
PowerINSPECT 2016

Quick Start Guide



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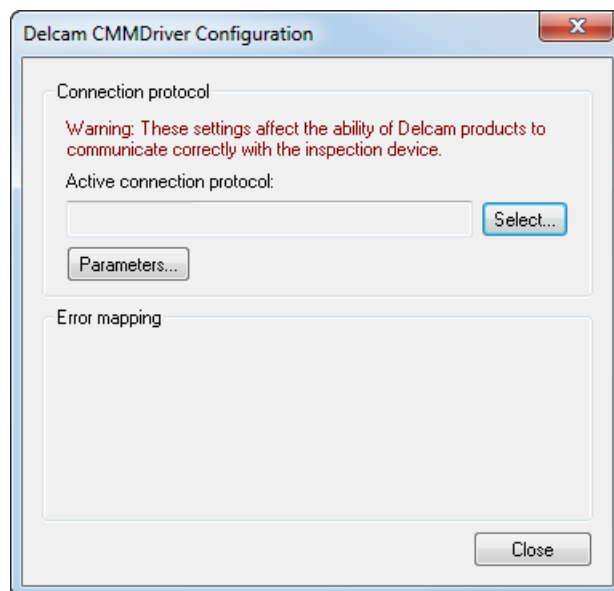
Starting PowerINSPECT for the first time

When you first start PowerINSPECT, you must specify the communication protocol used by the measuring device.

To select a protocol:

- 1 Double-click the PowerINSPECT icon on the desktop, or select **Start > All Programs > Delcam > PowerINSPECT > PowerINSPECT 2016**.

The **CMMDriver Configuration** dialog is displayed:




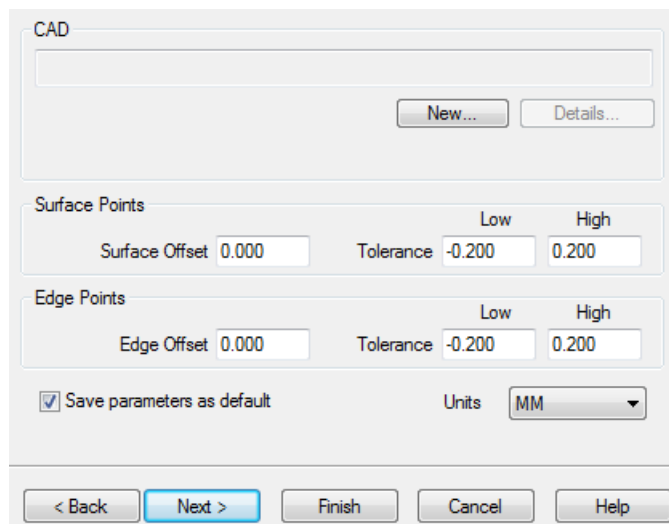
- 2 Click the **Select** button.
- 3 In the **Select Connection Protocol** dialog, select the communication protocol for your measuring device, and then click **OK**.
- 4 In the **CMMDriver Configuration** dialog, click **Close**.

Create an inspection document

In PowerINSPECT, each part is inspected using an inspection document. The document specifies the CAD model of the part (if available); the features to be used to align the part with its CAD model; the type of alignment to be used; and the features to be measured on the part.

To create an inspection document for a part:

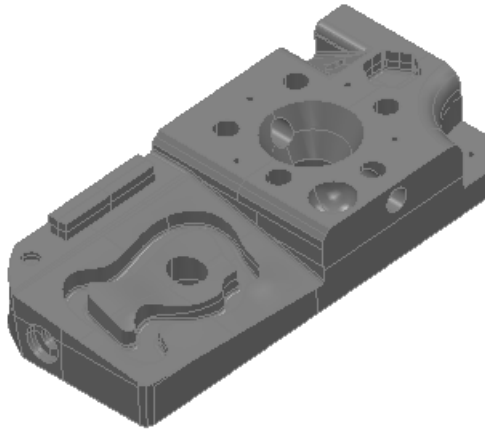
- 1 Select the **File > New Document Wizard** menu option or click  in the **Main** toolbar to display the **New Document Wizard**.
- 2 Select **Measurement with a single CAD part** to create an inspection using a CAD model.
- 3 Click **Next**.



- 4 Click the **New** button.
- 5 In the **Open** dialog, select the model of the part you want to inspect and click **Open**.

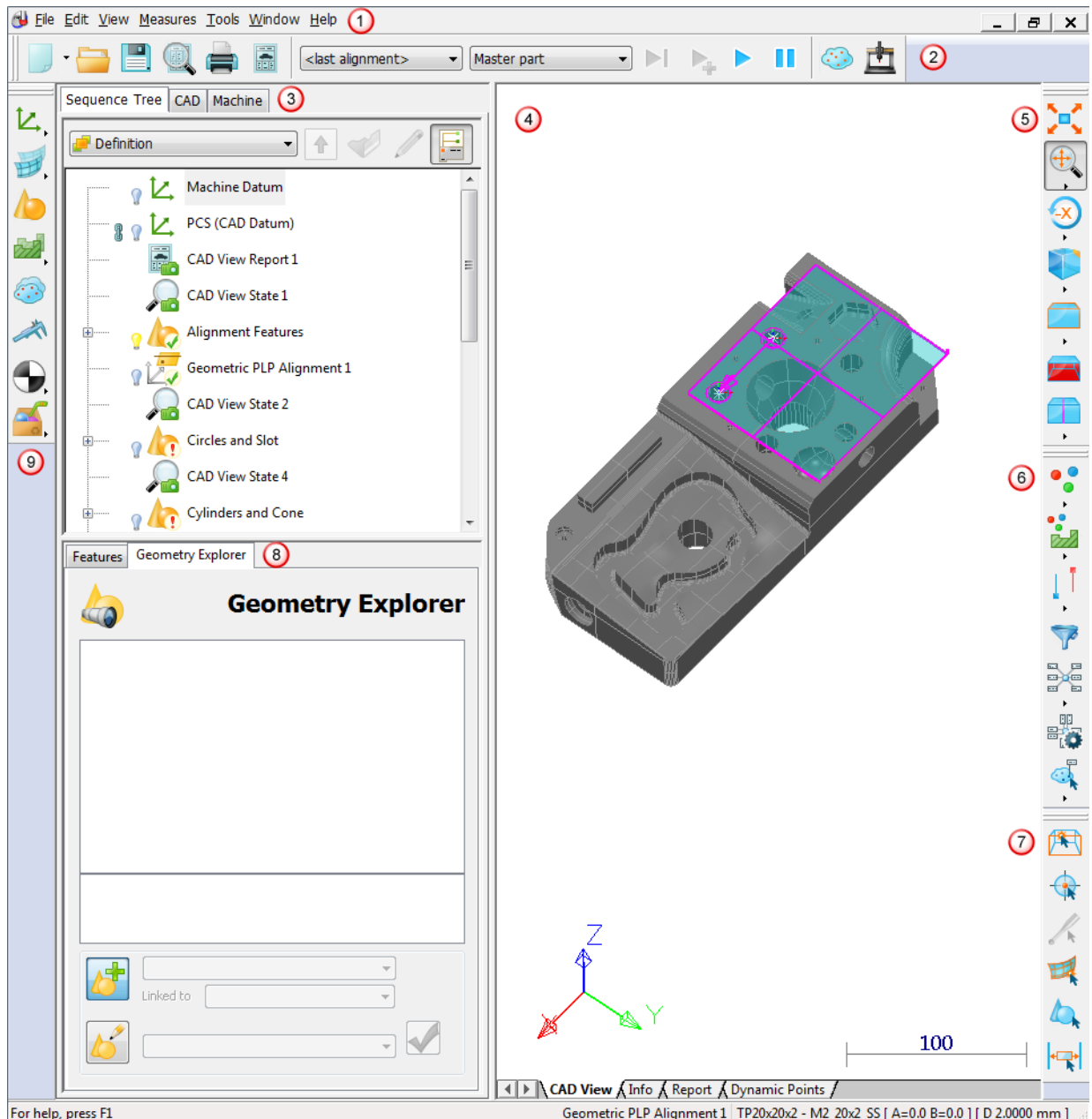
For example, to use the sample model provided with PowerINSPECT, select [DemoBlock2008\(CMM+Arm\).ddz](#) in the **Samples** folder of your PowerINSPECT installation folder.


- 6 In the wizard, click **Finish** to open the document and display the model in the **CAD View** tab. For example:




Navigating the PowerINSPECT screen

When you have selected the CAD model you want to work with, it is displayed in the PowerINSPECT screen together with the toolbars and tabs you need to create, run and manage an inspection. For example:



Area	Description
①	<p>The Menu bar contains options that enable you to import, edit and export data, to manipulate the screen, to play the inspection sequence, and to manage the configuration and tools available within PowerINSPECT. Many of the options are available in the toolbars and through keyboard shortcuts.</p>
②	<p>The Main toolbar enables you to create and save inspections; to choose the alignment you want to use; and to run the sequence of items to be measured.</p> <p>The Machine toolbar enables you to connect to and work with your measuring device. If this toolbar is not displayed, click  in the Main toolbar.</p>
③	<p>The Sequence Tree tab contains the inspection sequence. It specifies the items to be measured and the order in which they are to be measured.</p> <p>The CAD tab contains the details of the CAD model you loaded.</p> <p>The Machine tab enables you to view and calibrate the measuring device to which PowerINSPECT is connected.</p>
④	<p>The Graphics area contains several tabs that enable you to view the model and your measurement data:</p> <ul style="list-style-type: none"> ▪ The CAD View tab displays the models loaded in this inspection. ▪ The Info tab displays the details of the currently selected item in the inspection sequence. ▪ The Report tab displays an HTML report for all the items in the inspection sequence. ▪ Other tabs, such as the Section View tab, Dynamic Points tab, and Point Cloud tab display the details for specific types of item that can be added to the inspection sequence.
⑤	<p>The CAD View toolbar enables you to manipulate the model in the CAD View tab.</p>

Area	Description
6	The View Options toolbar enables you to display measurement information about features in the inspection sequence.
7	The Mouse Context toolbar controls the way in which the mouse interacts with the CAD view. It includes the Wireframe Checker  button, which enables you to pick the geometric features to be measured directly from the model.
8	The Features tab enables you to view and change the details of items in the inspection sequence. The Geometry Explorer tab displays the details of features selected from the CAD model using the Wireframe Checker.
9	The Item toolbar enables you to add items to the inspection sequence. These items specify the alignments to be used, the features to be measured, custom actions to be performed, and other operations, such as items that automatically change the CAD view or that can display information during the inspection.

Manipulating the CAD model

PowerINSPECT provides several tools to enable you to move, rotate, and resize the model displayed in the CAD view. These are located in the **CAD View** toolbar on the right of the PowerINSPECT window. Select:



to resize the model so that it just fits the CAD view.



to manipulate the CAD model using the left mouse button. Click and select a button to choose whether you want to use the mouse to zoom the model; to zoom on a selected area; to move the model; or to rotate the model around a specified point.












to rotate the CAD model using the left mouse button. Click and select a button to choose the axis and the direction of rotation.



to choose the direction from which the model is displayed. Click and select a button to choose view direction.

