Inspection Planner

Version 1.0

by

Wilcox Associates, Inc.

http://www.wilcoxassoc.com

Last Updated: July 27, 2007

Table Of Contents

PC-DMIS Inspection Planner 1

Inspection Planner: Introduction 1

Using PC-DMIS Inspection Planner 1

Installing Inspection Planner 1

Importing CAD Data 2

Inspection Planner CAD Files 2

Creating a New Inspection Plan 3

Creating Auto Geometry Features 4

Constructing Features 9

Adding Dimensions to Features 13

Creating an Alignment 18

Creating an Iterative Alignment 20

Inserting Comments 22

Deleting Inspection Plan Elements 22

Exporting an Inspection Plan 22

Changing the ID Setup 23

Defining Material Thickness 24

Viewing Toolbars and the Status Bar 24

Plan List Shortcut Menus 25

Graphics Display Window 26

Using Inspection Plans in PC-DMIS 33

Inspection Planner Parameters and Rules 33

Importing the Inspection Plan 38

Performing the Path Optimization 39

Dual Arm Path Optimization 42

Path Optimization Summary 43

Void Detection 43

Automatically Inserting Move Commands 44

Glossary 47

Index 49

PC-DMIS Inspection Planner

Inspection Planner: Introduction

PC-DMIS™ Inspection Planner (IP) works in conjunction with your CAD software package and PC-DMIS to aid the Design Engineer or Quality Engineer in designing a plan that determines which features of the geometry and GD&T/dimension to use in the inspection process.

PC-DMIS Inspection Planner only works with PC-DMIS Version 4.2 and above.

Inspection Planner (IP) 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. PC-DMIS then imports the plan and automatically generates a part program from it for either a CMM or CNC machine tool for in-process inspection.

Inspection Planner currently supports **CATIA V5** files.

This manual will walk you through the steps to create an Inspection Plan using your CATIA V5 file and how to use IP commands inside PC-DMIS to automatically generate a PC-DMIS part program.

This documentation covers these topics:

- Using PC-DMIS Inspection Planner
- Using Inspection Plans in PC-DMIS

Using PC-DMIS Inspection Planner

This section describes how to install and use PC-DMIS Inspection Planner (IP). IP allows you to perform various sub tasks needed to create an Inspection Plan that is stored in an IP CAD file and can also be exported as a text file for use with PC-DMIS.

The **Inspection Planner** option must be enabled on your portlock to use the Inspection Planner capabilities.

Installing Inspection Planner

1. Log on to the Wilcox ftp site: ftp.wilcoxassoc.com. Browse to the InspectionPlanner directory. Download **PCDMISInspectionPlanner.zip** to a specified directory on your hard drive.
2. Open the ZIP file and extract the **PCDMIS_Inspection_Planner_install.exe** file.
3. Double-click the extracted executable to install PC-DMIS Inspection Planner. Follow the on-screen instructions.
4. You can now launch Inspection Planner from the provided desktop shortcut or from the Inspection Planner program group.

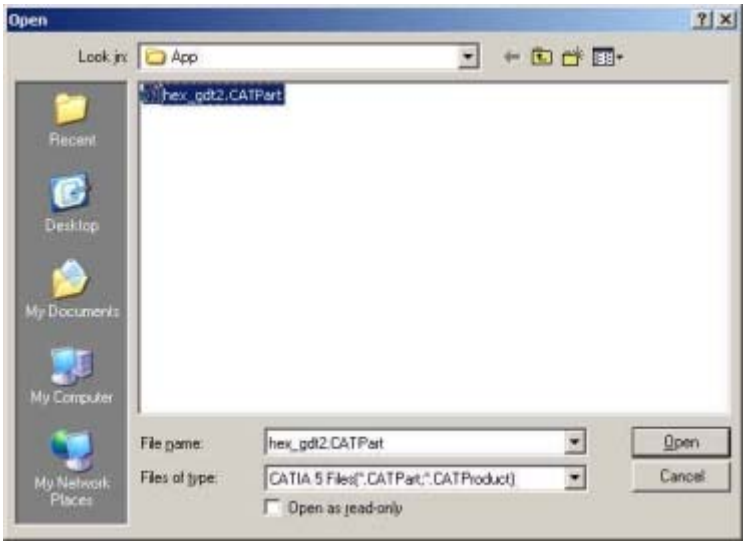
Importing CAD Data

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

Currently, Inspection Planner only supports **CATIA V5** files (*.CATPart and *.CATProduct).

To Import CAD Data:

1. Select the **File | Import | Catia 5** menu item. The **Open** dialog box appears.



Open dialog box

2. Browse to and select the needed CATIA 5 file. Click **Open** to import the file. Inspection Planner imports the CAD data with any accompanying GD&T/dimension information.

Important: In order to save an Inspection Planner CAD file, you must first import CAD data.

Inspection Planner CAD Files

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

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

The following command are used to Create, Open, Save and Close IP CAD files:

- Click **New** from the **File Operations** toolbar or select the **File | New** menu item. This creates a new IP CAD file named "Untitled". The file is not saved at this point, but would require you to save the file.

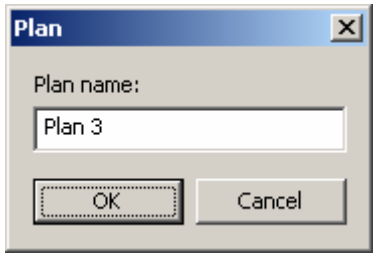
- Click **Open** from the **File Operations** toolbar or select the **File | Open** menu item. This opens the **Open** dialog box. Browse to and select the needed CAD file. Click **Open** to open the file.
- Click **Save** from the **File Operations** toolbar or select the **File | Save** menu item to save the file. When you select the **File | Save As** menu item, the **Save As** dialog box opens. Type the desired name in the **File Name** box and click **Save**.
- Select the **File | Close** menu item to close the current file without closing Inspection Planner.

Creating a New Inspection Plan

To create a new Inspection Plan, follow this procedure:

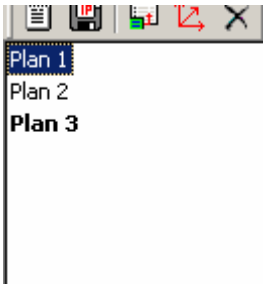
1. Open the **Plan** dialog box by doing one of the following:

   - Click **New Plan** from the **Tools** toolbar
   - Select the **Tools | New Plan** menu item.
   - Right-click any existing plan in the Plan list and select the **Insert 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 and then clicking the plan name again to activate an edit box with cursor. This can also be done by right-clicking the plan name and selecting the Rename option from the shortcut menu. Type the needed name and press **Enter**.

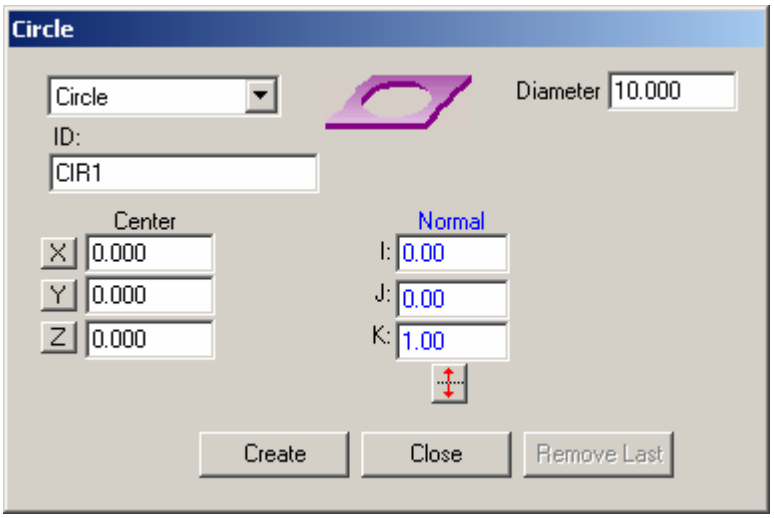
New Inspection Plans are embedded in the IP CAD file.

Creating Auto Geometry Features

Using the **Auto Features Geometry** toolbar or the **Features** menu you can create and add features to your Inspection Plan.

If a plan does not exist, you can create a feature and IP will automatically create a new plan for you. Otherwise, select the needed Inspection Plan from the plan list and begin creating features by doing the following:

1. From the **Auto Geometry Features** toolbar or the **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

Inspection Planner supports the following features:

	Surface Point		Circle		Polygon
	Edge Point		Ellipse		Cylinder
	Angle Point		Square Slot		Cone
	Corner Point		Round Slot		Sphere
	Line		Notch Slot	ED	Edge Scan
	Plane		Flush & Gap	3D	3D Scan

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 Auto 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** or **Create** when you are done. Click **Remove Last** to remove that last feature that was created via selection if the result is not acceptable.

The Auto Feature Geometry Dialog Box

Depending on the feature that you have selected, the "Feature" dialog box will display the option that are needed for that feature. The available options are:

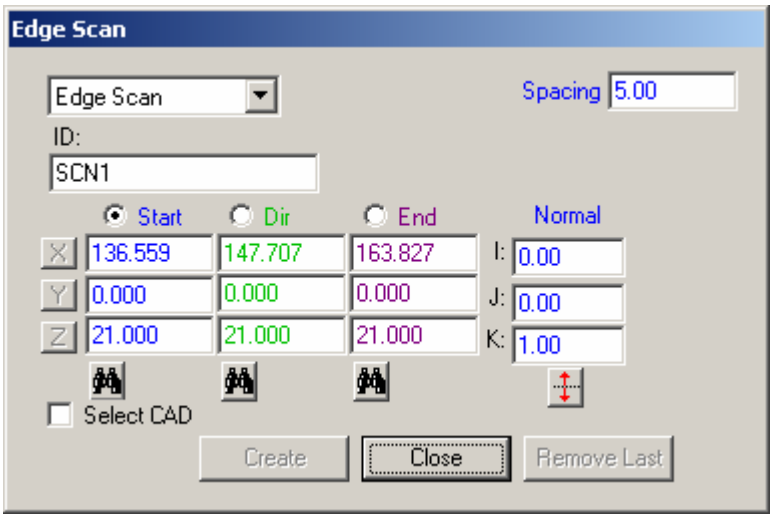
- **Feature Type:** Select the feature type from the drop-down list.
- **ID:** Provides the name for the feature that will be created based on the selected feature type.
- **XYZ:** These values define the Point, Center, Start or End positions for the Auto Feature.
- **IJK:** These values define the Surface, Edge, Normal and Angle Vectors for the Auto Feature.
- The **Find Nearest CAD Element** icon tells Inspection Planner to find the nearest CAD element in the Graphics 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 **XYX** axis buttons, only that value will be adjusted to the nearest CAD element.
- The **Find Vector** icon will pierce all surfaces along the XYZ point and IJK vector looking for the closest point. The surface normal vector will be displayed as the IJK NOM VEC but the XYZ values will not change. Only available for Surface Points.
- 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.

