GD&T) toolbar

With the **Dimensions (CAD GD&T)** toolbar you can quickly add CAD GD&T callouts and save them with your .cad model. When the part model is imported into PC-DMIS, these callouts will appear as CAD elements in the Graphic Display window. They do not perform any function inside of PC-DMIS other than providing visual information in the Graphic Display window.

Icon	Description
	Datum
	True Position
	Concentricity
	Coaxiality
	Circularity
	Cylindricity
	Straightness
	Flatness
	Perpendicularity
	Parallelism
	Total Runout
	Circular Runout
	Surface Profile
	Line Profile
	Angularity
	Symmetry

Dimensions (Plan) Toolbar



Dimensions (Plan) toolbar

With the **Dimension (Plan)** toolbar use icons to quickly access the options available from the **Insert | Dimension (Plan)** menu (See "Adding Dimensions to Features" topic for details).

Icon	Description
	Datum
	Location

	True Position
	Distance
	Angle
	Concentricity
	Coaxiality
	Circularity
	Cylindricity
	Straightness
	Flatness
	Perpendicularity
	Parallelism
	Total Runout
	Circular Runout
	Surface Profile
	Line Profile
	Angularity
	Symmetry
	Select All GD&T (See the "Adding all GD&T components to an Inspection Plan" section.)

Graphic Modes Toolbar

Graphic Modes toolbar

The **Graphic Modes** toolbar allows you to easily alter the display of the part on the screen. It also allows you easy access to the available **modes**.

The following buttons are available on this toolbar:

View Setup

Alters the display of the part in the Graphic Display window. See "Setting Up the View".

Scale to Fit

Enlarges or shrinks the views of the graphic in the Graphic Display window to fit your screen. See "Scaling the Drawing".

Repaint the Screen

Refreshes and redraws all the views of the part. See "Repainting the Screen".

Curve Mode

Places PC-DMIS into Curve mode (for use with wire-frame). See "Switching Between Curve and Surface Modes".

Surface Mode

Places PC-DMIS into Surface mode (for use with surface data). See "Switching Between Curve and Surface Modes".

Translate Mode

Places PC-DMIS into translate mode. See "Translate Mode".

2D Rotate Mode

Two dimensionally rotates the part for the selected view. See "2D Rotate Mode".

3D Rotate Mode

Three dimensionally rotates the part for the selected view. See "3D Rotate Mode".

Text Box Mode

Allows GD&T CAD callouts to be moved in the Graphic Display window. See "Text Box Mode".

Assembly

Displays a window allowing you to display or hide parts inside an assembly (a collection of part models). See "Working with Assemblies of Parts".

CAD Levels

Displays the **CAD Levels** dialog box which lets you define and display CAD levels for the currently displayed CAD model. See "Working with CAD Levels".

Coordinate System

Displays the **CAD Coordinate System** dialog box. This dialog box lets you create a new or select an existing coordinate system. See "Working with CAD Coordinate Systems".

CAD Lighting

Applies lighting, textures, and transparencies to CAD models. See "Setting Up Cad and Graphics".

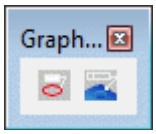
Rotation Options

Changes how items are displayed in the Graphic Display window during part rotation. See "Defining Rotation Options".

Display Symbols

Defines the symbol style and size used to represent CAD points, scanned points, and feature points in the Graphic Display window. See "Modifying Display Symbols".

Graphic Items Toolbar



Graphic Items toolbar

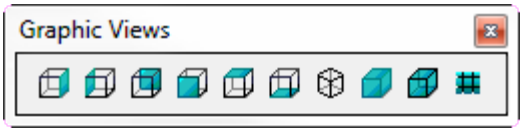
*The **Graphic Items** toolbar lets you easily show or hide identification labels for Features and GD&T callouts in the Graphic Display window.*

The following buttons are available on this toolbar:

Hides or displays Feature IDs.

Hides or displays CAD GD&T callouts.

Graphic Views Toolbar



Graphic Views toolbar

With the **Graphic Views** toolbar you can quickly and easily change how your part is displayed inside the Graphic Display window. These icons act the same as if you had used the **View Setup** dialog box to affect your view. See the "Setting Up the View" topic for information on setting up your views.

If you configured the Graphic Display window to show more than one view, these toolbar icons only affect the *blue pane* in the **Views** area of the **View Setup** dialog box.

For example, if you divided your screen into three views by using the **View** option from the **View Setup** dialog box, clicking icons from the **Graphic View** toolbar would only affect the top left (or blue) portion of the screen.

The following buttons are available on this toolbar:

Shows the X+ view of the part

Shows the X- view of the part

Shows the Y+ view of the part

Shows the Y- view of the part

Shows the Z+ view of the part

Shows the Z- view of the part

The following details relate to the above six items:

A fluid rotation animation will occur based on the view button selected. Planner will attempt to maintain the center point on the screen based on the view selected but no view scaling will occur.
If there is CAD at the center point, then that point on the CAD will stay at the center of the screen.
If there is no CAD at the screen center, then Planner will use the midpoint of the CAD extents in the view direction.

Shows an isometric view of the part

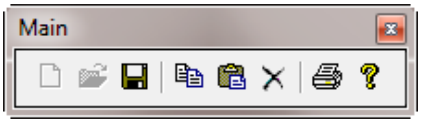
Switches between a wireframe or surface/solid view of the part

Switches surface edges view on and off.

Enhances edges of surfaces by adding a bold outline to the part. This becomes available when the **Solid** icon is selected.

Displays or hides the 3D Grid

Main Toolbar



Main toolbar

The **Main** toolbar lets you perform these main tasks.

The following buttons are available on this toolbar:

New

Creates a new .cad file to hold your inspection plans.

Open

Opens an existing .cad file.

Save

Saves the current .cad file storing the inspection plans therein.

Copy

Copies selected command(s), allowing you to paste the selection into another plan. This makes a copy of the selected item(s).

Paste

Pastes items stored in the Clipboard into the current plan.

Delete

Deletes selected items from the current plan, or if a plan is selected, it deletes the entire plan.