You can also change the CAD view using the mouse and keyboard:

To...	Use...
Zoom in and out of the CAD view	 and 
Zoom in on an area by drawing a box around it	 or  or 
Pan the CAD view	 or 
Rotate the CAD view in any direction — a transformation globe appears to show the rotation direction	 or 

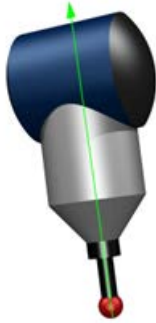
Probe compensation

PowerINSPECT provides two methods for determining the direction of probe compensation:

- Automatic probe compensation (see page 9) calculates the direction of compensation from the orientation of the probe when the first point is taken. This method is used by default.
- Pre-touch point probe compensation (see page 10) calculates the direction of compensation by taking a fresh-air point before taking the first measurement point.

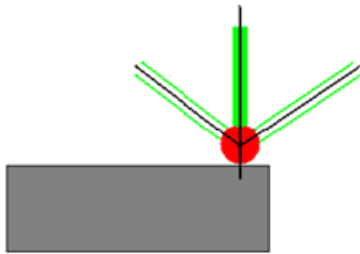
Automatic probe compensation

By default, PowerINSPECT automatically applies probe compensation in the direction of the vector running from the stylus tip to the last knuckle joint of the arm.



To use this method, you must ensure the probe is pointing at the surface being measured when you take a point. The following diagrams show how to position the probe:

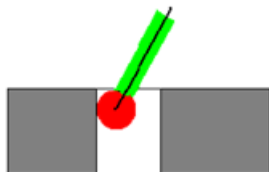
- Correct orientation of the probe relative to the surface.



- Incorrect orientation of the probe relative to the surface.



- When probing inside features, the probe must point towards the surface of the hole as near normal to the surface of the feature as possible. For example:



Pre-touch point probe compensation



Pre-touch point probe compensation enables you to specify the direction of probe compensation by taking a fresh-air point before taking points on the part. Use it when you are unable to orientate the probe towards the feature you want to measure.

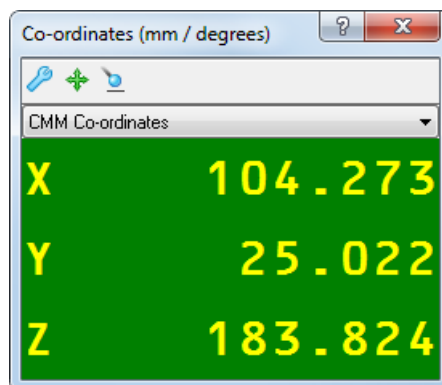
The way in which you use pre-touch points depends on the item to be probed:


- For geometric features, you must take a fresh air-point before the *first* touch point on the feature.
- For inspection groups, you must take a fresh-air point before *each* touch point in the group.


In both cases, PowerINSPECT determines the direction of probe compensation by calculating the vector between the touch point and the fresh-air point.

To use pre-touch points to determine the direction of probe compensation:

- 1 Click  to display the **Machine** toolbar.
- 2 Click  to display the **Co-ordinates** dialog. For example:




- 3 Select  to activate the pre-touch function.
- 4 Take a point on the fresh-air side of the first touch point.
- 5 Measure the feature as normal by taking the required number of points on the part's surface.

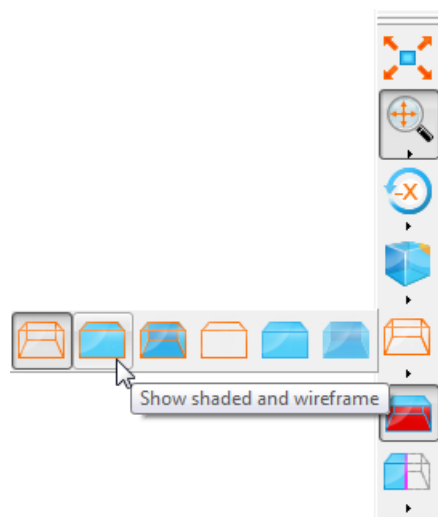
When you have finished, deselect  to deactivate the pre-touch function.

Aligning the part to the model

If you want to compare the measurements of the part with its nominal values, you must first align it to the CAD model. This matches the model to the part, and enables PowerINSPECT to report whether measured features are in or out of tolerance.

You can create several different types of alignment in PowerINSPECT; which one you choose depends on the part and your requirements. This section describes how to create a Geometric PLP alignment, which uses three geometric features, a Plane, a Line and a Point, to fix the position of the part in the X, Y, and Z axes. The following sections describe how to create these items and how to use them to create an alignment.

Before starting, click  below the **Shading mode** button in the **CAD View** toolbar (on the right of the PowerINSPECT window), and select the **Show shaded and wireframe** option.



This shades the CAD model making it easier to see the surfaces and features you are working with.



*This example of creating an alignment is described for demonstration purposes only. You can create all these items in one click using the **Plane, Circle, Circle PLP Alignment** button in the **Simple Measures Wizard**. For more information, refer to the online help.*

Stage 1: Create a geometric group

In PowerINSPECT, geometric items are stored in geometric groups. These enable you to organize the inspection sequence, and to keep related items together. The first step in aligning a part to its model is to create a geometric group for the items used in the alignment.

To create a geometric group:

- 1 In the **Item** toolbar (see page 4), click the **Geometric Group** button. The **Geometric Group** dialog is displayed.



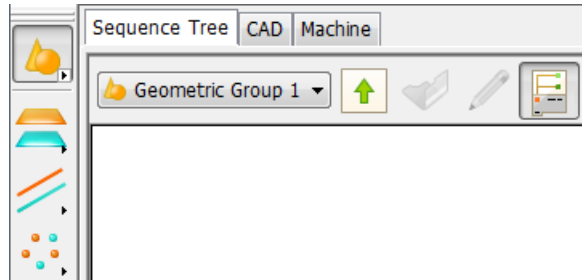
The screenshot shows the 'Geometric Group' dialog box. It has a title bar with a close button. Inside, there's a yellow sphere icon. On the right, there are three checkboxes: 'Automatically trim entities' (unchecked), 'Output in report' (checked), and 'Visible' (checked). Below these is a 'Group name' section with a text field containing 'Geometric Group 1' and a small icon button. Underneath is a 'Default tolerances' section with a table of tolerance values. At the bottom of this section is an 'Update tolerances of existing entities' checkbox. Below that is a 'Default coordinate system' section with a 'Coordinate system' dropdown menu set to '<Active Alignment>' and an 'Update coordinate system of existing entities' checkbox. At the very bottom is a 'Comment' text area and three buttons: 'OK', 'OK & Repeat', and 'Cancel'.

	Low Tol	High Tol
Positional tolerance:	-0.100	0.100
Dimensional/diametrical tolerance:	-0.100	0.100
Angular tolerance:	-0.100	0.100
Form tolerance:		0.100

This dialog specifies the name of the group. It also specifies whether the measurements of the items it contains are to be displayed in the CAD view; the default tolerances for items created in the group; and the coordinate system relative to which the measurement results are to be reported.

- 2 Click **OK** to accept the default settings for the group.

The group is added to the inspection sequence and automatically opened. The **Active Group** list at the top of the Sequence Tree changes to display the name of the group, and the **Item** toolbar is replaced with the **Geometry** toolbar, which enables you to add geometric items to the inspection sequence.



Stage 2: Create a probed plane

The first item to be measured in a Geometric PLP Alignment is the plane. In this example, it is used to specify the orientation of the part's Z plane.

To create a probed plane:

- 1 In the **Geometry** toolbar, click  to display the **Planes** pull-out toolbar, which displays the different types of plane you can create.
- 2 In the toolbar, click **Probed Plane**.



The **Probed Plane** dialog is displayed. Like the **Geometric Group** dialog, it specifies the name of the item, the coordinate system in which the results are reported and the measurement tolerances for the item.

Probed Plane

Name: Plane 1

☒ Use nominals

Visible ☒
Output in report ☒

Page 1 | Page 2 | Point Sources(0)

Coordinate System: <Active Alignment>
Fitting Algorithm: Least Square

Flatness
Maximum: 0.100

Normal Vector
Coordinate Type: Cartesian

	Nominal	Low Tol	High Tol	
I	0.00000	0.00000	0.00000	<input checked="" type="checkbox"/>
J	0.00000	0.00000	0.00000	<input checked="" type="checkbox"/>
K	1.00000	0.00000	0.00000	<input checked="" type="checkbox"/>

Centroid
Coordinate Type: Cartesian

X: 0.000 mm Y: 0.000 mm Z: 0.000 mm

Orientation
Coordinate Type: Cartesian

I: 1.00000 J: 0.00000 K: 0.00000

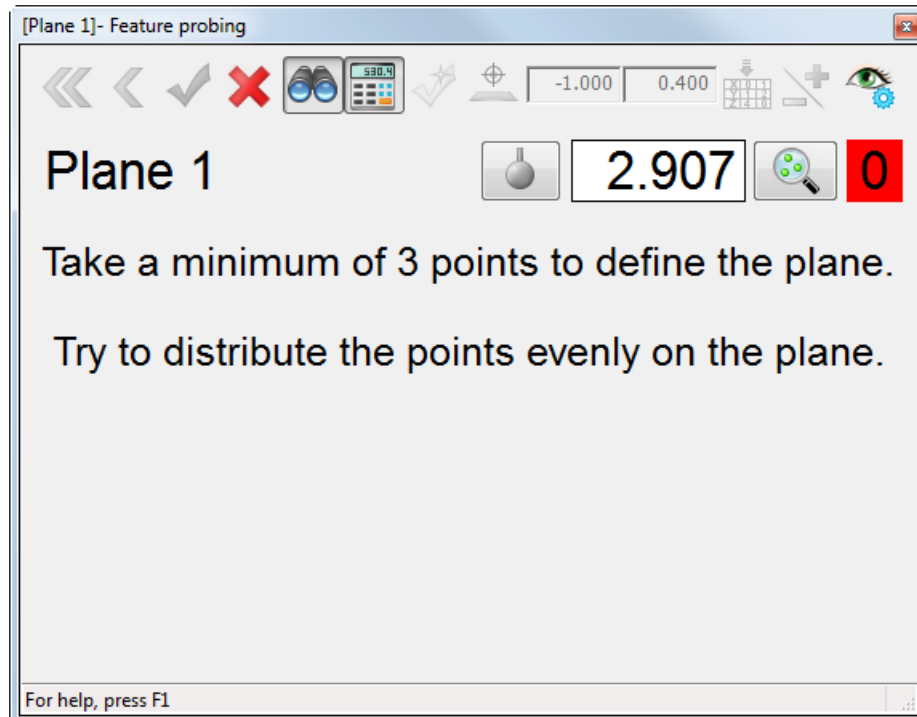
Length: 20.000

Comment:

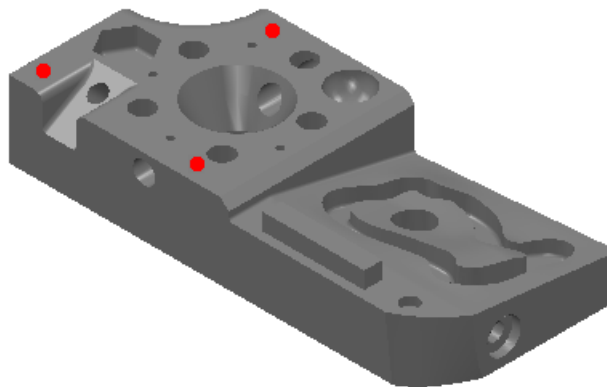
OK OK & Repeat Cancel Help


- 3 Click **OK** to accept the defaults.