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. See the "Performing a Perimeter Advanced Scan" topic in the Core PC-DMIS documentation.

To Create a Edge Scan Auto Geometry Feature:

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

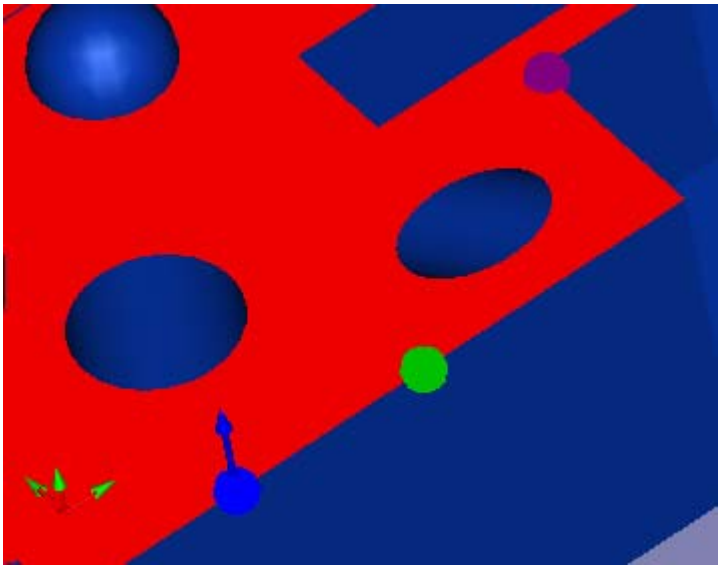


Edge Scan dialog box

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. 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** radio button 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

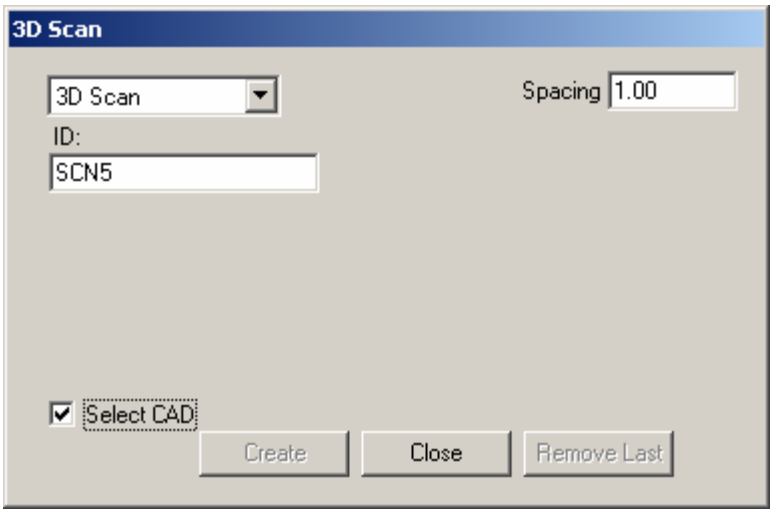
5. Click the **Find Nearest CAD Element** button to find the nearest CAD element in the Graphics Display window based on the XYZ location and any selected axis (or axes). This is especially useful when you have typed the XYZ values and need to update these values with the closest edge point on the CAD.
6. Click the **Flip Vector** button to flip the Normal vector.
7. Select the **Select CAD** option. When this option selected, the other parameters in the dialog box become unavailable.
8. 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.
9. Click **Create** to add the Edge Scan feature to the Inspection Plan, or click **Close** to close the dialog box without saving.
10. 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. See the "Performing a Patch Advanced Scan" topic in the Core PC-DMIS documentation.

To Create a 3D Scan Auto Geometry Feature:

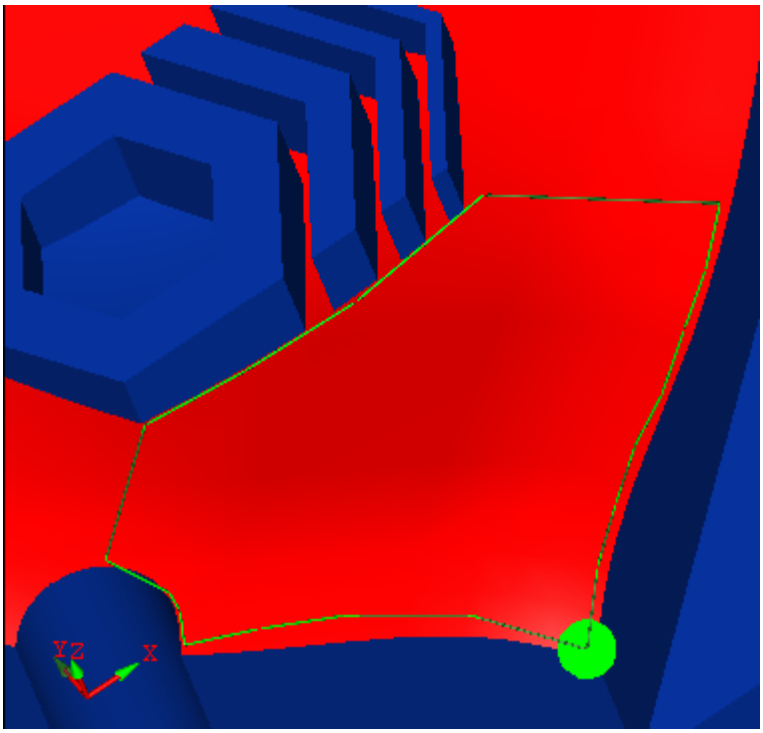
1. Click **3D Scan** from the **Auto Geometry Features** toolbar or select the **Features | 3D Scan** menu item. The **3D Scan** dialog box appears.



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.

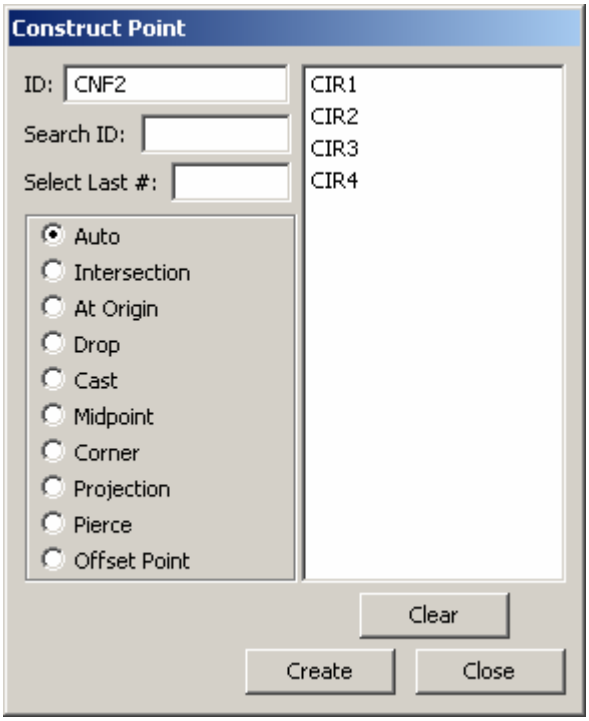
7. Click **Create** to add the 3D Scan feature to the Inspection Plan, or click **Close** to close the dialog box without saving.
8. Click **Remove Last** to remove the 3D Scan feature that was last added to the Inspection Plan.

Constructing Features

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

If a plan does not exist, you can create a constructed feature and IP will automatically create a new plan for you. Otherwise, select the needed Inspection Plan from the plan list and begin constructing features by doing the following:

1. From the **Constructed Features** toolbar or the **Constructed** 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

Inspection Planner supports the following constructed features:

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

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 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. 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.

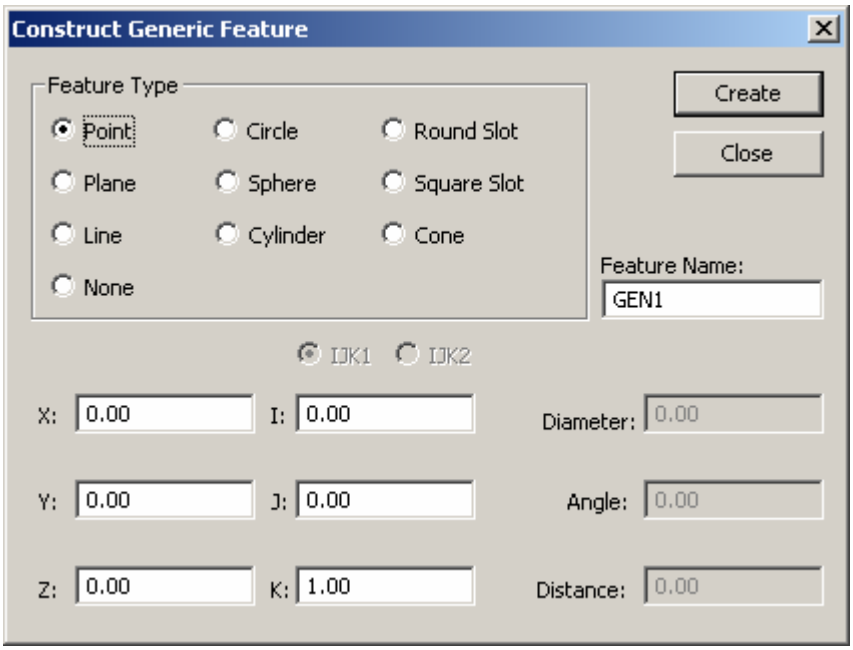
Note: When you create a **Constructed Feature Set** a point is created at the average of all the input centroids with the specified ID.

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 **Constructed | 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 **XYX** 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 -** Inspection Planner determines the best operation for the constructed feature based on the selected features. The is 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, Inspection 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.

Adding Dimensions to Features

Inspection Planner allows you create Dimensions for any feature in the Inspection Plan. The available dimension types are:

Location Dimension - See "Creating a Location Dimension".