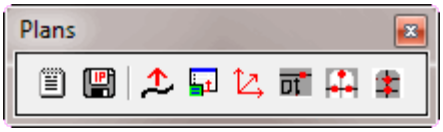
Print

Prints out the part model, feature labels and GD&T notes.

Index

Displays PC-DMIS Planner's help file.

Plans Toolbar



Plans toolbar

The **Plans** toolbar lets you perform these tasks.

The following buttons are available on this toolbar:

New

Displays a dialog that is used to create a new Inspection Plan.

Export Plan

Used to export a plan for use in other applications.

Thickness

Used to change a Thickness value of a feature.

Comment

Used to add a Comment to the Inspection Plan.

Alignment

Used to update the alignment in the Inspection Plan.

Perimeter Point

Used to update Perimeter Point settings in the Inspection Plan.

Slot Edge Points

Used to update Slot Edge Point settings in the Inspection Plan.

Tab Vector Points

Used to update Tab Vector Point settings in the Inspection Plan.

Auto Features Toolbar

The **Auto Features** toolbar lets you determine what features to include in your inspection plan.

Auto Features toolbar

Icon	Feature Type
	Surface Point
	Edge Point
	Angle Point

	Corner Point
	Line
	Plane
	Circle
	Ellipse
	Square Slot
	Round Slot
	Notch
	Flush and Gap
	Polygon
	Cylinder
	Cone
	Sphere
	Edge Scan
	3D Scan
	Freeform Scan

See "Creating Auto Features"

Tools Toolbar

Tools Toolbar

This toolbar lets you work with aspects of the Graphic Display window.

The following buttons are available on this toolbar:

Capture to Clipboard

Captures a screen capture of the Graphic Display window to your computer's Clipboard. See "Capturing the Graphic Display window".

Capture to File

Captures a screen capture of the Graphic Display window to a file. See "Capturing the Graphic Display window".

CAD Text Colors

Changes colors used for background, grid, highlight, and mouse-over highlight. See "Colors Tab".

CAD Info

Displays CAD information. See "Viewing CAD Information".

OpenGL Settings

Changes parameters that affect the display of the part in solid view. See "OpenGL Tab".

Edit CAD Vectors

Displays a dialog box that lets you easily change vectors for CAD surfaces. See "Editing CAD Vectors".

Status Bar

Select **View | Status bar** to show or hide the status bar located at the bottom of your screen. The status bar displays the following helpful information:

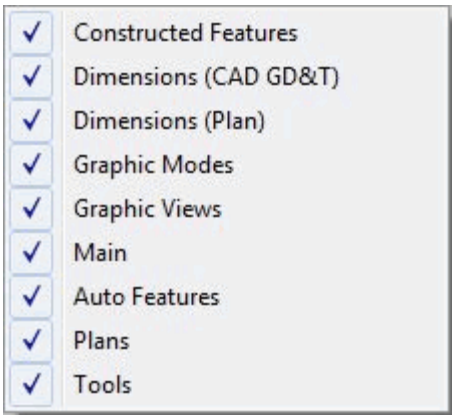
- Descriptions for toolbar items.
- Description of cad elements selected on the part during the creation of auto features.
- Status of operations and instructions for completing operations.

Using Shortcut Menus

Shortcut menus allow you to do common commands at the simple click of a mouse button. This section explains how to access the various shortcut menus and what the various menu options do. Shortcut menus exist when the program activates these locations.

Shortcut Menus on the Toolbar Area

When right-clicking on the **Toolbar** area, PC-DMIS Planner displays a list of the toolbars and other options that you can instantly access. These include:



The following describes the options available on this menu:

Constructed Features
Adds or removes the **Constructed Features** toolbar to and from the toolbar area.

Dimensions (CAD GD&T)
Adds or removes the **Dimensions (CAD GD&T)** toolbar to and from the toolbar area.

Dimensions (Plan)
Adds or removes the **Dimensions (Plan)** toolbar to and from the toolbar area.

Graphics Modes
Adds or removes the **Graphics Modes** toolbar to and from the toolbar area.

Main
Adds or removes the **Main** toolbar to and from the toolbar area.

Auto Features
Adds or removes the **Auto Features** toolbar to and from the toolbar area.

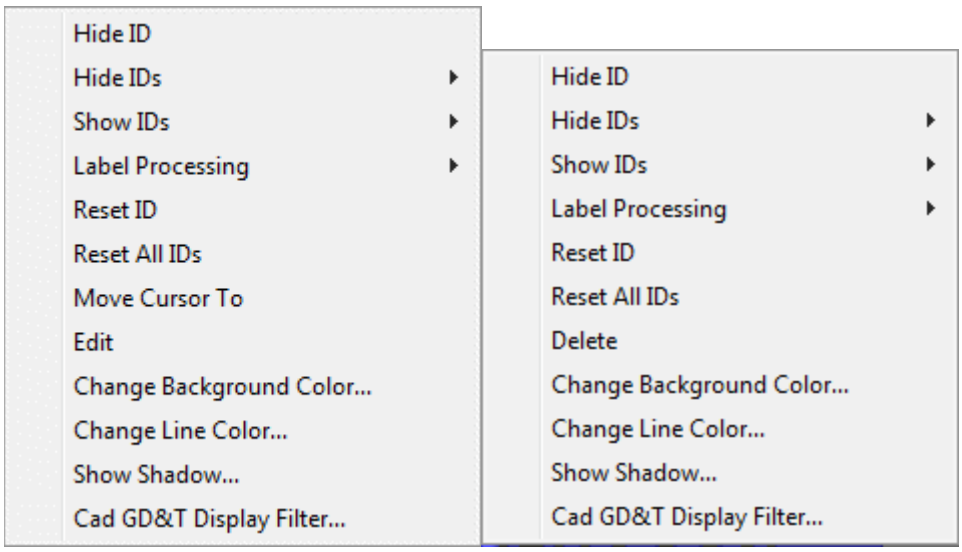
Plans
Adds or removes the **Plans** toolbar to and from the toolbar area.

Tools
Adds or removes the **Tools** toolbar to and from the toolbar area

These toolbars are discussed in detail in the "Viewing Toolbars and the Status Bar" chapter.