The **Feature Probing** dialog is displayed. It shows the name of the geometric item you are probing; the diameter of the probe; the number of points you have probed; and the minimum number of probe points required to measure this feature.



- 4 Probe three points on the part, as shown:




- 5 When you have taken the third point, the background of the point count changes from red to green to indicate you have taken enough points to measure the plane. Click the  button with the mouse to accept the points.

Stage 3: Delay measurement

Instead of probing each item as you add it to the inspection sequence, you can delay measurement until you have created an item for all the features you want to measure.

To delay the measurement of items, click the **Delay Measure**




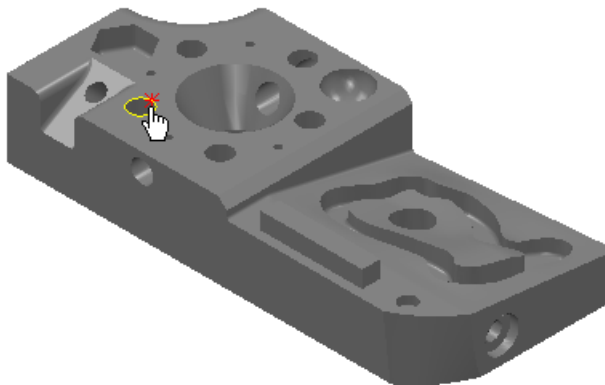
button on the **Main** toolbar. The button changes to . You can now add several items to the sequence without stopping; PowerINSPECT prompts you to take points only when you play the inspection sequence (see page 19).

Stage 4: Create two probed circles

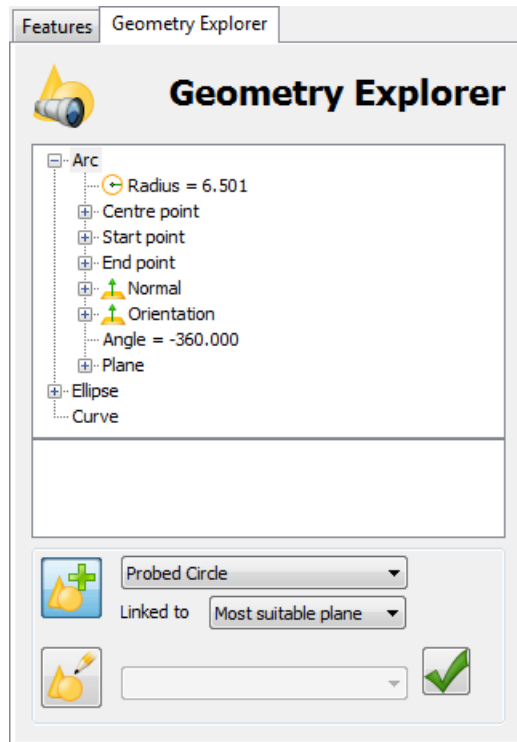
As well as specifying the items you want to inspect using the **Item** toolbar, you can create items by selecting the features directly from the CAD model. This example demonstrates how to create inspection items for the two probed circles used in the alignment. The centre of the first circle is used to determine the origin of the alignment. The centre of the second circle is used to construct a line in Stage 5 (see page 18).

To create two probed circles:

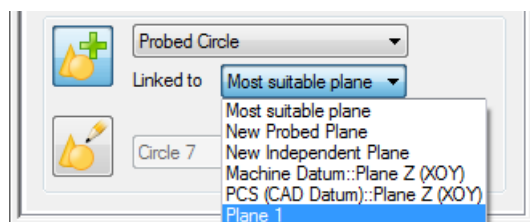
- 1 Select the **Wireframe Checker**  button in the **Mouse Context** toolbar at the bottom-right of the PowerINSPECT window.
- 2 Move the mouse pointer to the edge of the circle shown below.




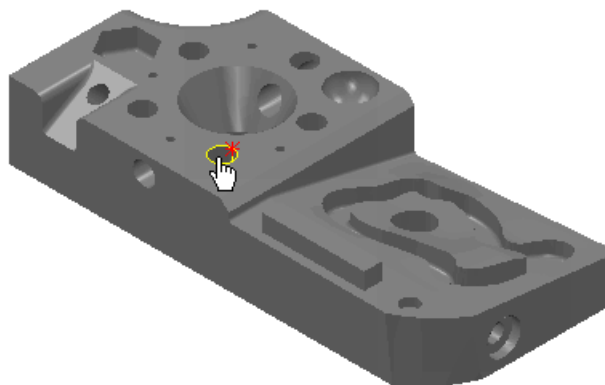
- 3 When the circle is highlighted in yellow, click the left mouse button. The circle's nominals are displayed in the **Geometry Explorer** tab.



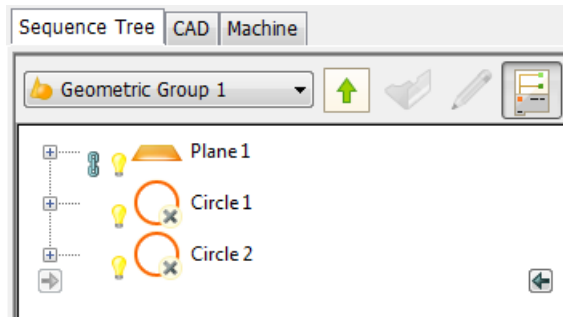
- 4 In the **Linked to** list, select *Plane 1* to associate the circle with the plane you created in Stage 2.



- 5 Click  to add the circle to the inspection sequence.
- 6 Move the mouse to the edge of the circle shown below, and repeat steps 3 to 5 to create a second circle item.




Notice the symbols shown alongside the items you have added to the inspection sequence.



 indicates the item is unmeasured.

 indicates the item is displayed in the CAD view.


 indicates the item is used by other items in the sequence. You cannot delete a referenced item until you have deleted the items that use it.

- 7 Click the light-bulb symbol next to the plane item to hide it in the CAD view.

Stage 5: Construct a line

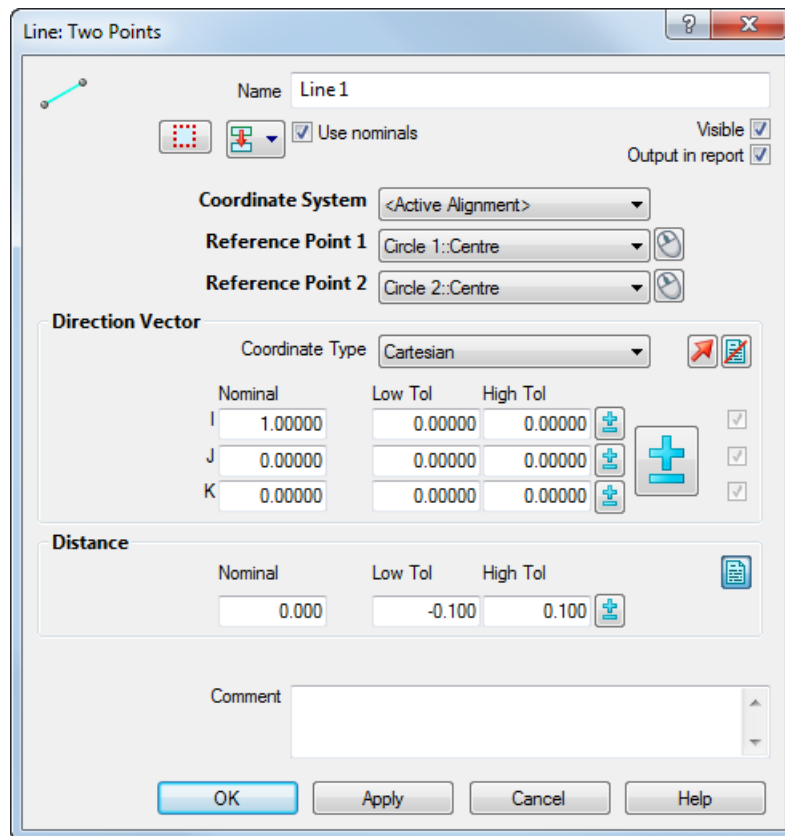
In PowerINSPECT, you can create measurements from other items; this is called constructing an item. This section describes how to construct a line using the centres of the circles you created in Stage 4 (see page 16). The line is used to specify the positive direction for the alignment's X axis.

To construct a line for the alignment:

- 1 In the **Geometry** toolbar, click the **Lines**  button. In the **Lines** pull-out toolbar, select the **Line: Two Point** button.



The **Line: Two Points** dialog is displayed.



- 2 The line is created from the items selected in the **Reference point 1** and **Reference point 2** lists. In this case, the centres of the circles are used as the start and end points of the line, and the positive direction for the alignment's X axis runs from **Reference point 1** to **Reference point 2**.
- 3 Click **OK** to add the line to the inspection sequence.

Stage 6: Take points for the unmeasured features

When you have selected the **Delay Measure** button in the **Main** toolbar, PowerINSPECT does not prompt you to probe each feature as it is created. Instead, when you are ready to measure the items you have created, you must choose one of the play options in the **Main** toolbar. You can select:



to measure the selected item in the sequence.




to measure all unmeasured items in the sequence.

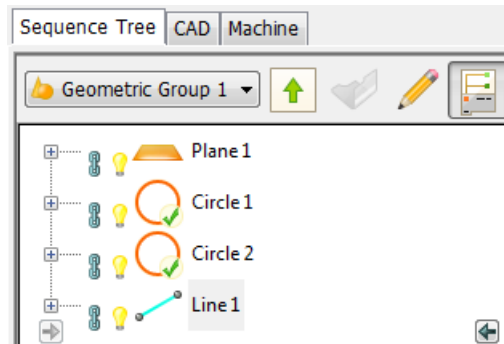



to add points to the selected item in the sequence.


To measure all unmeasured items in the inspection sequence:

- 1 Click the **Play All**  button on the **Main** toolbar. PowerINSPECT prompts you to take the points for each unmeasured item (**Circle 1** and **Circle 2**) in the order in which they appear in the sequence.


When you have completed the measurements, the symbols shown alongside the measured items change.



 indicates the item's results are within tolerance.

 indicates one or more of the item's results are out-of-tolerance.


- 2 If you want to remeasure an out-of-tolerance item, right-click its entry in the inspection sequence and select **Play Item** from the context menu; or select its entry in the sequence, and then click


the **Play Item**  button in the **Main** toolbar.

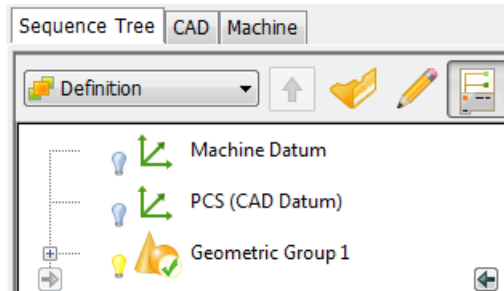
Stage 7: Create an alignment

Geometric PLP alignments enable you to align a part to its model using a plane, line, and point for which you know the nominal coordinates. This type of alignment allows you to use any method of creating the component items. It also allows you to modify the alignment by amending the details of the geometric items or by remeasuring them individually.