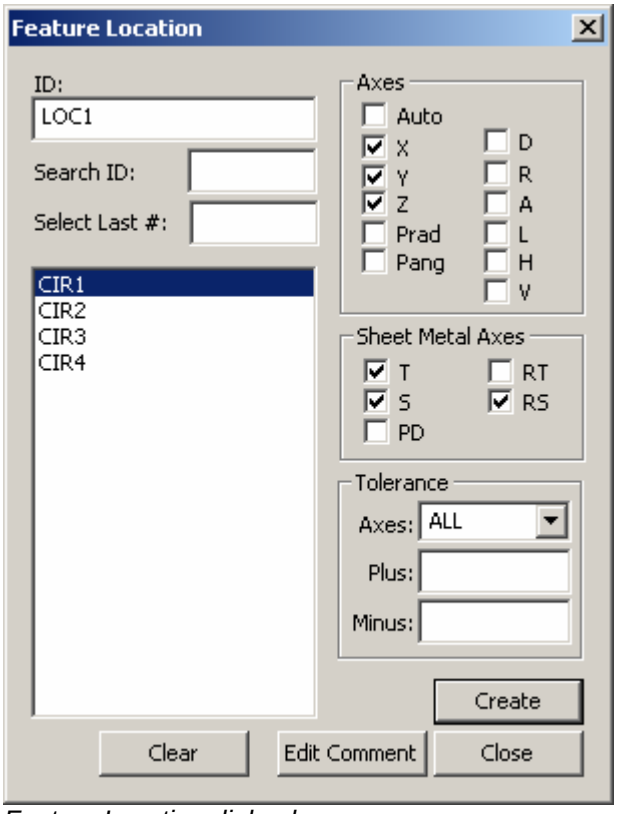
Distance Dimension - See "Creating a Distance Dimension".

Angle Dimension - See "Creating an Angle Dimension".

Creating a Location Dimension

To Create a Location Dimension:

1. Click **Location Dimension** from the **Dimensions** toolbar or select the **Dimension | Location** menu item. This opens the **Feature Location** dialog box.



Feature Location 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.

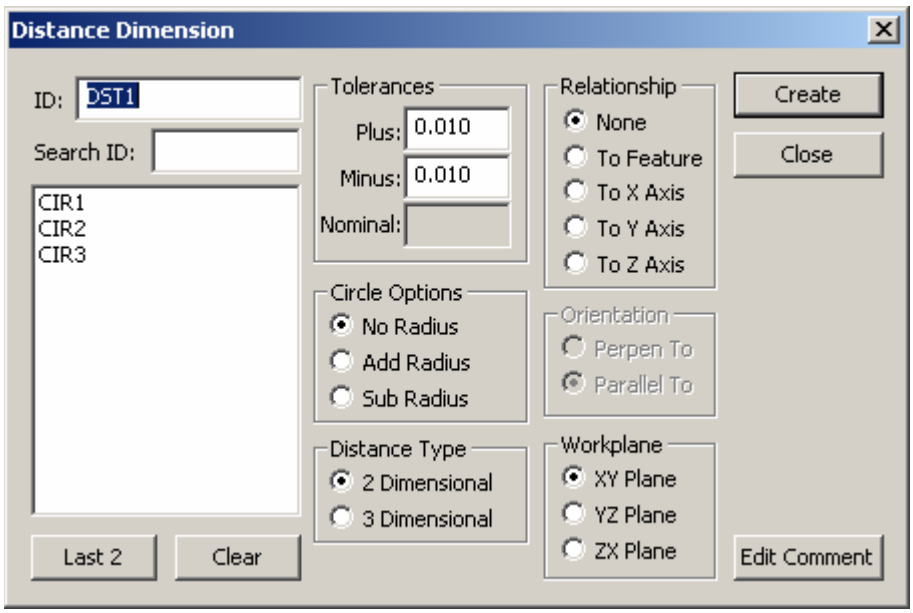
4. Select the appropriate **Axes** and **Sheet Metal Axes** for the selected feature(s). If you select Auto, Inspection Planner will automatically select the default **Axes** and **Sheet Metal Axes**. For example, if an IP point is selected, then the **X**, **Y**, **Z**, and **T** (vector) axes will be selected.
5. Provide the **Tolerance** values for each axis selected, by selecting the axis from the **Axes** drop-down box and typing the **Plus** and **Minus** tolerance values.

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 Dimension** from the **Dimensions** toolbar or select the **Dimension | Distance** menu item. This opens the **Distance Dimension** dialog box.
2.



Distance Dimension dialog box

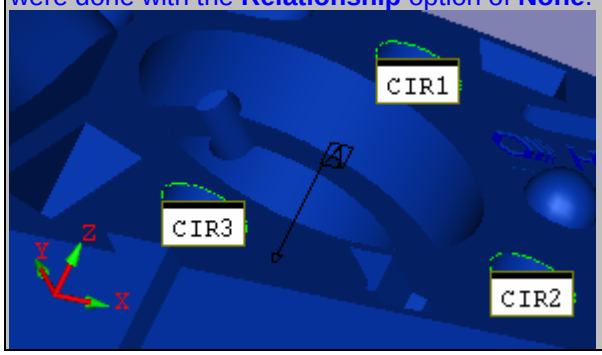
2. If needed, type an ID for this distance dimension in the **ID** box.
3. Select 2 features for which a distance dimension will be created by:

   - 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). You may also edit the **Nominal** distance by which the tolerance values will be evaluated.
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. The measured distance is also projected to the selected **Workplane**. 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. Select the **Workplane** to which the line measuring the distance between two features or one feature and the selected **Relationship** option is projected. This provides the nominal distance along that workplane for the selected features. You can choose from the **XY Plane**, **YZ Plane**, or **ZX Plane**. This option is only available when the **2 Dimensional Distance Type** is selected.

Workplane Examples: In the image below, the distance between **CIR1** and **CIR3** in the **XY Plane** would measure the direct line (diagonal) between the two features which results in **86mm**. However the same two circles in either the **YZ Plane** or **ZX Plane** would measure the line in the respective plane, resulting in a measurement of **60.811mm**. Additionally, the measurement between **CIR1** and **CIR2** in the **ZX Plane** would measure **0.0mm**, while in the **XY Plane** or **YZ plane** it would measure **60.811mm**. These examples were done with the **Relationship** option of **None**.

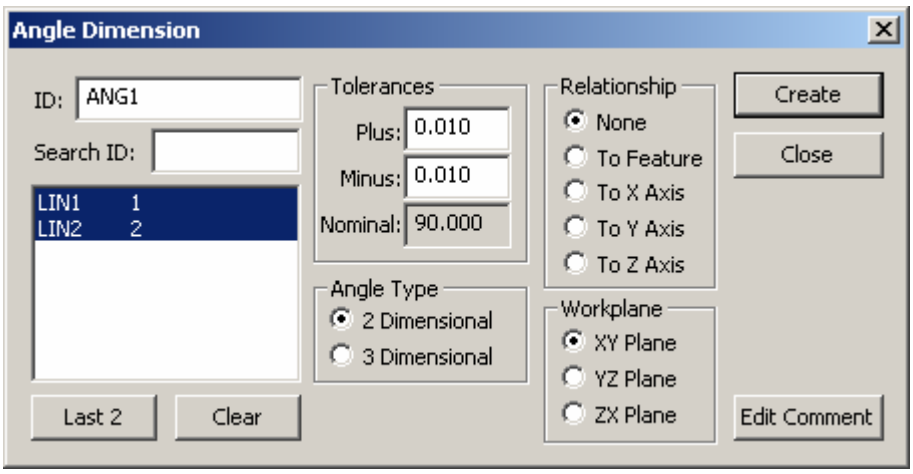


10. To add a comment to the distance dimension commands, click **Edit Comment** to open the **Edit Comments** dialog box.
11. Click **Create** to add the distance dimension to the Inspection Plan, or click **Close** to close the dialog box without saving.

Creating an Angle Dimension

To Create an Angle Dimension between two features:

1. Select **Angle Dimension** from the **Dimensions** toolbar or select the **Dimension | Angle** menu item. This opens the **Angle Dimension** dialog box.



Angle Dimension dialog box

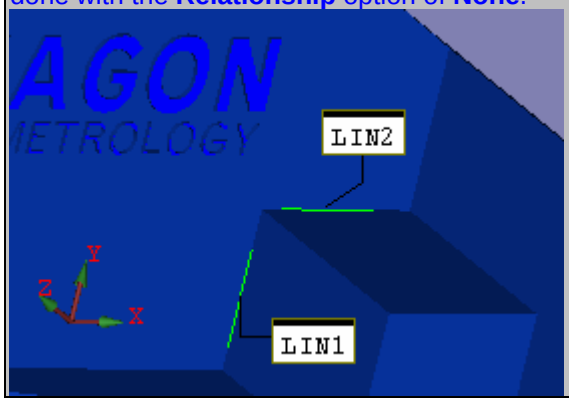
2. If needed, type an ID for this angle dimension in the **ID** box.
3. Select 2 features for which angle dimensions will be created by:

   - 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). You may also edit the **Nominal** angle by which the tolerance values will be evaluated.
5. Specify the **Angle Type**. The **2 Dimensional** option will measure in a two-dimensional plane relative to the specified feature vector or XYZ axis. The measured distance is also projected to the selected **Workplane**. For **2 Dimensional** options you must also specify the **Relationship** and **Workplane** options. **3 Dimensional** option will measure the best fit angle created between the two features.
6. 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.
7. Select the **Workplane** to which the measured angle between two features or one feature and the selected **Relationship** option is projected. This provides the nominal angle within the specified workplane for the selected features. You can choose from the **XY Plane**, **YZ Plane**, or **ZX Plane**. This option is only available when the **2 Dimensional Distance Type** is selected.

Workplane Example: In the image below, the angle between **LIN1** and **LIN2** in the **XY Plane** would measure and angle of **90** degrees. However the measured angle for the same two lines in either the **YZ Plane** or **ZX Plane** would be **0** degrees. This example was

done with the **Relationship** option of **None**.