Shortcut Menus for Feature IDs or CAD GD&T Callouts

To access this shortcut menu, right-click on a feature ID label or CAD GD&T callout inside the Graphic Display window. You can do this in any mode when right-clicking on a feature ID. For this to appear on a CAD GD&T callout you must be in **Text Box** mode.



Shortcut menus for Feature ID Label (left) and CAD GD&T Callout (right)

Note: Right-clicking on a GD&T callout will display many of these same menu items; however, only the show and hide menu items function in this context.

The following describes the options available on these menus:

Hide ID option
This menu option hides a single ID label in the Graphic Display window. A hidden label still exists; it's merely hidden from view.

Show IDs, Hide IDs option
These submenus let you show or hide the different feature ID labels. You can choose to show or hide these types of ID labels:

- All
- Feature Labels
- Feature Control Frames

If you choose to hide the ID labels, the ID labels still exist and are merely hidden from view.

Label Processing option
This submenu contains the **Automatic Label Positioning** option. This menu option allows you to perform a one time automatic positioning of feature ID labels inside the current view.

Reset ID option
This option moves a single ID label back to its original location next to the feature.

Reset All IDs option
This option moves all ID labels back to their original locations next to their respective features.

Move Cursor To option
Select this menu option to select the command in the Plan list associated with the feature ID.

Edit option
This lets you edit the command by accessing the dialog box used to originally create the command.

Delete option
This option only appears if you are in Text Box mode. It deletes the ID label for the selected CAD GD&T callout (Feature Control Frame) from the Graphic Display window.

Change Background Color option
This option displays the **Change Label Options** dialog box. You can use this dialog box to change the background color for the feature's ID label. You can choose to change all feature labels to this new color or just the current label type.

You can make your changes the default settings by clicking the **Default** button.

Note: This does not function with Feature Control Frames.

Change Line Color option
This option displays the **Change Label Options** dialog box. You can use this dialog box to change the border color for the feature's ID label. You can choose to change all feature label borders to this new color or just the current label type.

You can make your changes the default settings by clicking the **Default** button.

Note: This does not function with Feature Control Frames.

Show Shadow option
This option displays the **Change Label Options** dialog box. You can use this dialog box to display or hide a small

shadow under the feature's ID label. You can choose to add a shadow to all label types or to just the current label type.

You can make your changes the default settings by clicking the **Default** button.

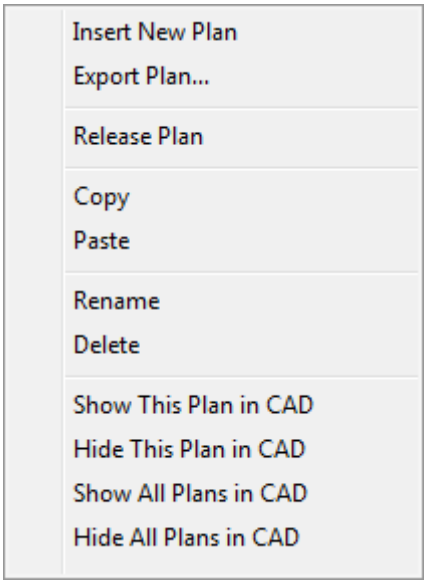
Note: This does not function with Feature Control Frames.

CAD GD&T Display Filter option
This option displays a dialog box letting you filter the display of any CAD GD&T callouts on the Graphic Display window. See "Adding GD&T CAD Callouts" for more information.

Shortcut Menus in the Plan List

Two shortcut menus provide various options when you right-click plans or plan commands in the Plan list.

Plan Shortcut Menu



Plan Shortcut Menu appears when you right-click a Plan item

The following describes the options available on this menu:

Insert New Plan
Creates a new Inspection Plan at the bottom of the Plan list.

Export Plan
Exports the selected plan to a .ip file. See "Exporting an Inspection Plan".

Release Plan
Causes the selected plan to be embedded when the PC-DMIS Planner CAD file is saved. When you import the PC-DMIS Planner CAD file into PC-DMIS, you will also be able to import the embedded Inspection Plan(s).

Copy
Copies all of the command items for the selected plan to the internal clipboard.

Paste
Pastes all of the commands from the internal clipboard to the selected plan.

Rename
Provides an edit box with a cursor to allow you to type the new name for the plan.

Delete
Deletes the selected plan.

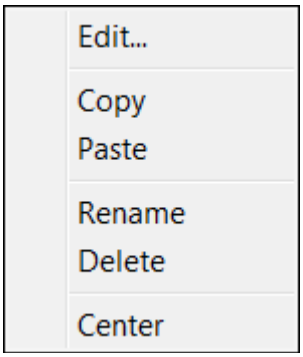
Show This Plan in CAD
Displays elements from the selected plan in the Graphic Display window.

Hide This Plan in CAD
Hides elements from the selected plan in the Graphic Display window.

Show All Plans in CAD
Displays elements from all plans in the Graphic Display window.

Hide All Plans in CAD
Hides elements from all plans in the Graphic Display window.

Plan Command Shortcut Menu



Plan Command Shortcut Menu appears when you right-click a command item

The following describes the options available on this menu:

Edit
Displays the dialog box for the selected command. Make any needed changes and click **Apply**, **OK**, or **Create** to save your changes. You may also double-click on commands in the Plan list to open the associated dialog box. You can also right-click on a feature ID label inside the Graphic Display window, and select **Edit** there as well.

Copy
Copies the selected command to the clipboard.

Paste
Pastes the command from the clipboard to the selected plan.

Note: Changes made to commands that are copied from one plan to another will be affected in every instance of that command (e.g. renaming CIR1 to CIRCLE1 will rename every copy of that command).

Rename
Provides an edit box with a cursor to allow you to type a new name for the command.

Delete
Deletes the selected command.

Center
Centers the selected feature in the Graphic Display window.