To create the alignment:

- 1 Click  at the top of the Sequence Tree to close the geometric group.

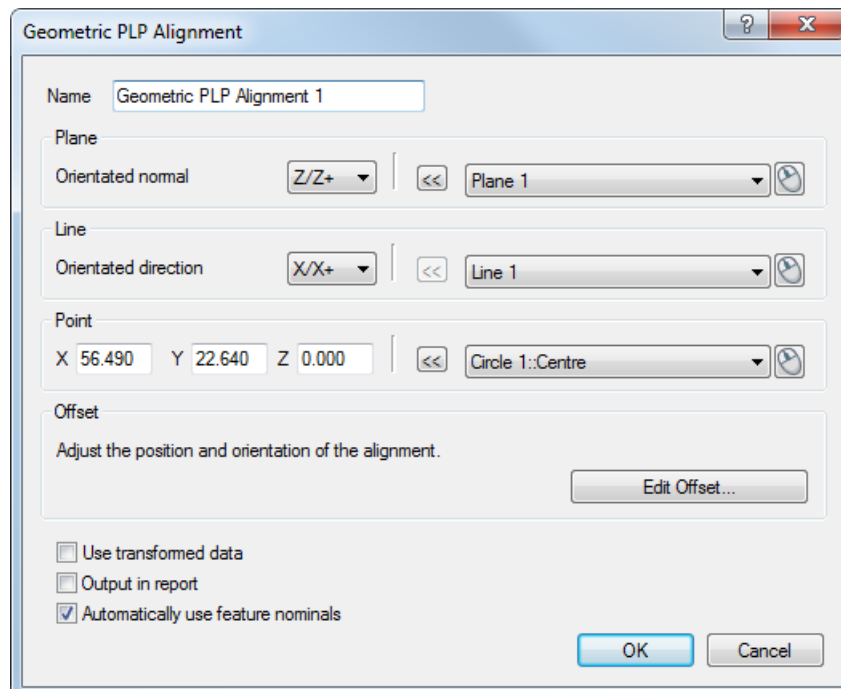
The **Active Group** list changes to *Definition* to show you have closed the group and are working at the top-level of the inspection sequence. The in-tolerance  icon is displayed on the group to indicate the measurement status of the items it contains.



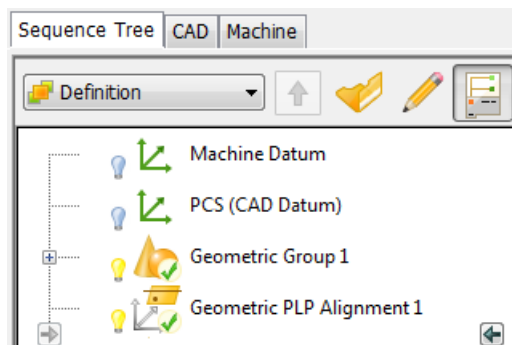
2 In the **Item** toolbar, click , to display the **Alignment** pull-out

toolbar, and then click the **Geometric PLP**  button to create an alignment. The **Geometric PLP Alignment** dialog is displayed:

- In the **Plane** area, **Oriented normal** is $Z/Z+$ to indicate the positive direction of the Z axis is the same as the direction of the specified plane's normal vector (up from the part).
- In the **Line** area, **Oriented direction** is $X/X+$ to indicate the positive X direction is in the direction of the specified line.
- In the **Point** area, the **XYZ** boxes are set to the location of the selected point item: *Circle1::Centre*.



- 3 In the **Line** area, select *Line 1*, then click **OK**. This creates the alignment and adds it to the inspection sequence.



Inspecting the part



PowerINSPECT can measure 2D and 3D geometry as well as inspect 3D non-geometric surfaces. The following sections describe the methods for:

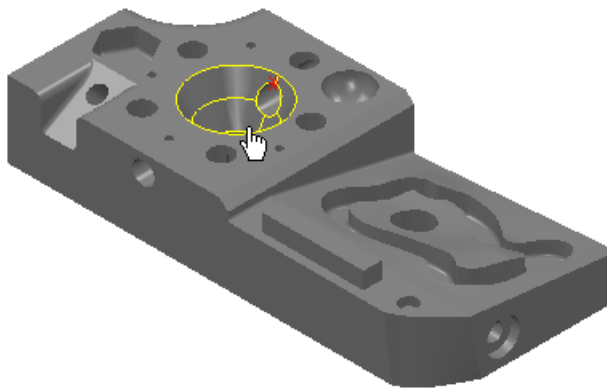
- measuring a cone (see page 24);
- measuring a slot (see page 26);
- inspecting a surface (see page 30);
- optimizing the alignment (see page 35);
- displaying detailed feature information (see page 43);
- viewing the measurement results (see page 39);
- printing the report (see page 50).

Measuring a cone


You measure a cone by probing a minimum of seven points evenly spread across the cone surface. It is important to keep the axis of the stylus as near normal to the surface of the feature as possible.

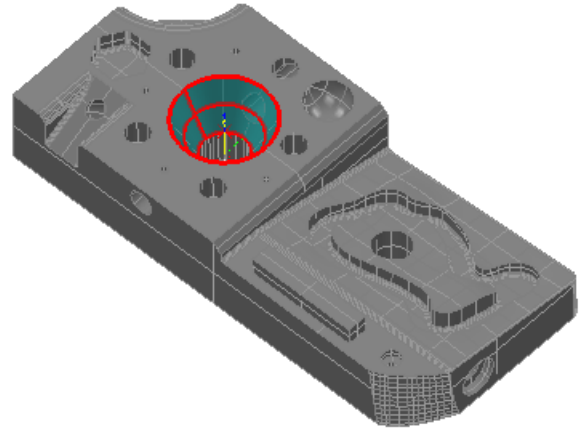
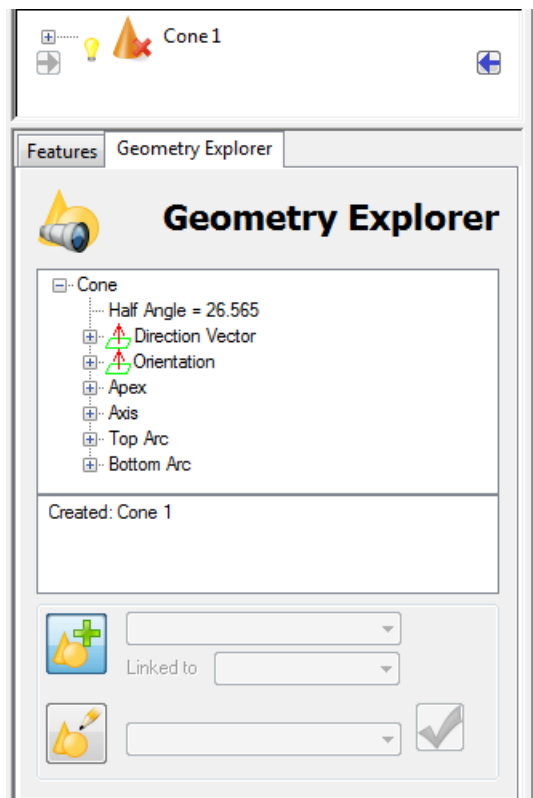
To measure a cone:

- 1 Make sure the **Delay Measure**  button on the **Main** toolbar is selected.
- 2 Create a geometric group (see page 12) to hold the new items to be measured.
- 3 Select the **Wireframe Checker**  button so you can pick features from the CAD model.
- 4 To create 3D items using the **Wireframe Checker**, you must change its picking mode. To do this, right-click the CAD view and select **Surface** from the context menu, then left-click the CAD view to close the menu.
- 5 In the CAD view, move the mouse over the cone to be measured. When the cone is highlighted in yellow, click the left mouse button. For example:



The cone is displayed in red, and its details are displayed in the **Geometry Explorer** tab.

- 6 Click the  button to add the cone to the inspection sequence.

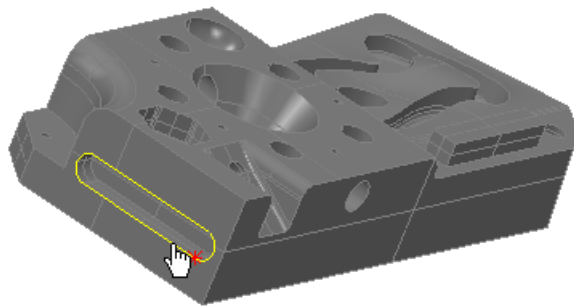


Measuring a slot

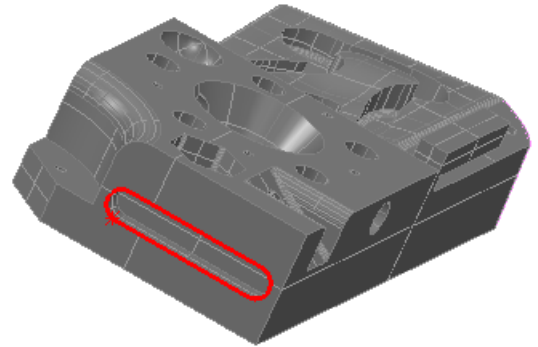
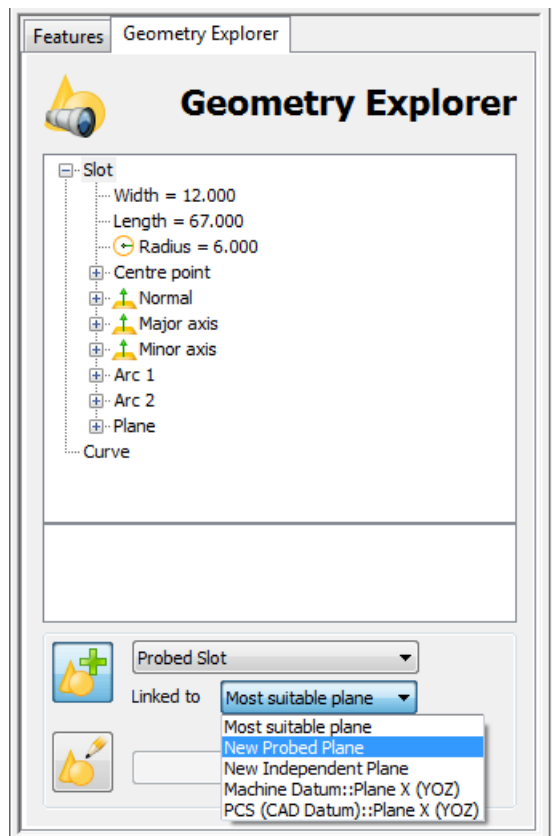
Slots are 2D pieces of geometry so they need to be associated with, or referenced to, a plane. You must probe a minimum of six points to measure them.