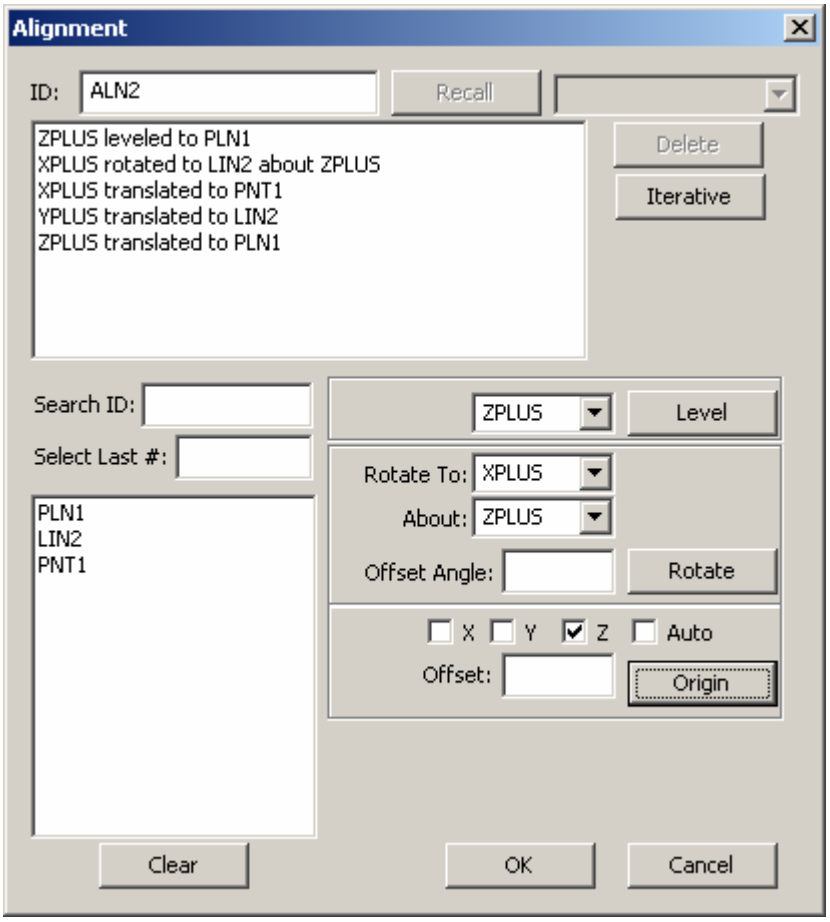


8. To add a comment to the angle dimension commands, click **Edit Comment** to open the **Edit Comments** dialog box.
9. Click **Create** to add the angle dimension to the Inspection Plan, or click **Close** to close the dialog box without saving.

Creating an Alignment

Once you have created features, you can create or modify alignments:

1. Click **Alignment** from the **Tools** toolbar or select the **Tools | 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.

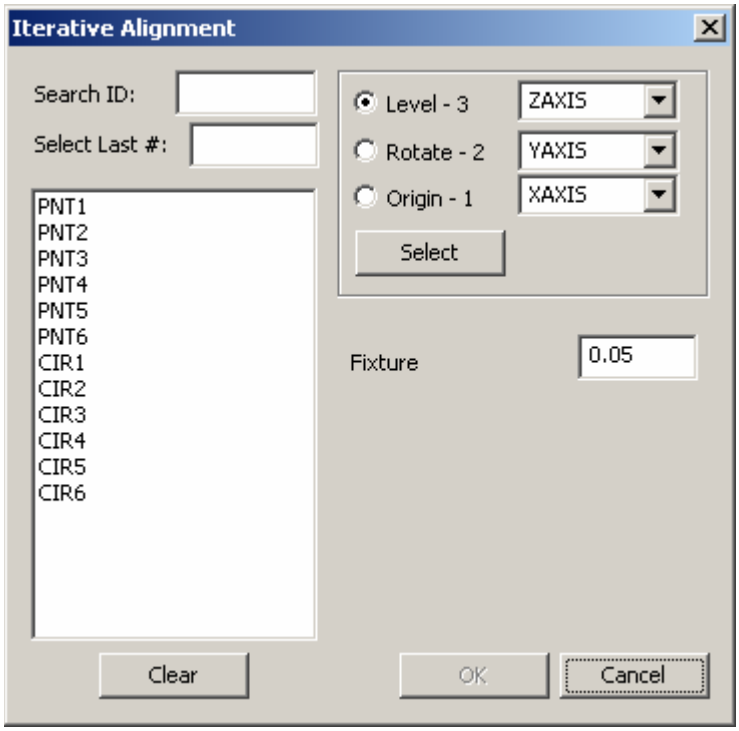
See the "Creating an Iterative Alignment" topic for Iterative Alignment information.

Creating an Iterative Alignment

Aside from standard alignments, capability exists to create Iterative (3-2-1)Alignments using Inspection 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.

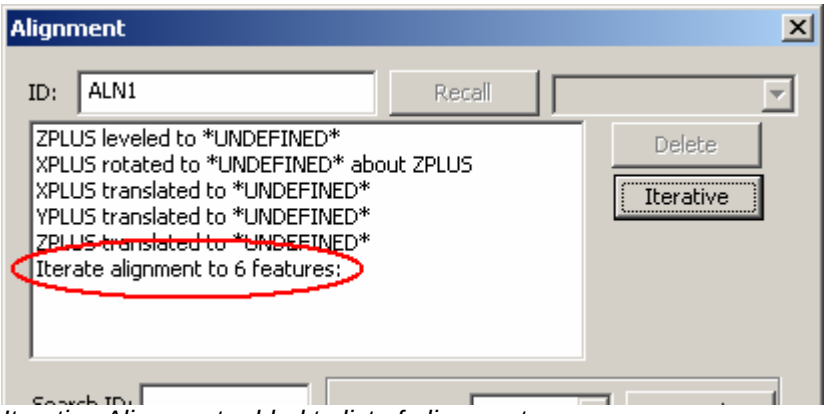
• Click **Clear** to clear the previously selected features.

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:



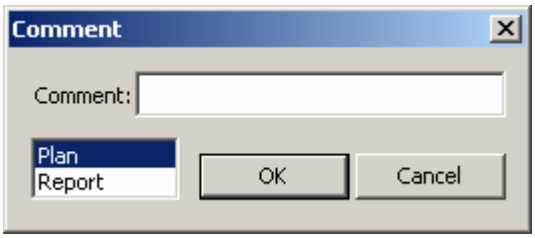
Iterative Alignment added to list of alignments

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 instructions to operators, programmer notes, or notes that will show up on your Inspection Plan reports.

1. Click **Comment** from the **Tools** toolbar or select the **Tools | 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 will display during execution and require the operator to click **OK**, to continue. **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.

Deleting Inspection Plan Elements

Select the 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 **Tools** toolbar.
- Select the **Tools | 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.

Exporting an Inspection Plan

It is not necessary to export an Inspection Plan (IP file) if you plan to import the IP CAD file into PC-DMIS. The IP CAD file contains any embedded Inspection Plans. Otherwise, exporting a plan for use in PC-DMIS is another method to make Inspection Plans available for PC-DMIS.

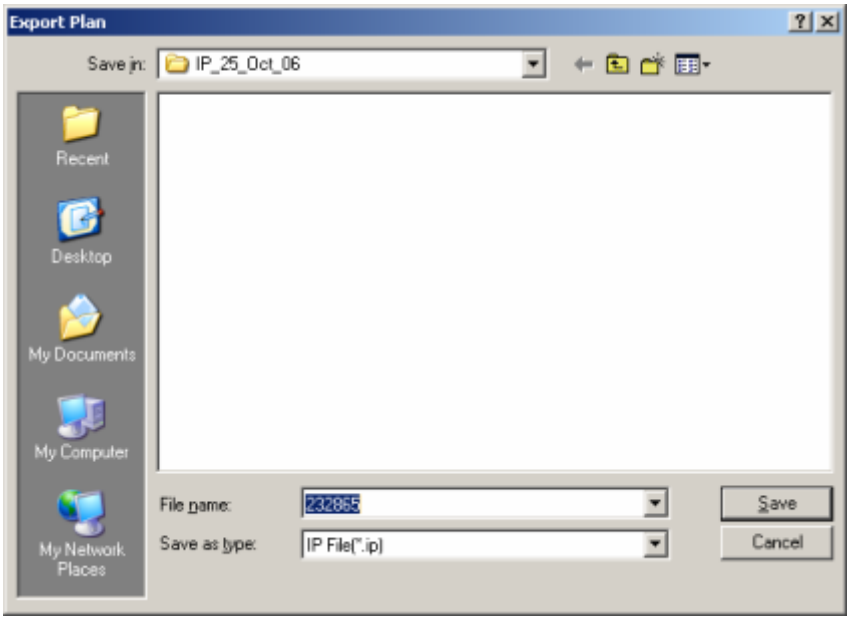
To export a new PC-DMIS Inspection Plan, follow this procedure:

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

    - Click **Export Plan** from the **Tool**s toolbar

- Select the **Tools | Export Plan** menu item
- Right-click the needed plan and select the **Export Plan** option from the shortcut menu

The **Export Plan** dialog box appears.

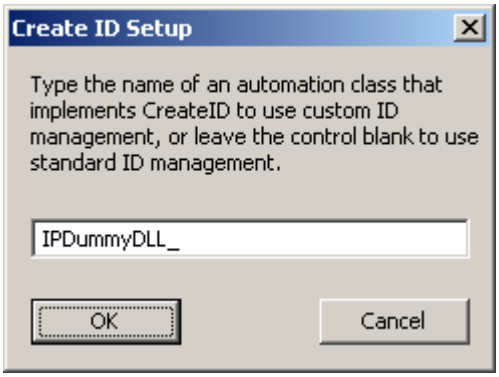


Export Plan dialog box

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

Changing the ID Setup

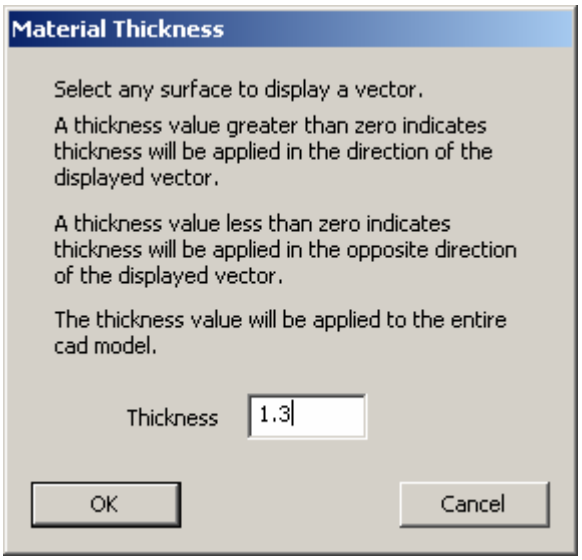
The **Create ID Setup** dialog box allows you to provide the name of the automation class that will be used for the creation of IDs within Inspection Planner.



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 **Tools** toolbar. The **Material Thickness** dialog box appears.



2. Select any surface to display the surface vector. If there is an associated thickness with the surface vector the **Thickness** value will be provided.
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

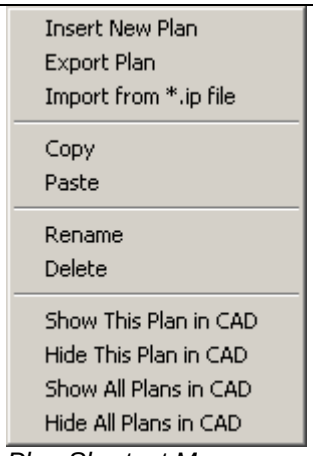
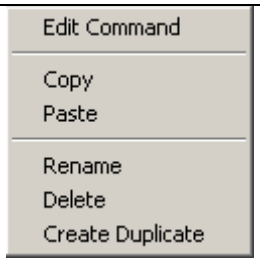
You can customize the toolbars that are displayed by toggling them on or off by selecting the toolbar name from the **View | Toolbars** menu. Menu items with a check mark are displayed, otherwise they are not displayed. You can also close undocked toolbars by clicking the **X** in the corner of the toolbar.