Configuring 3Dconnexion 3D Mouse Devices

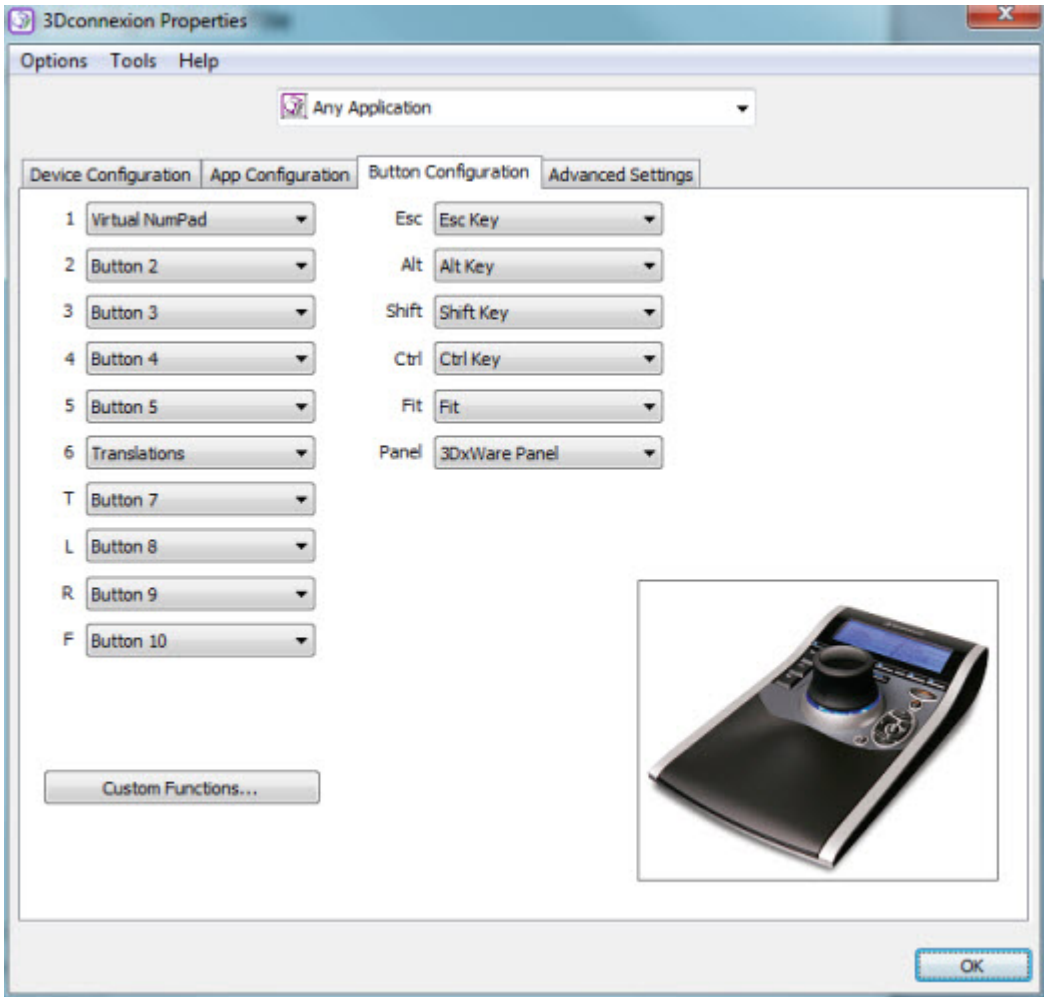
3Dconnexion 3D Mouse devices communicate with PC-DMIS via the 3DxWare interface. Information can be found in the following topics.

Note: Contact a 3DConnexion representative for support on these devices.

Editing the 3DxWare Configuration File

1. Start the PC-DMIS or PC-DMIS Planner (previously known as Inspection Planner) application.

2. Open the 3Dconnexion Control Panel by double-clicking the 3DxWare icon located on your system tray



3Dconnexion Properties dialog box

a. See if the **Application** drop-down list in the top-center of the 3Dconnexion Properties dialog box contains an entry for something like “WAI Applications”.

b. If “WAI Applications” is available, then select it as the current configuration and follow the information in the topic "Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices". Otherwise, continue with this procedure.

c. If "WAI Applications" is not available and you only see "Any Application", select **File | Save As** and save the name of the “Any Application” configuration to "My PC-DMIS", "PC-DMIS Planner", "WAI

Applications", or whatever you like. This configuration will be used by PC-DMIS and PC-DMIS Planner to define the rotation speed, etc settings.

d. Close the 3Dconnexion Properties dialog box.

3. Close your PC-DMIS or PC-DMIS Planner application.

4. Depending on your operating system, navigate to one of the following:

- If you use a Windows XP operating system, access Windows Explorer and navigate to the following location, where MyAccount refers to your user account on the computer:

C:\Documents and Settings\MyAccount\Application Data\3Dconnexion\3DxWare\profiles

- If you are use a Windows Vista or later operating system, access Windows Explorer and navigate to the following location, where MyAccount refers to your user on the computer:

C:\Users\ MyAccount\AppData\Roaming\3Dconnexion\3DxWare\profiles

5. Using a text editor (such as Microsoft's WordPad or Notepad), edit the most recent “user??.scg” file (for example, user00.scg).

- In the “[IDENTIFICATION]” section of this file, verify this is the file you just saved by comparing the “APPLICATION_NAME” to what you entered in step 2c above. Note that a comma might be preceded by a backslash (\).

```
[IDENTIFICATION]
APPLICATION_NAME = "My PC-DMIS\, PC-DMIS Planner"
```

6. In the “[CONFIGURATION]” section, edit the “APP_CONTROLS_BUTTONS” and “EXECUTABLES” values to enable them for use with either the PC-DMIS or PC-DMIS Planner executable.

```
[CONFIGURATION]
APP_CONTROLS_BUTTONS = TRUE
EXECUTABLES = { "PCDLRN.exe", "InspectionPlanner.exe" }
```

7. As you can see, the above “EXECUTABLES” indicates that this configuration file will be used by both PC-DMIS and PC-DMIS Planner.