To measure a slot:

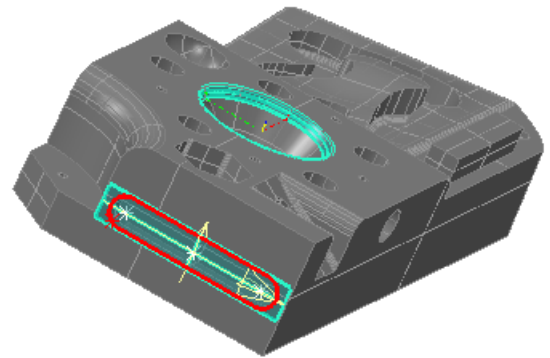
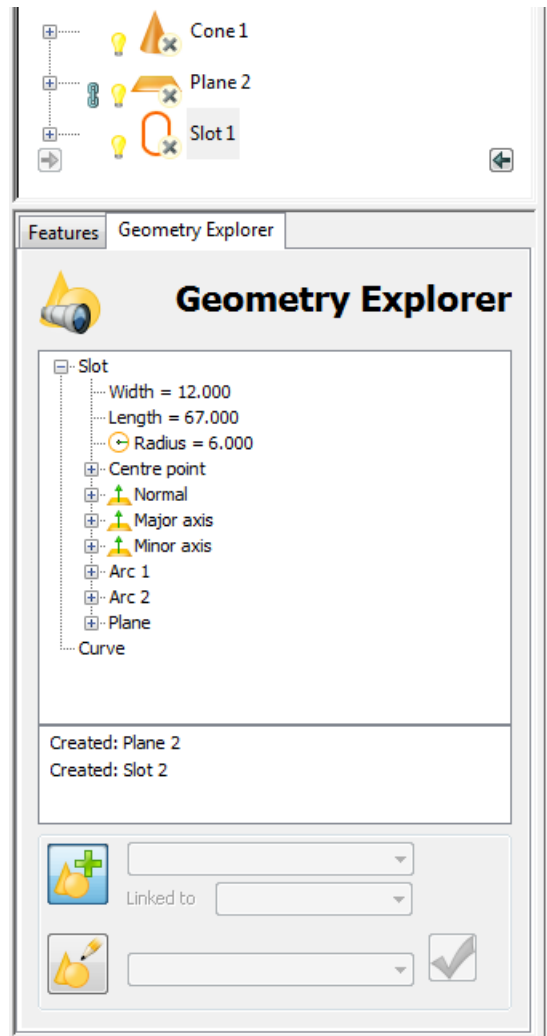
- 1 To enable 2D feature picking, right-click the CAD view and select **Wireframe** from the context menu. Left-click the CAD view to close the menu.
- 2 In the CAD view, move the mouse over the slot to be measured. For example:



- 3 When the slot is highlighted, click the left mouse button to select it. The slot's details are displayed in the **Geometry Explorer** tab.
- 4 In the **Linked to** list of the **Geometry Explorer** tab, select *New Probed Plane*. This creates a new reference plane for the slot.



- 5 Click the  button to create the item. The slot and the plane are added to the inspection sequence.



- 6 Double-click the **Slot 1** icon in the inspection sequence to display details of the item and check that the new probed plane is shown in the **Reference Plane** list.

Probed Slot

Name Slot 1

☒ Use nominals Visible ☒
Output in report ☒

Page 1 Page 2 Point Sources(0)

Coordinate System <Active Alignment>

Material Side Hole (ID)

Fitting Algorithm Least Square

Reference Plane Plane 3

Centre

Coordinate Type Cartesian

	Nominal	Low Tol	High Tol
X	0.000 mm	-0.100	0.100
Y	37.500 mm	-0.100	0.100
Z	-25.000 mm	-0.100	0.100

Length

☒ Overall ☐ Centres

	Nominal	Low Tol	High Tol
Overall	67.000	-0.100	0.100

Width

☐ Radius ☒ Width

	Nominal	Low Tol	High Tol
Width	12.000	-0.100	0.100


Offset/Thickness


-0.000 ☐ Calculate from measurement

Comment

OK Apply Cancel Help

- 7 Click **Cancel** to close the dialog.

In the inspection sequence, the  symbol is shown on the Plane 2 item to indicate the plane is referenced by other items in the sequence.

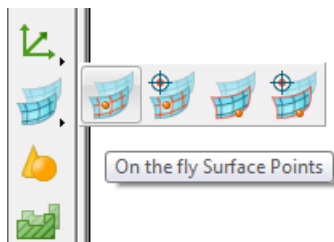
- 8 Click  to close the geometric group.

Inspecting a surface

Surface inspection groups enable you to inspect non-geometric surfaces on a part, and to compare the results with the CAD model. You can display the results in the CAD view using the **Inspection Point Display** options in the **View Options** toolbar, or view detailed information about each point using the **Report** and **Info** tabs.

To create a surface inspection group:

- 1 In the **Item** toolbar (see page 4), click the **Inspection Groups** button to display the pull-out toolbar, then click the **On the fly Surface Points** button.




The **Inspection Group: Surface Points** dialog is displayed. It enables you to specify the type of points to be taken, the method to be used to take the points, the naming rules for the points, tolerances and so on.

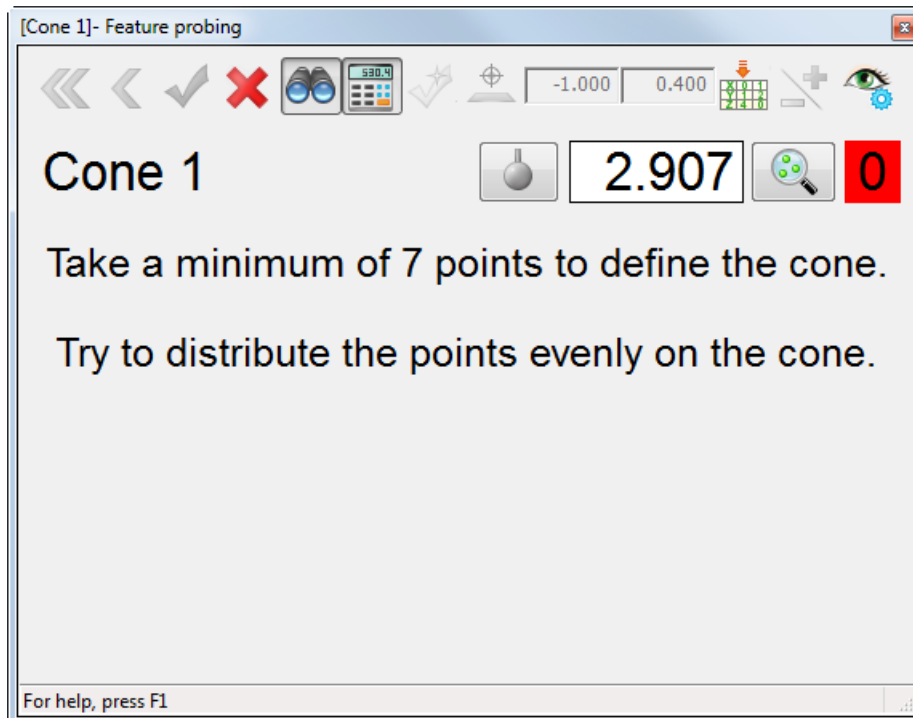
- 2 Ensure the **Output in report** check box is selected. This enables you to display the measurements as labels in the CAD view (see page 40), and in the report (see page 46).
- 3 By default, you must measure a minimum of six points for a surface inspection group, but you can change this value to suit the surface you want to measure. Enter **8** in the **Minimum points** box to specify the minimum number of points to be measured on this surface.
- 4 Click **OK** to add the group to the inspection sequence.



Measuring the unmeasured items

There are now several unmeasured items in the inspection sequence.

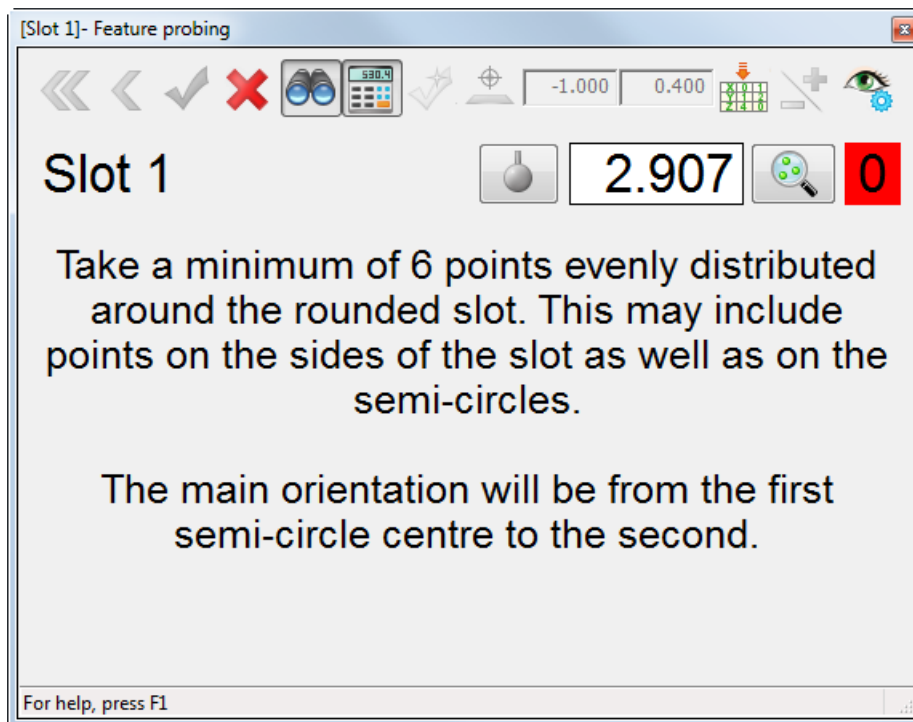
To measure these items:

- 1 Click the **Play All**  button on the **Main** toolbar. This causes PowerINSPECT to play all the unmeasured items. The **Feature Probing** dialog is displayed for the first unmeasured item in the inspection sequence.



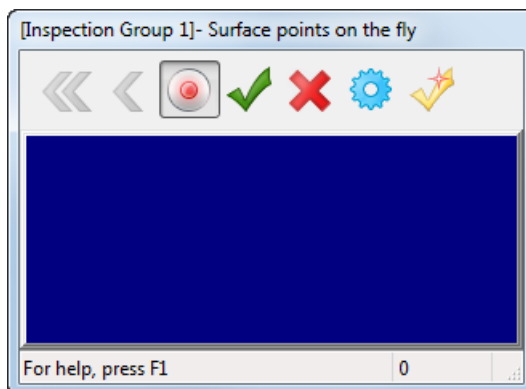
- 2 Take at least seven points from the cone. Click  to accept the points.
You are prompted to measure the reference plane you created for the slot.
- 3 Take three points on the plane and click  to accept the points.

You are prompted to measure the slot.

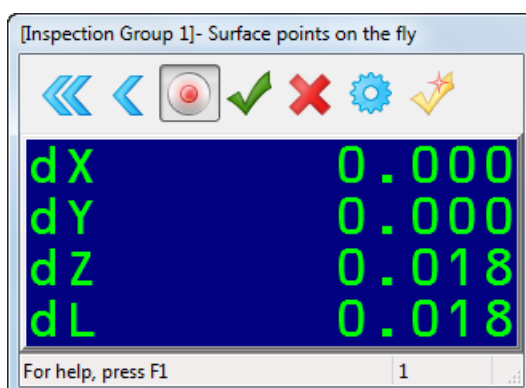


- 4 Take at least six points from the slot and then accept them.


The **Surface Points on the Fly** dialog is displayed for the surface inspection group.