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 List window. Undocked (floating) toolbars can also be moved to any position over the Graphics Display Window.

Select the **View | Status Bar** menu item to toggle the display of the Status Bar, which displays along the bottom of the Inspection Planner application window.

Plan List Shortcut Menus

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*	• **Insert New Plan** - Creates a new Inspection Plan at the bottom of the Plan list. • **Export Plan** - Exports the selected plan to an IP file. • **Import from *.ip file** - Command not currently supported. • **Copy** - Copies all of the command items for the selected plan to the clipboard. • **Paste** - Pastes all of the commands from the 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 Graphics Display Window. • **Hide This Plan in CAD** - Hides elements from the selected plan in the Graphics Display Window. • **Show All Plans in CAD** - Displays elements from all plans in the Graphics Display Window. • **Hide All Plans in CAD** - Hides elements from all plans in the Graphics Display Window.
Plan Command Shortcut Menu	
 *Plan Command Shortcut Menu appears when you click on a command item.*	• **Edit Command** - Displays the dialog box for the selected command. Make any needed changes and click apply or create to save you changes. • **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. • **Create Duplicate** - Creates a duplicate of the selected command with the name of "Copy of <command name>" in the same plan.

Graphics Display Window

There are many tools that are available to adjust the way that CAD graphics are displayed, selected and edited in the Graphics Display Window. These tools are similar or the same as tools found in PC-DMIS. This document provides a basic overview for the use of these tools. See the core PC-DMIS document for more information on the tools described in the following topics:

- Changing the Graphic Mode
- Changing the Graphic View
- Editing CAD Vectors
- Viewing CAD Information
- Editing Screen Colors
- Capturing the Graphics Display Window
- Setting OpenGL Options

Changing the Graphic Mode

The **Graphic Modes** toolbar allows you to change the attributes and functionality of the **Graphics Display Window**. These functions are discussed in greater detail in the "Graphics Modes Toolbar" topic of the core PC-DMIS document.

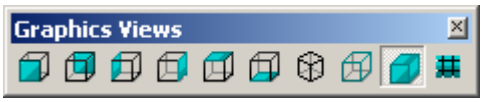
Graphics Modes Toolbar

Icon	Function
	View Setup - Opens the **View Setup** dialog box which allows you to change the layout for multiple panels of the Graphics Display Window and the viewing angles for each panel. You can also create levels and select the feature that are located on each level. The display of rulers, a 3D grid, surface or wire frame are other functions of this dialog box. Some options available here are also available on the **Graphics View** toolbar. You may also select the **View | Graphics Display Window | View Setup** menu item.
	Scale to Fit - Centers the display of the CAD image in the Graphics Display Window both vertically and horizontally. You may also select the **View | Graphics Display Window | Scale To Fit** menu item.
	Repaint to Screen - Refreshes the display of the Graphics Display Window to include all data that should be displayed. You may also select the **View | Graphics Display Window | Repaint The Screen** menu item.
	Curve Mode - This mode allows you to select CAD wire frame data from the Graphics Display Window. You may also select the **View | Graphics Display Window | Modes | Curve** menu item.
	Surface Mode - This mode allows you to select CAD surface data from the Graphics Display Window. You may also select the **View | Graphics Display Window | Modes | Surface** menu item.
	Translate Mode - This mode allows you to zoom in (click in bottom half) and out (click in top half) with right-clicks in the Graphics Display Window. Click and drag the to move the display to the desired location. You may also select the **View | Graphics Display Window | Modes | Translate** menu item.
	2D Rotate Mode - This mode allows you to rotate around the center of the current viewing angle by right-clicking and dragging in the Graphics Display Window. You

	may also select the **View | Graphics Display Window | Modes | 2D Rotate** menu item.
	3D Rotate Mode - This mode allows you to rotate the CAD relative to the center of the CAD elements by right-clicking and dragging in the Graphics Display Window. You may also select the **View | Graphics Display Window | Modes | 3D Rotate** menu item.
	Assembly - Opens the **CAD Assembly** dialog box which allows you to control the visibility of the elements that make up the assembly. You can also access the **CAD Transform** dialog box to reposition the CAD, Delete elements of the assembly, or access the **Coordinate System** dialog box. You may also select the **View | Graphics Display Window | CAD Assembly** menu item.
	Coordinate System - Opens the **Coordinate System** dialog box which allows you to **Create**, **Rename**, **Delete** or **Replace** the current coordinate system. You may also select the **View | Graphics Display Window | CAD Coordinate System** menu item.
	CAD Lighting - Opens the **Lighting and materials** dialog box which allows you to change Graphics Display Window lighting, transparency, CAD texture mapping, material colors, and clipping planes. You may also select the **View | Graphics Display Window | Lighting, Materials** menu item.
	Rotation Options - Opens the Rotate Options dialog box which allows you to set options to optimize the display of CAD elements when they are rotated or moved. You may also select the **View | Graphics Display Window | Rotate 2D/3D Options** menu item.

Changing the Graphic View

The **Graphic View** toolbar allows you to quickly change how your part is displayed inside the Graphics Display window.



Graphics Views Toolbar

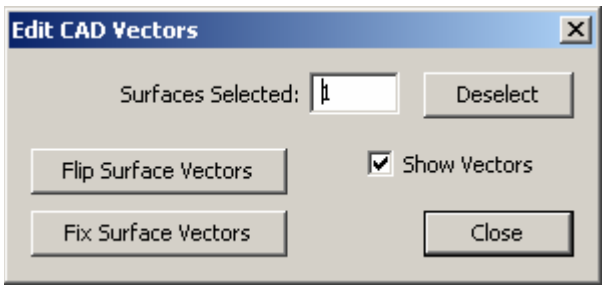
Icon	Displays	Icon	Displays	Icon	Displays
	X+ View		X- View		ISO View
	Y+ View		Y- View		Wireframe
	Z+ View		Z- View		Surface/Solid
					3D Grid

You may also toggle the display for surface, wire frame, and window grid under the **View | Graphics Display Window** menu and the viewing angles under the **View | Graphics Display Window | Views** menu.

Editing CAD Vectors

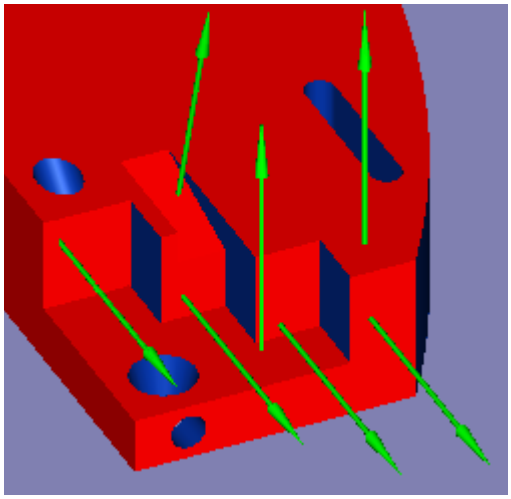
To edit the vectors for surfaces of your CAD:

1. Click **CAD Vectors** from the **CAD Vectors** toolbar or select the **View | Graphics Display Window | 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.

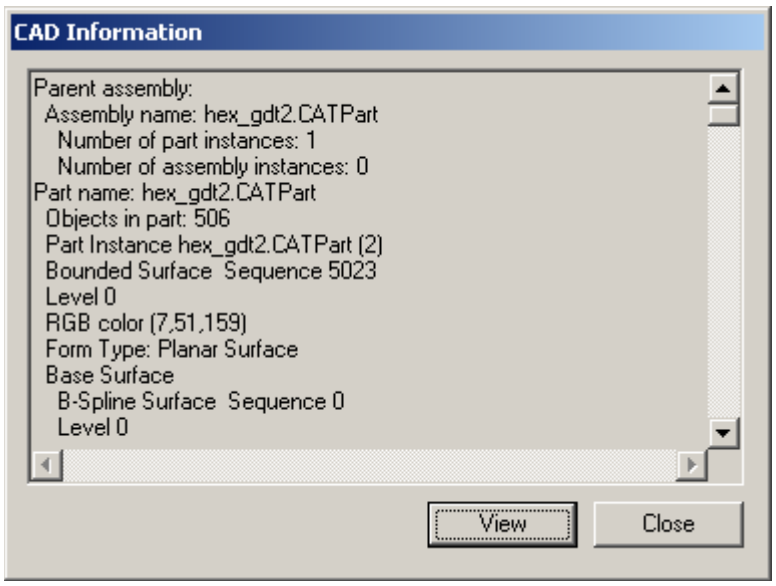
Viewing CAD Nominal Information

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 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 **Show Noms** from the **Menu Icons 1** toolbar or select the **View | Graphics Display Window | CAD Info** menu item. The **CAD Information** dialog box appears.



Cad Information dialog box

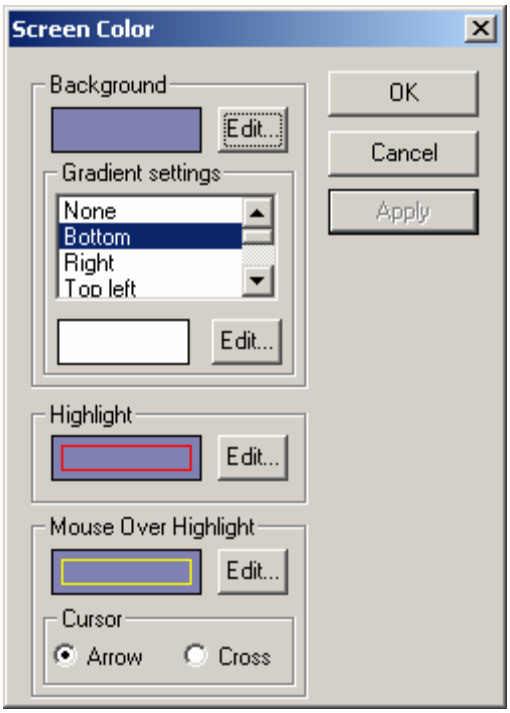
2. Select any CAD geometry element and the element information is displayed in the **CAD Information** dialog box.
3. Click **View** and the CAD element will flash on the screen.
4. Click **Close** when you are finished.

Editing Screen Colors

The **Screen Color** dialog box allows you to determine the colors to display within the Graphics 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 **Menu Icons 1** toolbar or select the **View | Graphics Display Window | Screen Color** menu item. The **Screen Color** dialog box appears.