8. Save and close the file.

9. You can now map custom functions to your device's buttons by following the information located in these topics:

"Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices"

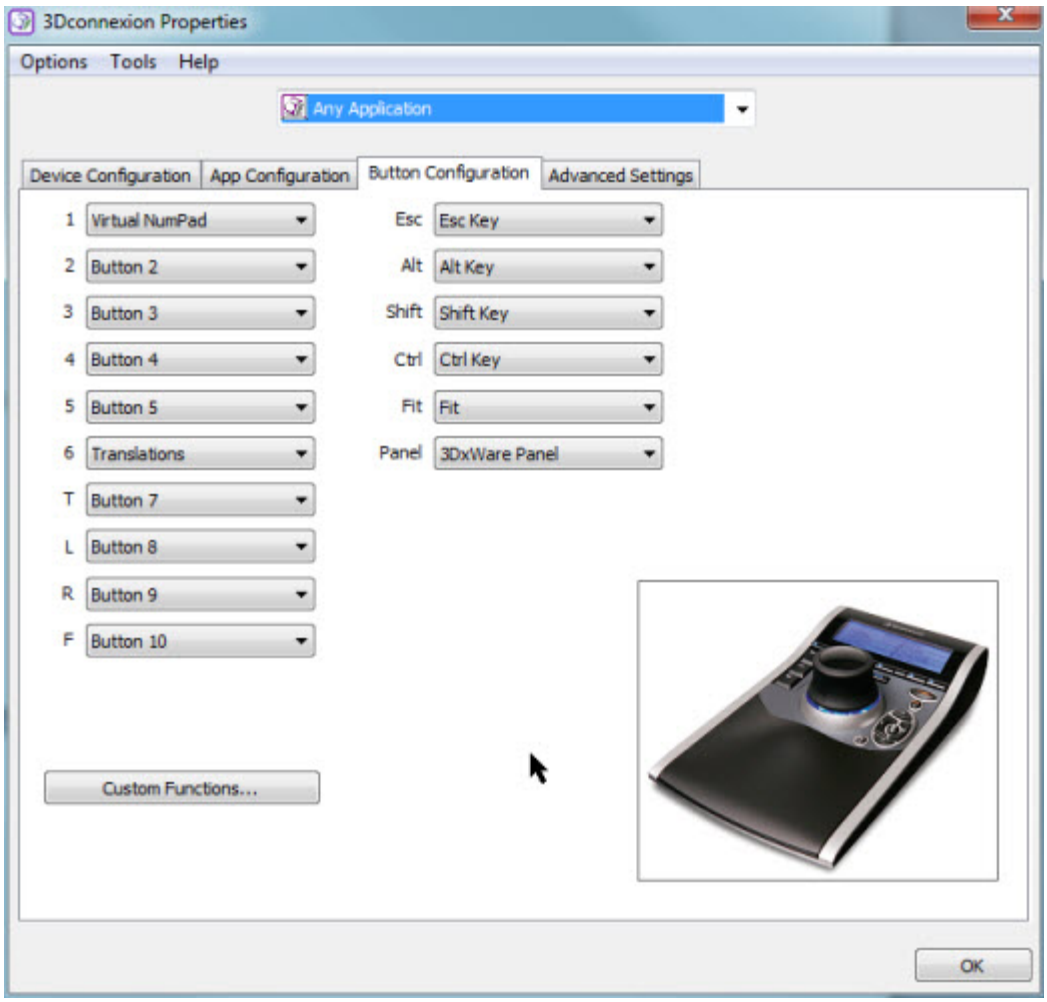
Assigning Functions to Buttons on 3Dconnexion 3D Mouse Devices

This topic describes how to map PC-DMIS or PC-DMIS Planner functionality to the available buttons on 3Dconnexion 3D Mouse devices by using the 3Dconnexion Properties dialog box.

1. Start PC-DMIS or PC-DMIS Planner.

2. Load a part program.

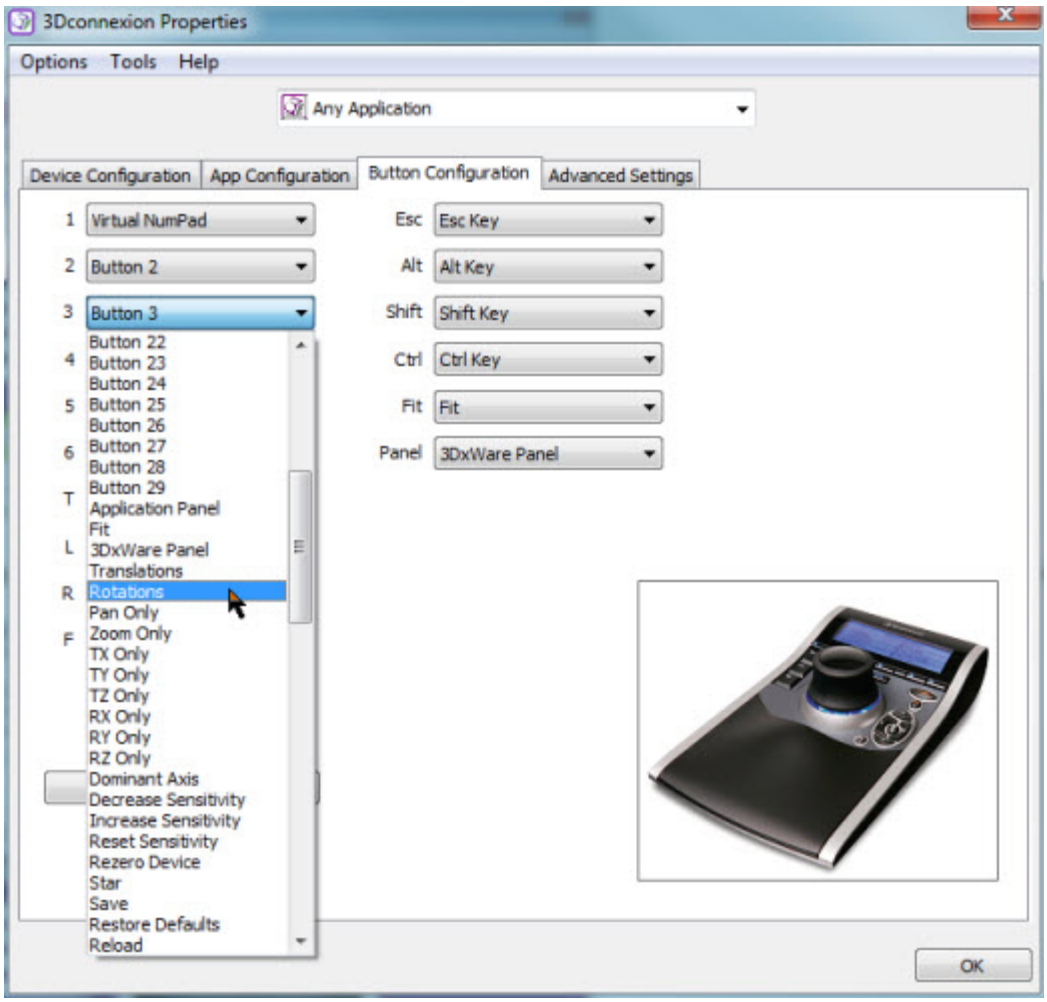
3. Double click the 3DxWare icon located in your System Tray. The 3DxWare configuration application, 3Dconnexion Control Panel, appears showing a graphical depiction of your motion device.