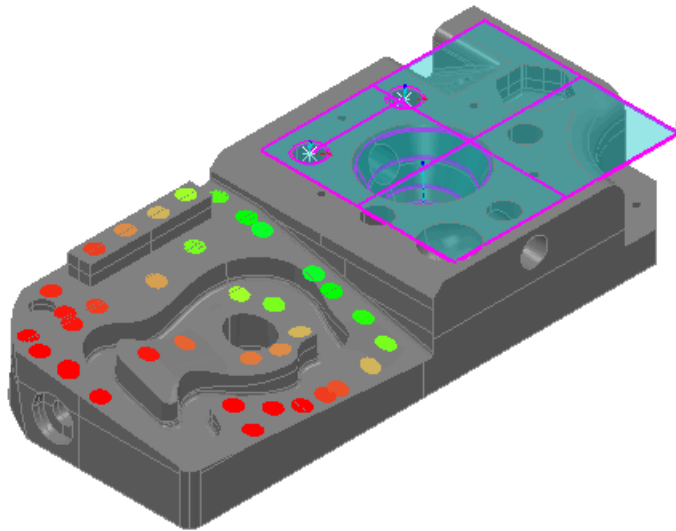
- 5 Take a point on the irregular sloping surface on the top of the part. PowerINSPECT compares it against the CAD model and updates the dialog, for example:




The **dL** value shows the distance from the point to the nominal surface; the **dX**, **dY**, and **dZ** values show the deviations of the measured location from the nominal in the X, Y and Z axes.

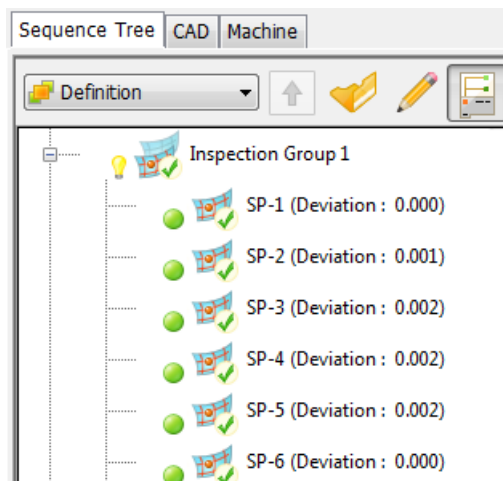
- 6 Take at least eight points and then click the  button with the mouse or press the red button on the arm to accept them.




PowerINSPECT returns to the CAD view, and displays a coloured spot for each point in the surface inspection group. For example:



The colour of each spot indicates whether it is within the tolerance specified for the inspection group:

- Red spots are above the high tolerance.
 - Blue spots are below the low tolerance.
 - Green spots are within tolerance.
- 7 To view the measurements for each point in the inspection group, click  to the left of the group's entry in the inspection sequence. The sequence lists each point, its deviation and a tolerance indicator. For example:




-  indicates the point is above the high tolerance.
-  indicates the point is below the low tolerance.
-  indicates the point is within tolerance.

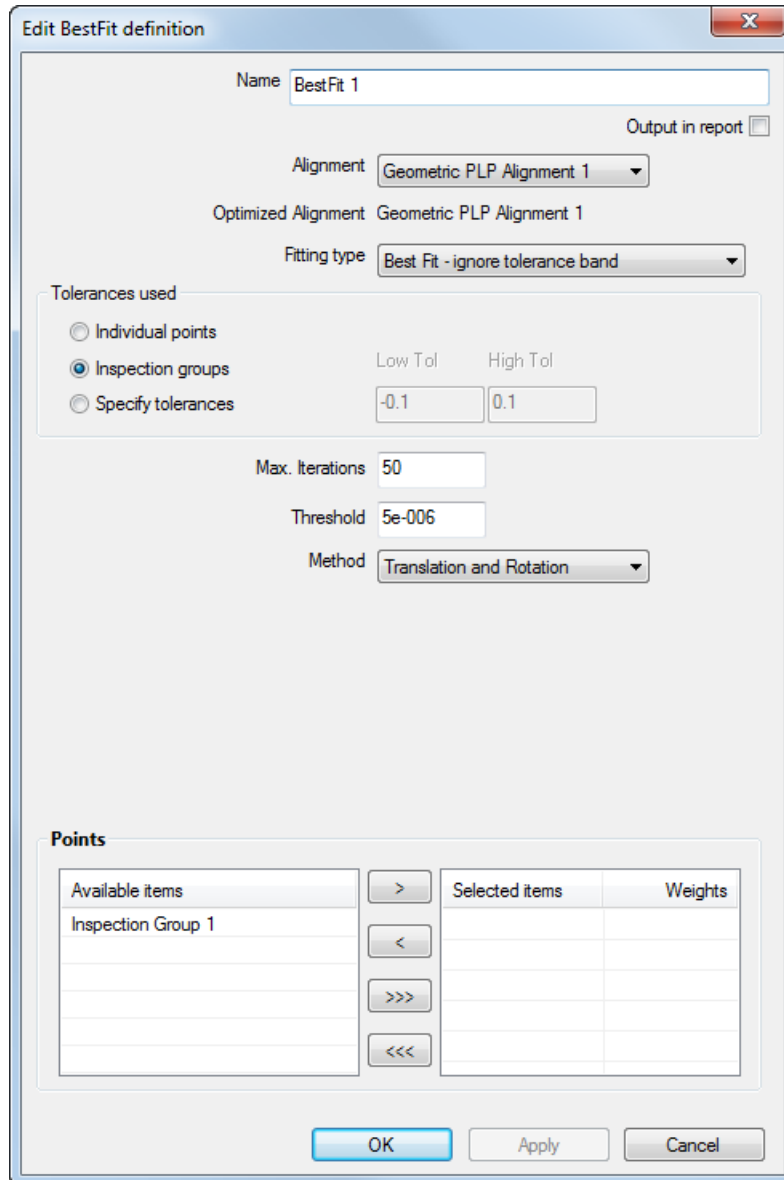
If points are out-of-tolerance, you can attempt to improve the alignment by adding a best fit item to the inspection sequence (see page 35).

Optimizing an alignment

When you have measured one or more inspection, section or point cloud groups, you can use Best Fit items to optimize the alignment of the part with the CAD model. PowerINSPECT compares the points of selected groups to the nominals and adjusts the alignment using one of five fitting methods.

To optimize the alignment:

- 1 In the **Item** toolbar (see page 4), click the **BestFit**  button. The **Edit Bestfit Definition** dialog is displayed.



Edit BestFit definition

Name: BestFit 1

Output in report: ☒

Alignment: Geometric PLP Alignment 1

Optimized Alignment: Geometric PLP Alignment 1

Fitting type: Best Fit - ignore tolerance band

Tolerances used:

- ☐ Individual points
- ☒ Inspection groups
- ☐ Specify tolerances

Low Tol: -0.1 High Tol: 0.1

Max. iterations: 50


Threshold: 5e-006

Method: Translation and Rotation

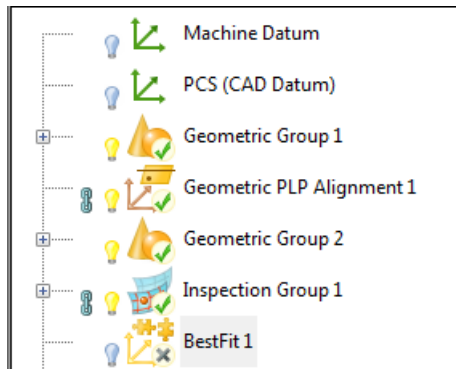
Points

Available items		Selected items	Weights
Inspection Group 1	>		
	<		
	>>>		
	<<<		

OK Apply Cancel

- 2 To use the inspection group you created to optimize the alignment, select its entry in the **Available items** list, and click  to move it to the **Selected items** list.
- 3 In the **Fitting type** list, select **Best Fit - ignore tolerance band**. This adjusts the original alignment so as to minimize the deviations of the group points.
- 4 In the **Method** list, select **Translation and Rotation**.

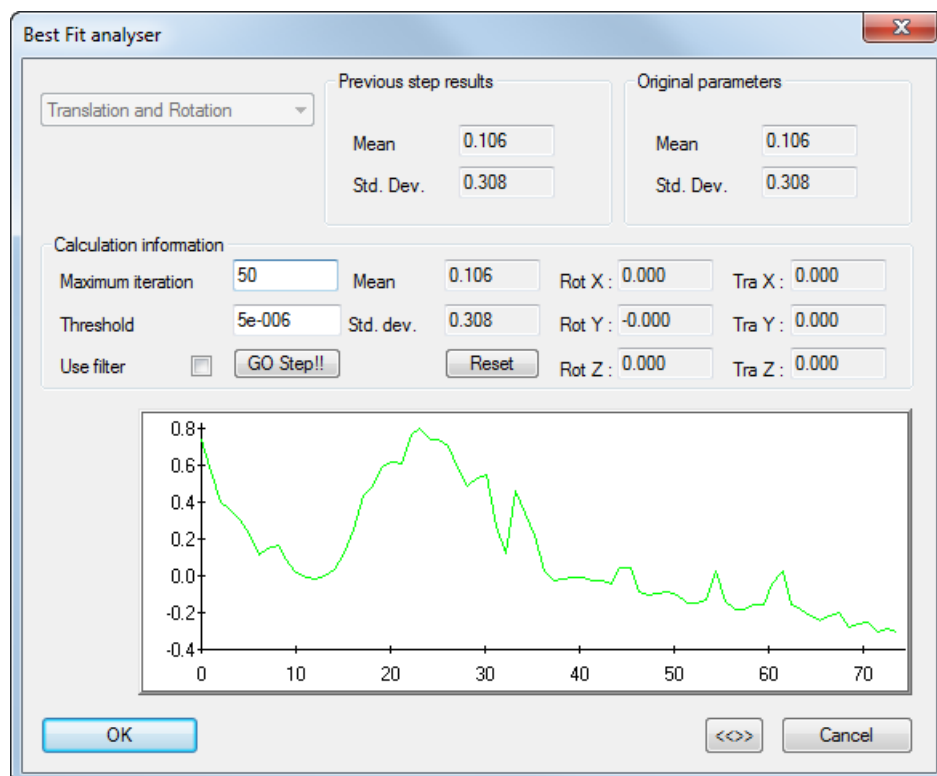
- 5 Click **OK** to add the **Best Fit** item to the inspection sequence. Like alignments, the Best Fit icon includes axes. This indicates that Best Fit items are treated in the same way as alignments.