Screen Color dialog box

2. Edit the following color values by clicking **Edit** and selecting the desired color from the **Color** dialog box. Click **Close** to return to the **Screen Color**.

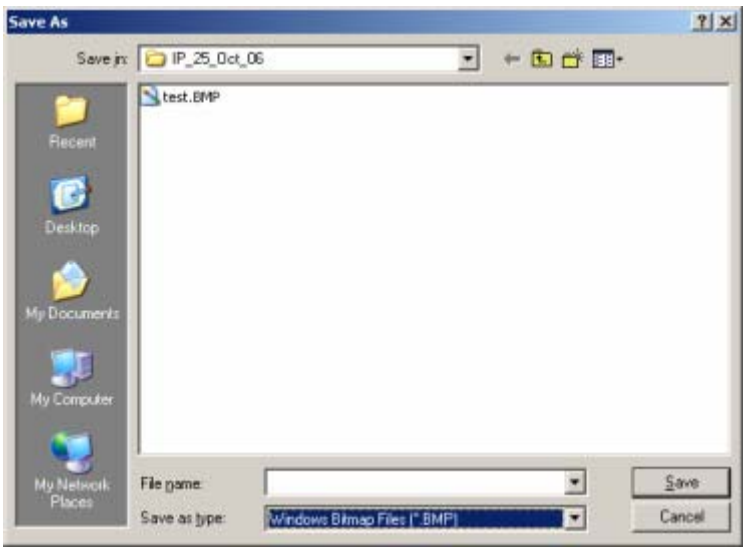
   - **Background** - Provides the background color for the screen or the primary background color when a gradient is specified.
   - **Gradient Settings** - When you set the gradient to something other than "None" in the list an additional **Edit...** button appears. This button allows you to define a secondary background gradient color. Inspection 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 Graphics 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. Click the **Apply** button to save the changes you have made and continue working in the Screen Color dialog box.
4. When finished, click the **OK** button, or click **Cancel** to undo any changes.

Capturing the Graphics Display Window

Inspection Planner allows you to capture an image the current view from the Graphics Display Window to the clipboard or a bitmap file.

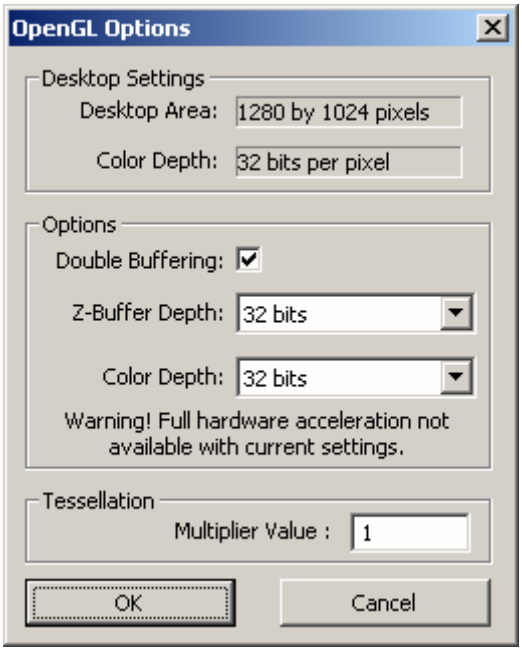
- Click **Capture To Clipboard** from the **Menu Icons 1** toolbar or select the **View | Graphics Display Window | Screen Capture To | Clipboard** menu item to capture the Graphics Display Window to the clipboard. This image can then be pasted into any windows application where the paste option is available.
- Click **Capture To File** from the **Menu Icons 1** toolbar or select the **View | Graphics Display Window | 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.

Setting OpenGL Options

The **OpenGL Options** dialog box allows you to change the options that affect display of the model in "Surface Mode". Click **OpenGL Settings** from the **Menu Icons 1** toolbar or select the **View | Graphics Display Window | OpenGL** menu item to open the **OpenGL Options** dialog box.



OpenGL Options dialog box

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

A note on 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 Graphics Display window), use a 1.6 resolution ratio to resolve this problem.

- **Options** - Controls the display characteristics of the model in solid view mode. Some combinations of these options may not be available because of limitations of your graphics card. Also, depending on how your graphics card supports hardware acceleration, some combinations of options may result in lowered graphics performance. If the current option settings will result in lowered performance, a warning message will appear at the bottom of the dialog box.
- **Tessellation -** Controls the drawn image by setting a tessellation multiplier in the **Multiplier Value** box. Inspection 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.

Using Inspection Plans in PC-DMIS

The following topics describe how to use the functionality added to PC-DMIS to automatically create a part program from your Inspection Plan. To create your part program you need to do the following:

- Define Inspection Planner Parameters and Rules
- Import the Inspection Plan you exported from your CAD package.
- Perform an optimization step on the imported plan,
- Ensure that the probe won't collide with the part by inserting proper move commands.

Once you have completed these simple tasks, you can then execute your imported inspection plan as a newly created PC-DMIS part program.

If you have problems using the Inspection Planner commands inside PC-DMIS, ensure that there is an *InsPlan.dll* file located in the directory where you installed PC-DMIS.

Inspection Planner Default File (IPD)

You must have an Inspection Planner Default file, or (IPD) to import an Inspection Plan into PC-DMIS. The "Inspection Planner Parameters and Rules" topic provides more information on this file.

The **IP Measure** option must be enabled on your portlock to use the Inspection Planner capabilities that are discussed in this section.

Inspection Planner Parameters and Rules

The **Inspection Planner Default (IPD)** file is used to define the parameters for **Inspection 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. You can also specify how you want to display comments and features.

The **Inspection 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.

Before importing an Inspection Plan from an IP CAD file or an IP file, you might want to set-up these files to apply to imported Inspection Plans.

Creating and Modifying an Inspection 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 | Preferences | Inspection Plan Defaults** menu option. The Modify Default Parameters dialog box appears.

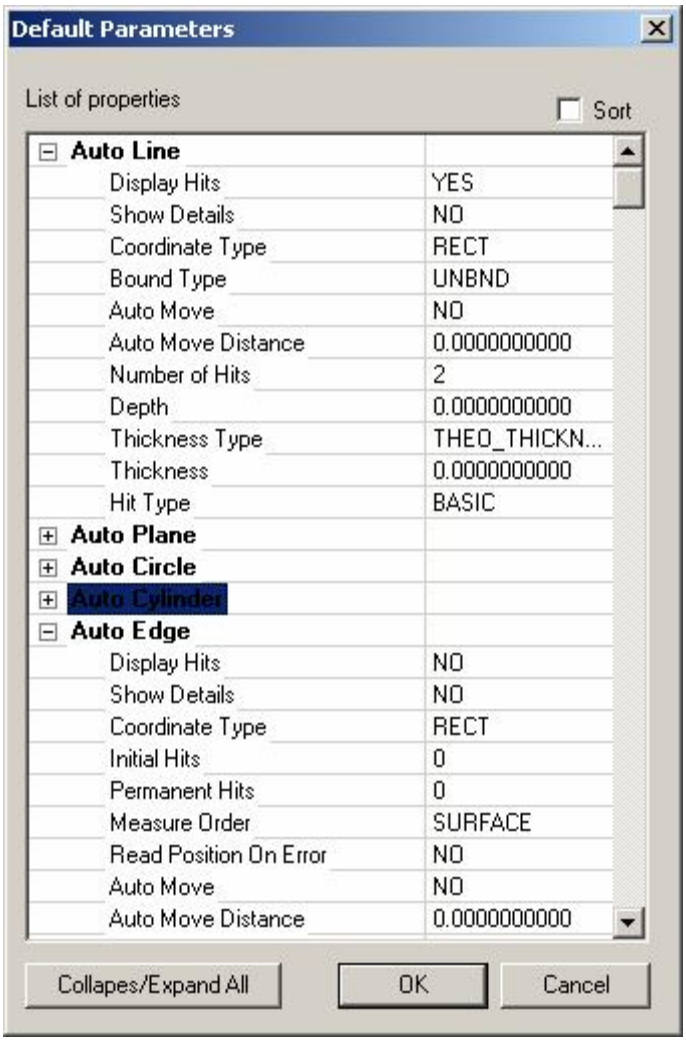
The **Modify Default Parameters** dialog box is password protected if password protection has been enabled for other settings in PC-DMIS. See the "Setup Options: General Tab" topic in the core PC-DMIS reference manual for more information.

Modify Default Parameters – Inspection 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 the **Open** button 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 IP

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. Click the **Collapse/Expand All** button to collapse or expand the **List of properties**.
8. Click **OK** to open the **Save As** dialog box, or **Cancel** to abort any changes that have been made.
9. Click **Save** to save the IPD file.
10. Click **Set as Default** if you want to make the file listed in the **Default Parameters File Name** box the default file.
11. Click **Exit** when finished.