3Dconnexion Properties dialog box showing a SpacePilot Device

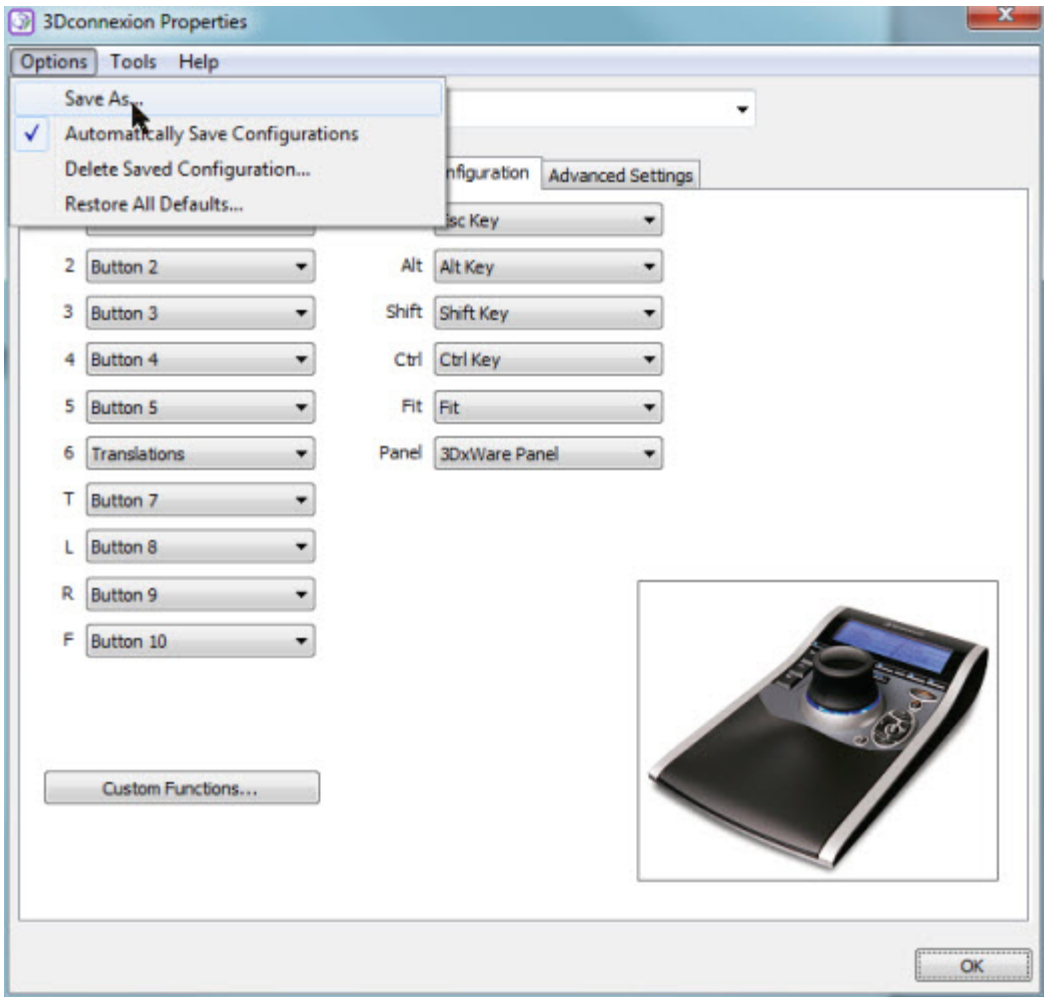
4. Select the correct configuration name by selecting it from the **Application** drop-down in the top-center of the 3Dconnexion Properties dialog box. If a relevant name doesn't appear in the list, you probably need to add it to the 3DxWare configuration file. See "Editing the 3DxWare Configuration File".

5. To change map functionality to a button, select the button's drop-down list, click the down arrow icon to its right, and select the new function from the list.



3Dconnexion Properties dialog box for SpacePilot Showing Mapping of a Button

6. If the **Configuration | Automatically Save Configurations** menu item is checked, your changes will be saved in your current configuration file.



3Dconnexion Properties dialog box with the Automatically Save Configuration Menu Checked

7. Make any other changes as needed.
8. Click **OK** to accept your configuration and close the dialog box.

3Dconnexion 3D Mouse Device Modes

These devices are always in a certain mode. A symbol appears in the lower right corner of the Graphic Display window to indicate the current mode for CAD PZR, PCDMIS pan, or PCDMIS Zoom.

Standard Modes

- Show cursor
- CAD PZR
- PCDMIS rotate
- PCDMIS pan
- PCDMIS zoom
- CAD PZR dominant - The image in the lower right corner of the Graphic Display window displays a “1” to signify that only 1 of the three functions (Pan, Zoom, or Rotate) will be used at a time.