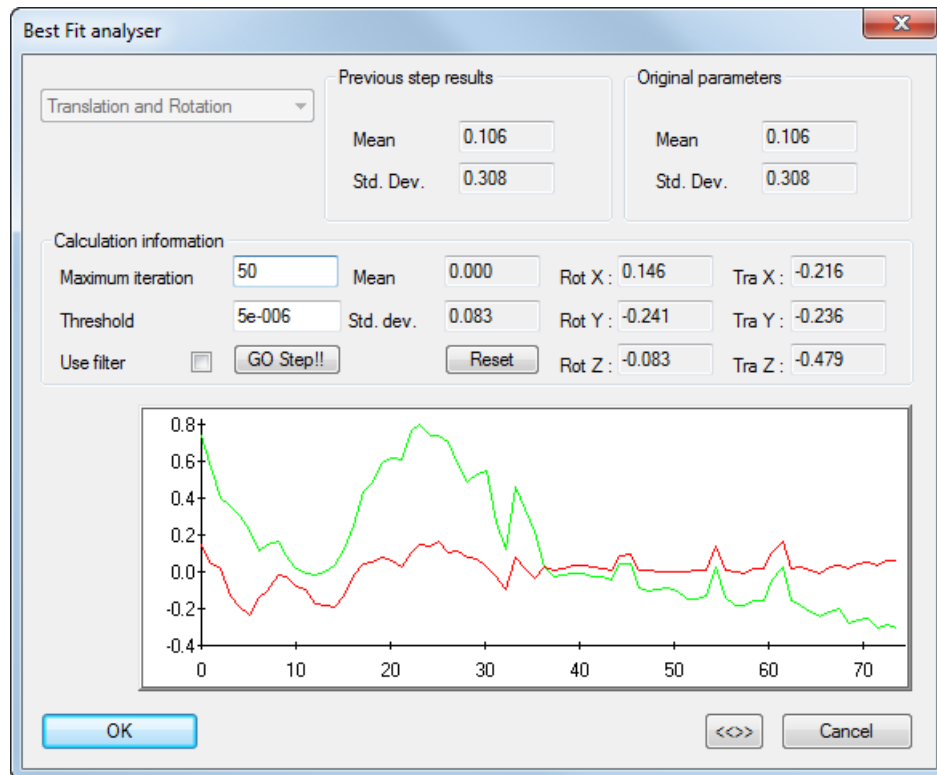
- 6 Right-click the **BestFit** item in the inspection sequence, and then select **Play Item** from the context menu.


The **Best Fit Analyser** dialog is displayed. For each step of the analyser, the mean and standard deviation for the points used in the calculation, and the rotations and translations applied to the alignment are shown. The analyser also specifies the maximum number of times that the calculation is to be repeated, and a threshold for the analysis.

For example:



- 7 Click **Go Step** to calculate the best fit. A graphical representation of the points' deviations is shown. The green line shows the measurement results before the best fit calculation; the red line shows the results after the best fit is applied.



- 8 If you want to change the scale of the graph, click . Click the button again to revert to the original scale.
- 9 When you are satisfied with the results of the optimization, click **OK** to close the dialog and apply the best fit.

PowerINSPECT automatically uses this optimized alignment, instead of the original alignment on which it is based, when reporting results.

Viewing measurement results

When you have inspected a part, PowerINSPECT provides several ways to view your results. This section describes how to:

- Display measurements in the CAD view (see page 40).
- View detailed information for an item (see page 43).
- View and print a report (see page 46).

Displaying measurements in the CAD view

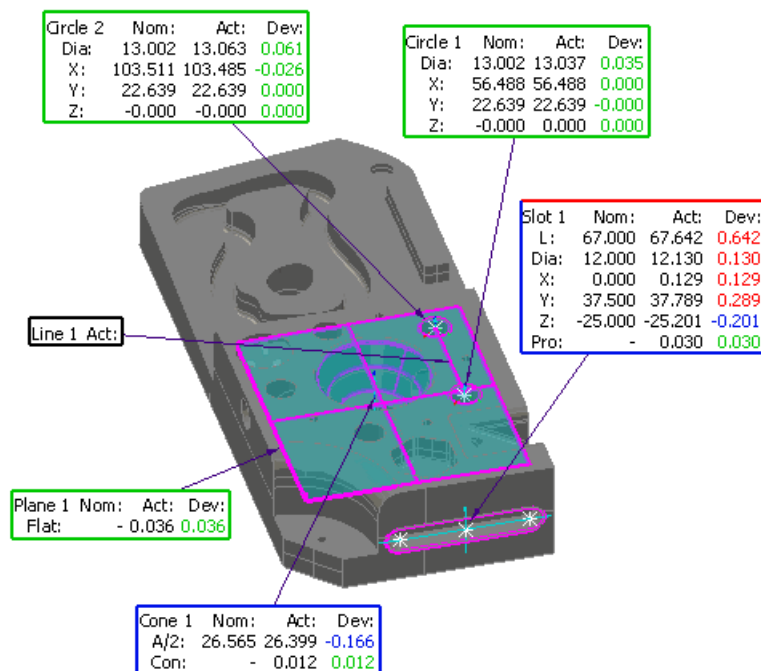
The **Show/Hide Labels** button in the **View Options** toolbar enables you to display the measurement details of geometric items in the CAD view. The labels that are displayed depend on the state of the Sequence Tree: if the Tree is displaying the Definition level of the inspection sequence, all measured items are shown in the CAD view; if a group is open in the Tree, only the labels of items in that group are shown.

To display labels for geometric items:

- 1 Display the Definition level of the inspection sequence.
- 2 In the **View Options** toolbar on the right of the PowerINSPECT



window, select the **Show/Hide Labels** button. Labels are displayed for all the items that can be seen in the CAD view. For example:



The measurements associated with each item are displayed in labels, together with an arrow that identifies the feature the item refers to. If you have chosen to compare an item's measurements with its nominals, the labels can also display the nominals, and the values are colour coded to indicate each measurement's status:

- Green indicates the value is within tolerance.

- Red indicates the value is above tolerance.
- Blue indicates the value is below tolerance.
- Black indicates the value is not measured (such as for constructed items) or the value has not been compared with its nominal.

The label borders are also colour coded. When some of an item's measurements are above tolerance and some of the measurements are below tolerance, the label border is displayed as a mixture of colours.

- 3 By default, PowerINSPECT automatically positions the labels around the CAD view. To position the labels manually:

- a Click  below the **Show/Hide Labels** button, and select

Show manual labels

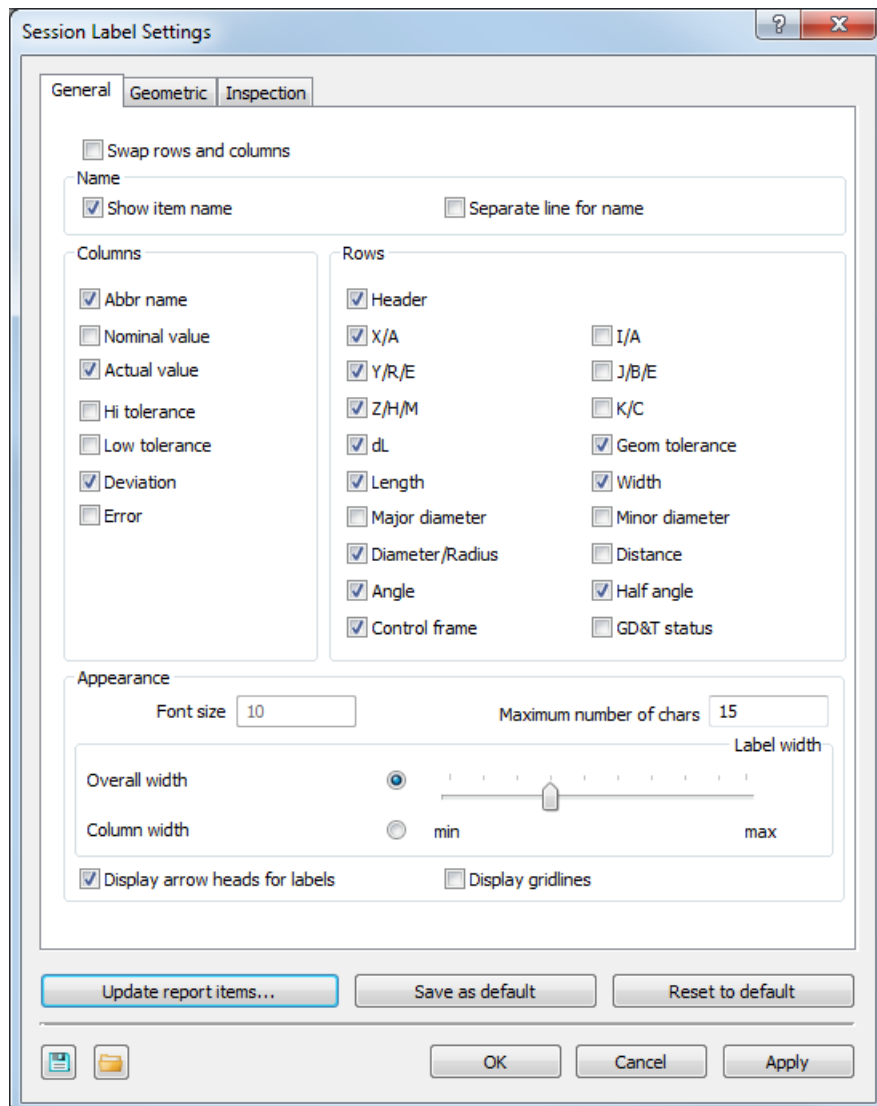


- b Left-click and hold a label, drag it to a new position, and release the mouse button.

- 4 In **Show manual labels** mode, you can also size the labels and choose the columns and rows that are displayed. To do this, click



in the **View Options** toolbar, or right-click a label, and select **Label Settings** from the context menu. The **Session Label Settings** dialog is displayed.



- 5 The dialog enables you to select the data that is displayed in labels and to apply limited formatting. Experiment with changing the settings in this dialog to see the effect on the labels.
- 6 When you have finished, click **Default** to return the dialog to its original settings, then click **OK** to close the dialog.

Displaying detailed feature information

In addition to viewing measurement information in the CAD view, you can see all an item's measurements using the **Info** tab. The tab contains the following areas:

Area	Information displayed
Information	The datum or alignment relative to which the item measurements are reported.
Links	Any sequence items used by the selected item. In the following example, the slot is measured by projecting it onto Plane 4 .
Parameters	The parameters controlling the measurement of the item.
Properties	The position and size of the feature, the tolerance values, and the deviation and error values (as seen in the report). The area also displays the form value of the item when appropriate.
Exported Items	The properties that can be used by other items in the inspection sequence.
List of probed points	The positions, vectors, and deviation from form for each probed point. The Used column indicates whether the point is included in the item's measurement calculations.

To use the **Info** tab:

- 1 Click the **Info** tab at the bottom of the PowerINSPECT window.



- 2 In the inspection sequence, select the item for which you want to display the details. For example:

Slot 1 (Probed Slot)

Information

Datum	<Active Alignment>
-------	--------------------

Links

Name	Link
Reference Plane	Plane 4

Parameters

Name	Used	Offset
Guided Measure	No	-1.000
Name	Value	
Material Side	Hole (ID)	
Fitting Algorithm	Least Square	
Name	Used	Value
Auto accept points	No	5

Properties

	Nominal	Lo-Tol	Hi-Tol	Actual	Deviation	Error
Centre	X	0.000	-0.100	0.100	0.129	0.029
	Y	37.500	-0.100	0.100	37.789	0.289
	Z	-25.000	-0.100	0.100	-25.201	-0.201
Overall length	67.000	-0.100	0.100	67.642	0.642	0.542
Width	12.000	-0.100	0.100	12.130	0.130	0.030
	Value		Calculated			
Offset/Thickness	0.000		No			
	Maximum		Actual		Error	
Profile	0.100		0.030		-	
	Nominal	Lo-Tol	Hi-Tol	Actual	Deviation	Error
Main Axis	I	0.00000	0.00000	0.00000	0.00018	0.00018
	J	1.00000	0.00000	0.00000	-1.00000	-2.00000
	K	0.00000	0.00000	0.00000	0.00084	0.00084

Exported Items

Name	Type	Description	Linked as
Slot 1::Centre	pwi_feature_Point	Slot 1::Centre	
Slot 1::Centre1	pwi_feature_Point	Slot 1::Centre1	
Slot 1::Centre2	pwi_feature_Point	Slot 1::Centre2	
Slot 1::Main Axis	pwi_feature_Line	Slot 1::Main Axis	
Slot 1::Small Axis	pwi_feature_Line	Slot 1::Small Axis	
Slot 1	pwi_feature_RoundedSlot	Slot 1	
Slot 1::Normal Axis	pwi_feature_Line	Slot 1::Normal Axis	

☐ List of probed points (probe centre) CAD coordinates ▼

As with labels, when the measurements are compared to the item's nominals, different colours are used to indicate whether the measurements are above (red text), below (blue text) or within tolerance (green text).