Inspection 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 Commar
' 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
```

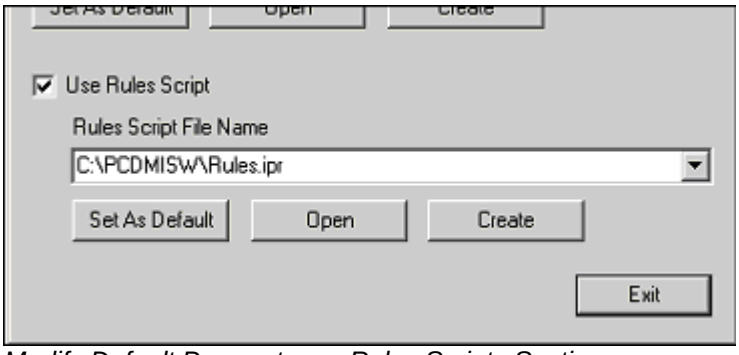
Basic Script Editor containing sample code for IP rules

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 Inspection Planner Rules (IPR) script, follow this procedure:

1. Select the **Edit | Preferences | Inspection Plan Defaults** 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 reference manual 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

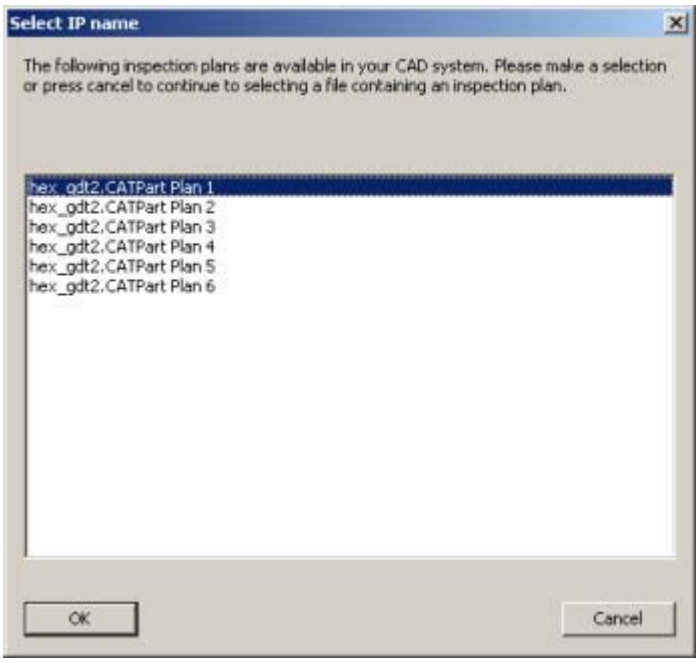
1. Select the **Edit | Preferences | Inspection Plan Defaults** menu option. The **Modify Default Parameters** dialog box appears.
2. From the **Rules Script File Name** drop-down select or browse to the needed file.
3. Click **Set as Default** to make the file listed in the **Rules Script File Name**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

Importing the Inspection Plan

You can import the Inspection Plan and any rules by either importing a IP CAD file that has the Inspection Plan embedded in it, or by importing a previously exported Inspection Plan.

To import an embedded Inspection Plan from an IP CAD file:

1. Select **File | Import | CAD**. The **Open** dialog box appears.
2. Browse to and select the needed IP CAD file and click **Import**. PC-DMIS imports the IP CAD file.
3. Select **File | Import | Inspection Plan**. The **Select IP Name** dialog box appears.



Select IP Name dialog box

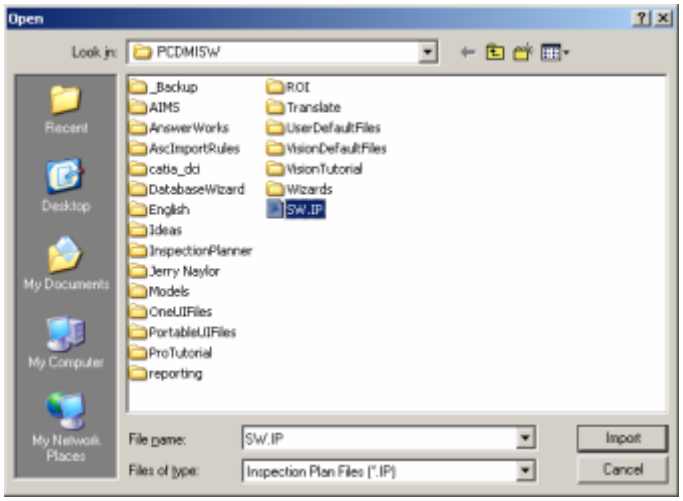
4. Select the needed Inspection Plan and click **OK** or click **Cancel** to browse to and select an exported Inspection Plan via the **Open** dialog box.
5. Follow any on-screen instructions.

- Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Inspection Planner Parameters and Rules" topic for more information.
- If the insertion point is not at the end of the part program, you will be prompted to move the cursor to the end of the part program before continuing.
- Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See the "Performing the Path Optimization" topic for more information. Review the "Path Optimization Summary" and click **OK**.
- Specify the options for "Automatically Inserting Move Commands" and click **OK** to complete this process or click **Cancel** to skip this step.

Your imported Inspection Plan will be proceeded by a start comment that provides basic information for the imported Inspection Plan (date/time, IP file name with path, IPD file name and path). An end comment is also added to indicate the end of the translated Inspection Plan.

To import an Inspection Plan from an exported text file:

1. Select **File | Import | Inspection Plan**. An **Open** dialog box appears letting you select a file with an ".IP" filename extension.



Open dialog box

2. Browse to the directory containing your Inspection Plan.
3. Select the Inspection Plan, and click **Import**.
4. Follow any on-screen instructions.

- Select the **IPD** file that will be applied to the imported Inspection Plan if you have not previously selected it. See the "Inspection Planner Parameters and Rules" topic for more information.
- If the insertion point is not at the end of the part program, you will be prompted to move the cursor to the end of the part program before continuing.
- Specify the **Optimize Path** options and click **OK**, or click **Skip** to ignore this step. See the "Performing the Path Optimization" topic for more information. Review the "Path Optimization Summary" and click **OK**.
- Specify the options for "Automatically Inserting Move Commands" and click **OK** to complete this process or click **Cancel** to skip this step.

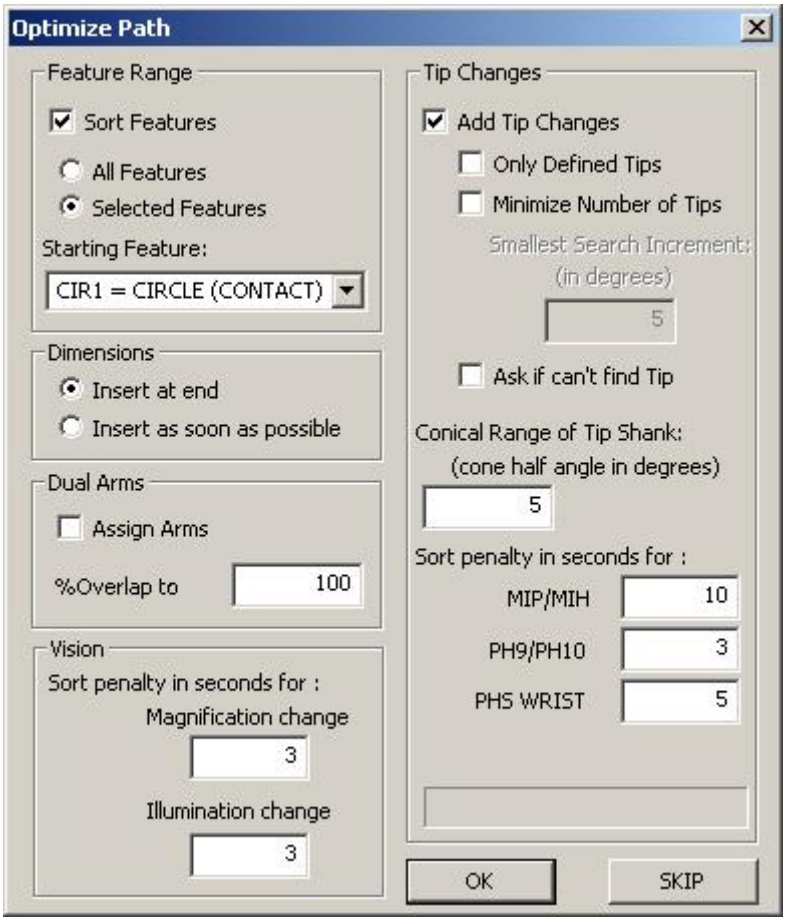
Your imported Inspection Plan will be proceeded by a start comment that provides basic information for the imported Inspection Plan (date/time, IP file name with path, IPD file name and path). An end comment is also added to indicate the end of the translated Inspection Plan.

Performing the Path Optimization

Once PC-DMIS finishes importing your inspection plan, you will see that the PC-DMIS edit window contains the commands needed to measure the desired features. Next, you need to optimize the order PC-DMIS takes to measure the features and what tip angles to use and when. This is called a "path optimization".

To perform a path optimization:

1. Select **Edit | Optimize Path.** the Optimize Path dialog box appears.



Optimize Path dialog box

2. Determine what features you want to optimize. you can choose the **All Features** option button or the **Selected Features** option from the **Features Range** area. ***All Features*** sorts all the features, **Selected Features** only sorts those features selected in the edit window.

The **Selected Features** option only becomes enabled if you select multiple features from the edit window *before* accessing the **Optimize Path** dialog box.

3. Choose a feature from the **Starting Feature** drop-down list. Path optimization will start from this selected feature.
4. Decide when to insert dimensions from your inspection plan into the part program by selecting the appropriate options from the **Dimensions** area. You can insert dimensions at the end of all the measured features by selecting **Insert at end** or as soon as possible after each measured feature by selecting **Insert as soon as possible**.
5. Determine whether or not you want to allow PC-DMIS to insert tip changes. If so, select the **Add Tip Changes** check box. The items in the **Tip Changes** area become enabled for selection and you should follow the bullet list here:

- Select the **Only Defined Tips** check box if you want PC-DMIS to assign only defined tips to features. If an appropriate tip can not be found to measure a feature, then the feature will be assigned T?A?B?. This feature will be added to the list of **Features without tips** which is displayed to the user in the summary dialog at the completion of

the Optimize Path process. If you clear this check box, PC-DMIS will also use undefined tips. Prior to execution, PC-DMIS will require that you calibrate any undefined tips.

- Select the **Minimize Number of Tips** check box if you want PC-DMIS to use a minimal amount of tips to measure the features in your part program. PC-DMIS will only change the tip's angle when absolutely necessary. A progress bar at the bottom of the dialog box appears when you select this check box.
- Select the **Ask if can't find Tip** check box if you want PC-DMIS to prompt you when it cannot determine the optimal tip for a given feature. This will allow you to manually specify the most appropriate tip.
- Type an angle value into the **Conical Range of Tip Shank** box. This determines the conical tolerance for added tip changes. This allows PC-DMIS to utilize a tip that has a shank vector that is within the tolerance of the optimal vector angle for a given feature. For features such as points, lines, and planes there is a built in conical range of 90 degrees since a plane with a vector of 0, 0, 1 could be measured by t1a0b0 or t1a90b0. In the case of these features, PC-DMIS adds the user entered value to the built-in range. The proposed tip angle is also checked by running collision detection (silently) to determine if the tip will actually work.
- **Sort penalty in seconds for: MIP/MIH, PH9/PH10, PHS WRIST** – These values in seconds are an approximation of the time taken by a CMM to change tips for various categories of probes. The default penalties mean that all features measured with a given tip will be grouped together and sorted within that tip group. For example, If the penalty were set to zero, this would allow the Optimize Path process to sort features solely based on the distance traveled between features while completely ignoring tip changes.
- **Vision / Sort penalty in seconds for: Magnification Change, Illumination Change** – These values in seconds are only an approximation of the time taken by a Vision Probe to change magnification or Illumination for a given feature. The default penalties mean that all features that use the same magnification or illumination will be grouped together and sorted within that group. For example, If the penalty were set to zero, this would allow the Optimize Path process to sort features solely based on the distance traveled between features while completely magnification or illumination changes.

6. Click **OK** to perform the path optimization; PC-DMIS does the following:

- Takes the tip vector and the cone tolerance and determines the best tip angle for each feature. For example, the optimal tip angle for a cylinder will be based on the cylinder vector, which may or may not correspond to an exact tip angle.