Demo Mode

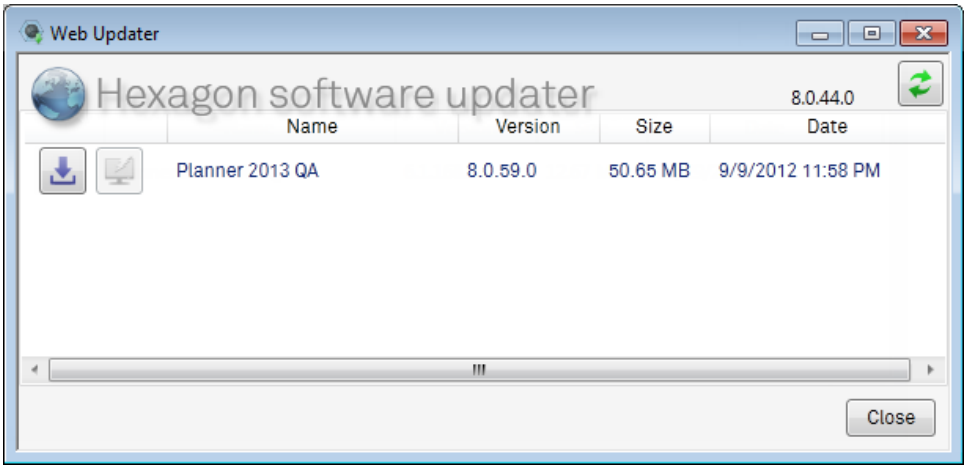
These devices can also place the Graphic Display window into a demo mode. This mode causes the Graphic Display window to rotate automatically several times a second. To enter or exit the demo mode, press the Ctrl key and press the programmed **Scale to Fit** button on the device.

Updating the Software

If you're connected to the Internet, you can use the **Help | Check for Updates** menu item to easily check for, download, and then apply updates to your software. You can also choose to have the software automatically check for updates each time you start your software by choosing the **Help | Automatic Check at Startup** menu item. This provides a convenient way to keep your software up-to-date without downloading and manually installing entirely new versions of the software.

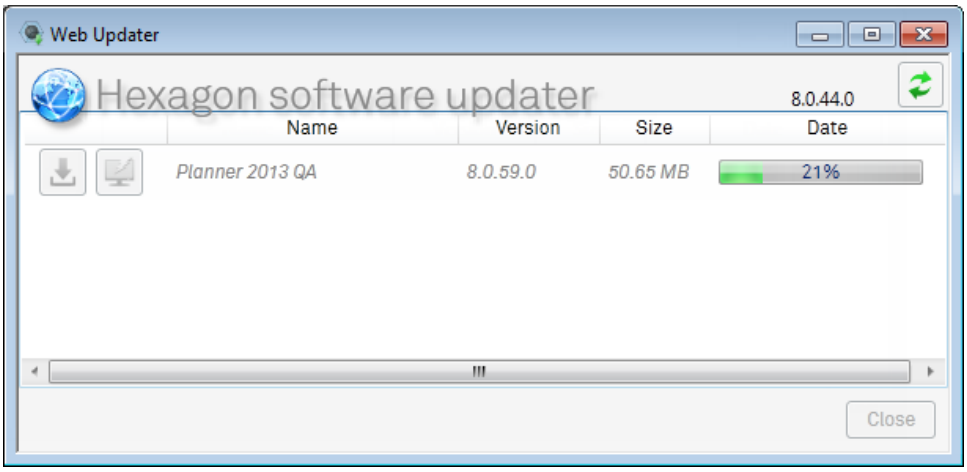
Note: You must be logged in with Administrator privileges to download and install any updates, though a user with Standard privileges can check if an update exists. Your firewall must not block this address: http://www.wilcoxassoc.com/WebUpdater

When the dialog box first opens, or whenever you click the **Refresh list of updates** icon , the software updater queries Hexagon Metrology servers, looking for available software updates. If an update exists, it appears in the Hexagon software updater. Your current build of the software is listed in the gray area of the updater. The new build you will download and install, displays its name, version, and size.

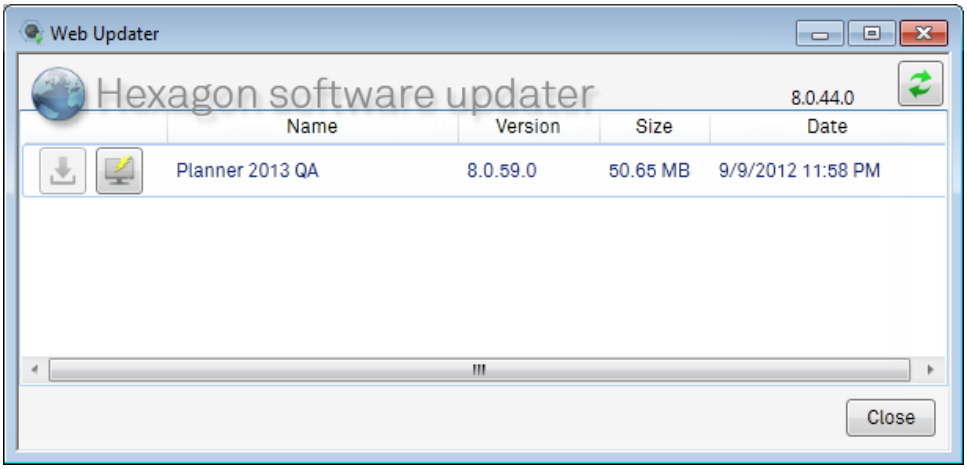


Click the **Download the Update** icon for the update you want to download and install.

The download process begins.



Once the download finishes, it's ready to install.



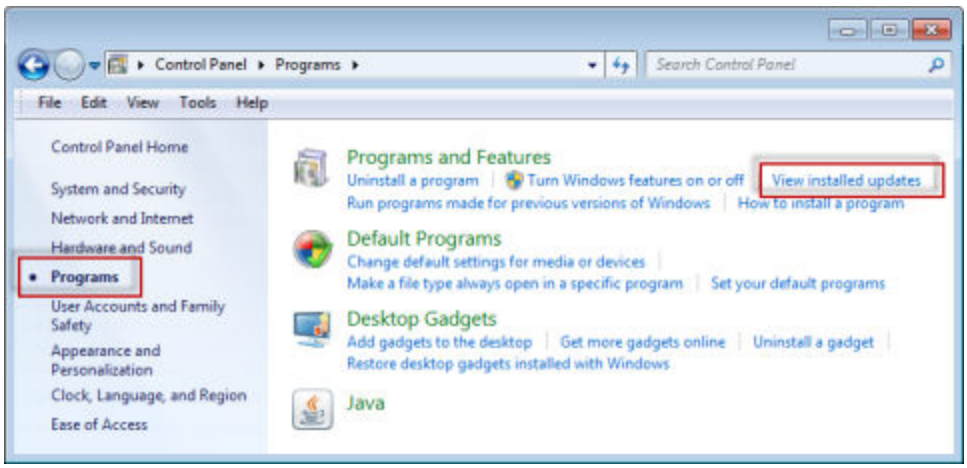
Click the **Install the Update** icon for the update you want to install.