- 3 To display the details of the probed points used to measure the feature, select the **List of the probed points** check box. For example:

☒ List of probed points (probe centre) CAD coordinates

	X	Y	Z	I	J	K	Deviation	Used
Probe diameter: 6.000 mm								
Point 1	2.025	68.596	-25.521	-0.50820	-0.76843	0.38890	0.000	Yes
Point 2	1.462	6.983	-24.880	-0.90338	-0.12931	0.40887	0.000	Yes
Point 3	1.547	20.826	-22.109	-0.90147	-0.22654	0.36882	-0.015	Yes
Point 4	1.421	53.304	-22.166	-0.85911	-0.39249	0.32846	0.015	Yes
Point 5	1.735	53.720	-28.267	-0.82833	-0.41891	0.37201	0.015	Yes
Point 6	3.048	20.775	-28.270	-0.83782	-0.32906	0.43563	-0.015	Yes

To specify the location relative to which the values are measured, select an entry in the drop-down list.

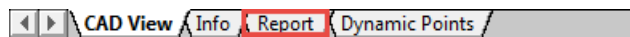


To print the contents of the **Info** tab, click  on the **Main** toolbar.

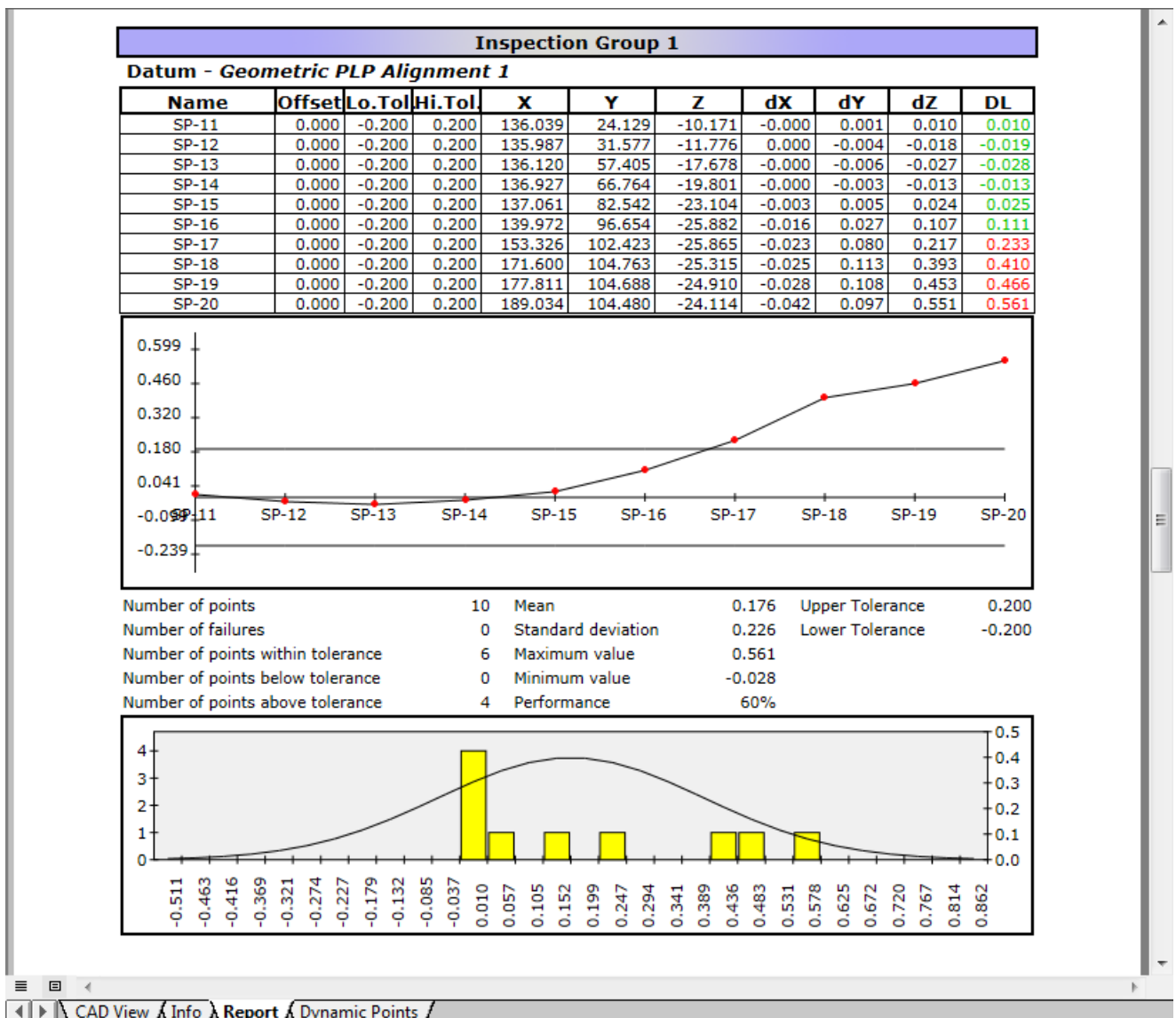
The report

The **Report** tab enables you to create a detailed breakdown of the measurements for all items in an inspection. You can display the information using the report templates supplied with PowerINSPECT, or you can create your own templates and use different layouts for different applications.

To display the measurement results in a report, click the **Report** tab at the bottom of the PowerINSPECT window:



The measurement results are displayed. Use the scroll bar on the right of the report to display all the details.



Reports can be made customizable. This enables you to specify inspection-specific information, such as the part number and the name of the person who carried out the inspection.

To customize your report:

- 1 Select the **Measures > Parameters** menu option.
- 2 In the **Measure Parameters** dialog, click the **Report** tab.

The screenshot shows the 'Measure Parameters' dialog box with the 'Report' tab selected. The dialog has a title bar with a question mark and a close button. Inside, there are four tabs: 'Part Compensation', 'GD & T', 'Measured Data Scaling', and 'Report'. The 'Report' tab is active, showing a 'Template File' field with the path 'D:\templates\HTML\Reports\StandardReport.pxm' and a 'Browse' button. Below this is a table with two columns: 'Name' and 'Value'. The table contains six rows: 'Customer' (Your customer company name here), 'Customer contact' (Your contact person), 'Customer fax No.' (Your customer fax No.), 'Customer phone No.' (Your customer phone No.), 'Datum' (Your Datum), and 'Description' (Your part description here). Below the table are two buttons: 'Reset to defaults' and 'Options'. At the bottom left is a checkbox labeled 'Save as default parameters'. At the bottom right are three buttons: 'OK', 'Cancel', and 'Help'.

Name	Value
Customer	Your customer company name here
Customer contact	Your contact person
Customer fax No.	Your customer fax No.
Customer phone No.	Your customer phone No.
Datum	Your Datum
Description	Your part description here

- 3 Click the **Browse** button, select **HTML** in the **Open File** dialog, followed by the folder in which the template is located, and then select the .pxm file for the required template. Click the **Open** button.
- 4 Click **Reset to defaults** to refresh the report screen and extract the variables into the report header.
- 5 Change the text in the **Value** column as required.
- 6 Select the **Save as default parameters** check box to set the currently selected template as the default.
- 7 Click **OK** to close the dialog and update the report.
- 8 Display the report again, and check it contains the changes you made.

Controlling the information included in a report

PowerINSPECT offers total flexibility in the display of geometric groups and the attributes of features within a group. All geometric items, with the exception of planes, are displayed by default, and so are the major attributes of the displayed items.



*Each **CAD View Report** item is displayed in the inspection report, as well as in the **Info** tab (see page 43).*

Excluding features contained in a geometric group

You can exclude items from the report using the item's dialog. Similarly, you can exclude all the items in a group using the group's dialog.

For example, to exclude a geometric group from the report:

- 1 Select Geometric Group 1 in the inspection sequence, and click



the button at the top of the **Sequence Tree**. The **Geometric Group** dialog is displayed.

- 2 Deselect the **Output in report** check box to exclude all features in the group from the report.
- 3 Click **OK** to save your changes and close the dialog.

Excluding measurements contained in a feature

In addition to excluding whole items from the report, you can exclude individual measurements. This is useful if you want to highlight only the important measurements for an item.

For example, to exclude measurements from a circle:

- 1 Double-click **Circle 1** in the inspection sequence. The **Probed Circle** dialog is displayed.

Probed Circle

Name: Circle 1

☒ Use nominals

Visible ☒
Output in report ☒

Page 1 | Page 2 | Point Sources(0)

Coordinate System: <Active Alignment>

-89.995 270.005

Material Side: Hole (ID)

Fitting Algorithm: Least Square

Reference Plane: Plane 1

Centre

Coordinate Type: Cartesian

	Nominal	Low Tol	High Tol	
X	-56.488 mm	-0.100	0.100	<input checked="" type="checkbox"/>
Y	22.639 mm	-0.100	0.100	<input checked="" type="checkbox"/>
Z	0.000 mm	-0.100	0.100	<input checked="" type="checkbox"/>

Diameter

☐ Radius ☒ Diameter

	Nominal	Low Tol	High Tol	
Diameter	13.002	-0.100	0.100	<input checked="" type="checkbox"/>

Offset/Thickness

-0.000 ☐ Calculate from measurement



Circularity

Maximum 0.100

Comment

OK Apply Cancel Help

- 2 In the **Centre** area, deselect the **Output in report** check box to the right of the **Z** coordinate to exclude the coordinate from the report and the circle's label in the CAD view.

Alternatively, to exclude all the **Centre** coordinates, click the **Output in report**  button. The icon changes to . Click the button again to reinstate the measurement in the report.



PowerINSPECT ignores excluded measurements when determining whether an item is in- or out-of-tolerance. This can affect the status icon shown in the inspection sequence and the border colour of the item's label.

- 3 When you have made your selections, click the **Apply** button to save your changes and leave the dialog open.




*Deselecting the **Output in report** check box for a measurement also excludes it from the item's label. You can confirm whether a measurement is included in the report by clicking the **Show/Hide Labels** button in the **View Options** toolbar, and checking the item's label (see page 40) in the CAD view.*

- 4 Click **OK** to save your changes and close the dialog.

Printing the report

To print the results displayed in the **Report** tab:



- 1 Click  to open the **Print Manager** dialog.
- 2 Select the required printer and click the **Print** button.

The results are output to the selected printer using the currently selected report template.