- Sorts the features by the tips used to measure them.
- Determines the best path for the probe to take based on distances between features and the tip change.
- Adds dimensions either at the end of the program or after each feature.
- Adds constructed features as soon as possible.

See the "Dual Arm Path Optimization" topic for information on Dual Arm programs.

PC-DMIS will not sort across alignment or loadprobe commands. Additionally, PC-DMIS removes all MOVE/POINT commands, but preserves any motion commands or other parameters that may affect measurement, such as retract distance, prehit distance, etc.

Path Optimization ensures that features that have the RMeas option will not be measured prior to the features to which their measurement is relative.

Dual Arm Path Optimization

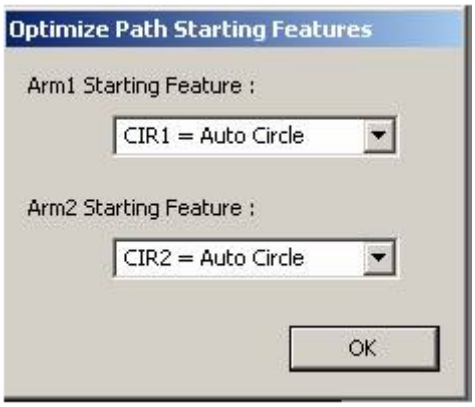
Path Optimization for dual arm programs are similar to single arm programs with a few exceptions. The **Dual Arms** area of the **Optimize Path** dialog box is enabled and the **Starting Feature** pull-down list is hidden. To complete Path Optimization for dual arms, follow the procedure described in "Performing the Path Optimization" and then complete the following steps:

1. Select the **Assign Arms** check box. The Optimize Path process will assign arms to each of the **Selected Features** (or **All Features**).

    - If all probe hits of a feature can be measured by Arm1, then it will be assigned to Arm1.
    - If Arm1 cannot measure a feature, but Arm2 can reach all of the probe hits that feature, then it is assigned to Arm2.
    - If neither arm can measure the feature, then Arm1 is assigned and the feature is added to the list of **Dual Arm Errors** / **Features that cannot be measured** which is displayed to the user in the summary dialog at the completion of the Optimize Path process.

Dual Arms are of the Optimize Path dialog box

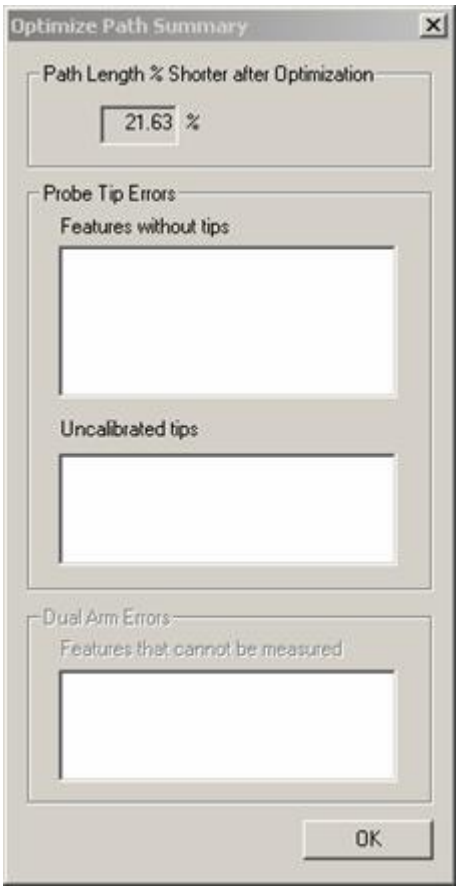
2. Specify the Overlap percentage in the **%Overlap to use.** This controls how the machine volume is divided between the two arms. At 0%, each arm would only be allowed to measure features that are within the half of the machine volume that is accessible to that arm. At 100%, Arm1 would be assigned to all the features.
3. Click **OK** to begin the Path Optimization process. This displays the **Optimize Path Starting Features** dialog box. This allows you to specify the starting feature for each arm



Optimize Path Starting Features dialog box

Path Optimization Summary

Once the Path optimization has completed, the **Optimize Path Summary** dialog box shows the results of the path optimization.



Optimize Path Summary dialog box

The dialog box reports the following information:

Path Length % Shorter after Optimization – This value refers to the percent reduction of the optimized path length compared to the normal path length. For dual arm machines, it gives a percentage for both arms.

Probe Tip Errors – Features that were not assigned to a probe tip are listed in the **Features without tips** list. You may need to define new probe tips to ensure that each feature has a tip that can measure it. The **Uncalibrated tips** list shows the uncalibrated probe tips that you plan to use to measure features in your part program. You may need to calibrate these tips before executing the part program.

Dual Arm Errors – Features that the program was unable to measure by either arm will be included in this list.

Void Detection

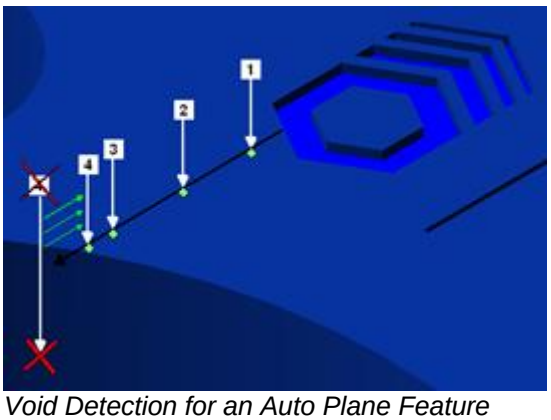
Void Detection prevents hits from being taken in empty spaces (such as holes). This functionality has been added to these Auto Features:

- Auto Vector Point
- Auto Surface Point
- Auto Edge Point
- Auto Line
- Auto Plane
- Auto Circle
- Auto Round Slot
- Auto Cylinder

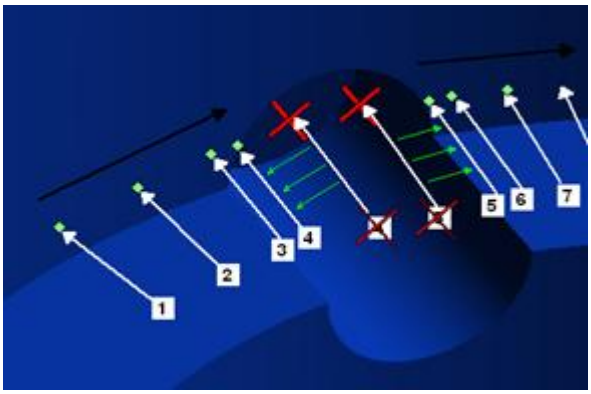
Without Void Detection, an auto feature may place hits in holes and other empty spaces in the CAD surface and the part.

PC-DMIS uses void detection if your hardware key (or portlock) has been programmed with the **IP measure** item. PC-DMIS automatically applies void detection to the generation of the probe hits when you create a supported auto feature by using the parameters from corresponding **Auto Feature** dialog box (such as **# hits**, **Start Angle** and **End Angle** for an auto circle). Any probe hits that would normally fall into a void are moved so that they reside on the surface of the selected feature, at least, a probe radius distance from any edge of that surface. Requested sample hits for Auto Circles or Auto Cylinders are also automatically generated with void detection.

The following examples show when Void Detection would be useful for accurately and safely measuring an Auto Feature. Hits denoted with an “X” are determined to be Voids and are corrected to the edge of the auto feature surface.



Void Detection for an Auto Plane Feature



Void Detection for an Auto Cylinder Feature

Automatically Inserting Move Commands

Once you import your part program and have finished Path Optimization, you should ensure that when the probe moves to measure your features, it will not collide with your part. Collision Avoidance is accomplished by inserting move commands. To do this, use PC-DMIS’s **Auto Insert Move** menu option to automatically create clearance moves for a range of features:

Note: Move commands can be placed between the range of features specified, but not prior to the first feature. A clear path is assumed to the probe start position and the first feature.

1. Select the **Operation | Graphics Display Window | Auto Insert Moves** menu option. The **Automatically create clearance moves** dialog box appears.

Automatically create clearance moves

2. Select the first feature for the range of features from the **Start** list.
3. Select the last feature for the range of features from the **End** list.
4. Type a value in the **Collision safety** box. This value provides a buffer zone around the probe tip so that PC-DMIS considers close misses as collisions.
5. Type a value in the **Insert clearance moves** box. This value defines the distance that the probe moves away from the part when it detects a collision while trying to move the probe to the next feature. PC-DMIS attempts to go directly to the next feature, but compensates for collisions by inserting clearance move commands. It continues this process until it defines a collision free path to the next feature. More than one move command may be inserted between features.
6. Select the **Delete existing moves** check box if you want to delete any pre-existing move commands in the part program.

For Dual Arm programs, PC-DMIS will insert “Move Exclusive” commands as necessary to guard against the two arms colliding during execution.

Glossary

I

Inspection Planner: Refers to the stand-alone application that is used to prepare CATIA V5 files to import into PC-DMIS for inspection. It should not be confused with the Inspection Planner components that exist within PC-DMIS.

IP: This is the acronym for PC-DMIS Inspection Planner. It is also used to refer to the stand-alone Inspection Planner application.

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.

Index

3

3D Scan Feature 7

A

Alignments 18

Angle Point 4

Auto Geometry Features

Creating 4

Dialog Box Options 5

B

Background Color 30

C

CAD file

Importing 2

CAD Information 29

CAD Vectors

Editing 28

Capture To Clipboard 31

Capture To File 31

CATIA V5 File 2

Circle 4

Comments

Inserting 22

Cone 4

Constructed Features 9

Operations 12

Corner Point 4

Cylinder 4

D

Dimension

Distance 15

Dimensions 13

Angle 17

Location 14

E

Edge Point 4

Edge Scan Feature 6

Ellipse 4

F

Fix Surface Vectors 28

Flip Surface Vectors 28

Flush & Gap 4

G

Generic Feature 11

Gradient Settings 30

Graphic Modes 26
Graphic View toolbar 27
Graphics Display Window 26

H

Highlight Color 30

I

ID Setup 23
Inspection Plan
  Deleting Elements 22
  Exporting 22
  Importing 38
  New 3
Inspection Planner 1
  CAD Files 2
  Installing 1
  Introduction 1
  Status Bar 24
  Toolbars 24
  Within PC-DMIS 33
Inspection Planner Default File 33
Inspection Planner Parameters 33
Inspection Planner Rules 33, 36
IPD 33
IPR 33, 36

L

Line 4

M

Material Thickness 24
MOHL 30
Mouse Over Highlight 30
Move Commands
  Inserting 44

N

Notch Slot 4

O

OpenGL Options 32

P

Path Optimization
  Dual Arm 42
  Performing 39
  Summary 43
Plane 4
Polygon 4

R

Round Slot 4

S

Screen Colors 30

Shortcut Menus..... 25

Sphere 4

Square Slot 4

Surface Points.....4

V

Void Detection.....43