The install process closes your software, and automatically installs the new update. Once the install process finishes, restart the application.

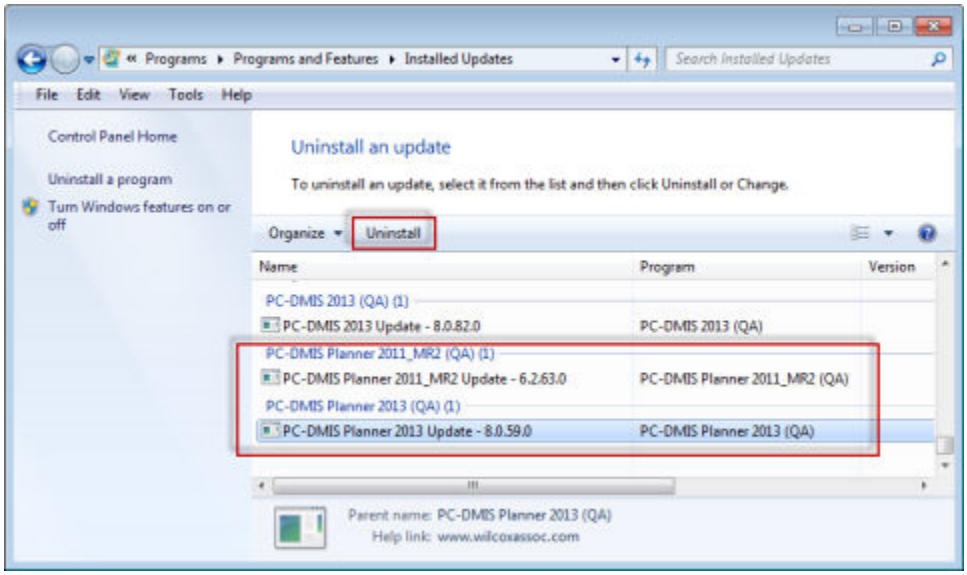
Restoring a Previous Version

If you've applied an update, you have the ability to roll-back and restore your installation to a previous version of the software. To do so, follow this procedure:

1. Access your computer's Control Panel.
2. Under Programs, click View installed updates.



3. Navigate through the list of updates to the specific software item to remove. Select the item you want to remove and click **Uninstall**.



PC-DMIS reverts back to the state of the previous build that was installed. If you previously installed one or more patches, you can revert each patch until your install reverts back to the baseline for the installed patches.

Note: All the update patch files are stored as .msp files in appropriate subdirectories located under the C:\ProgramData\WAI\Updates\PC-DMIS-Planner\ directory.

Index

3
3D Scan Feature 139
A
Alignments 173
Angle Point 133
Assemblies 80
Auto Geometry Features
  Creating 133
  Dialog Box Options 135
B
Background Color 96
C
CAD file
  Importing 49
CAD Information 95
CAD Vectors 94
  Editing 94
Callouts 170, 194
Capture To Clipboard 102
Capture To File 102
CATIA 5 DCI 66
CATIA V5 File 49
Change Management
  in Planner 124
  Options 124
Circle 133
Comments
  Inserting 178
Cone 133
Constructed Features 149, 152, 199
  Operations 152
Corner Point 133
Cylinder 133
D
Datums 169
DCI 66
Dimension 155
  Distance 158
Dimensions 155, 156, 158, 160, 200, 201
  Angle 159
  Location 156
Direct CAD Interfaces 66
Disable High Quality Transparency 37
Display Symbols 13
DXF 51
E
Edge Point 133, 187
Edge Scan Feature 137
Ellipse 133
Exporting 48
F
Fix Surface Vectors 94
Flip Surface Vectors 94
Flush & Gap 133
G
GD&T Callouts 38, 170
Generic Feature 151
Gradient Settings 96
Graphic Items toolbar 204
Graphic Modes 71, 202
Graphic View toolbar 72, 204
Graphics Display Window 71
H
Highlight Color 96
I
Importing 48
Inspection Plan 64, 108
  Exporting 67
  Importing 64
  New 108
Inspection Planner Default File 43
Inspection Planner Parameters 42
Inspection Planner Rules 45
Inspection Plans 64
  Updating 124
IPD 43
IPR 45
ISO Dimension Tolerances 193
Iterative Alignment 176
L
Line 7, 133
M
Material Thickness 198
Measured Features 133
MOHL 96
Mouse Over Highlight 20, 96
N
Notch Slot 133
O
OpenGL Options 103
P
Perimeter Points Feature 142
Plane 33, 133
Planner
  CAD Files 107
  Introduction 1
  Status Bar 199
  Toolbars 199
Polygon 133
Preferences 5
Pro/Engineer DCI 66
R
Releasing Inspection Plans 108
Repainting the screen 73
Round Slot 133
S
Saving Inspection Plans 108
Screen Colors 96

Screen Counters 105
Section Cuts..... 179
Shortcut Menus..... 210, 213
Slot Edge Points 187
SolidWorks..... 58
Solidworks DCI 66
Sphere 133
Square Slot 133
Surface Points..... 133

T

Tab Vector Points 190
Text Box Mode..... 77
Transforming CAD 88

U

Unigraphics DCI..... 66
Updating Inspection Plans 124
Updating the Software 222

V

Vector Points..... 190

Glossary

D

DCI: This is the acronym for Direct CAD Interfaces.

DCT: This is the acronym for Direct CAD Translator.

I

IP: This is the acronym for Inspection Plan.

IP CAD File: This differs from a PC-DMIS CAD file in that it has embedded Inspection Plans.

IPD: This is the acronym for Inspection Planner Defaults. Defaults are defined in a file that contains the parameters that are applied to the Inspection Plan.

IPR: This is the acronym for Inspection Planner Rules. A rules script is created using Basic Script Editor within PC-DMIS. Rules are applied to the Inspection Plan.

P

PC-DMIS Planner: Refers to the stand-alone application that is used to create inspection plans.