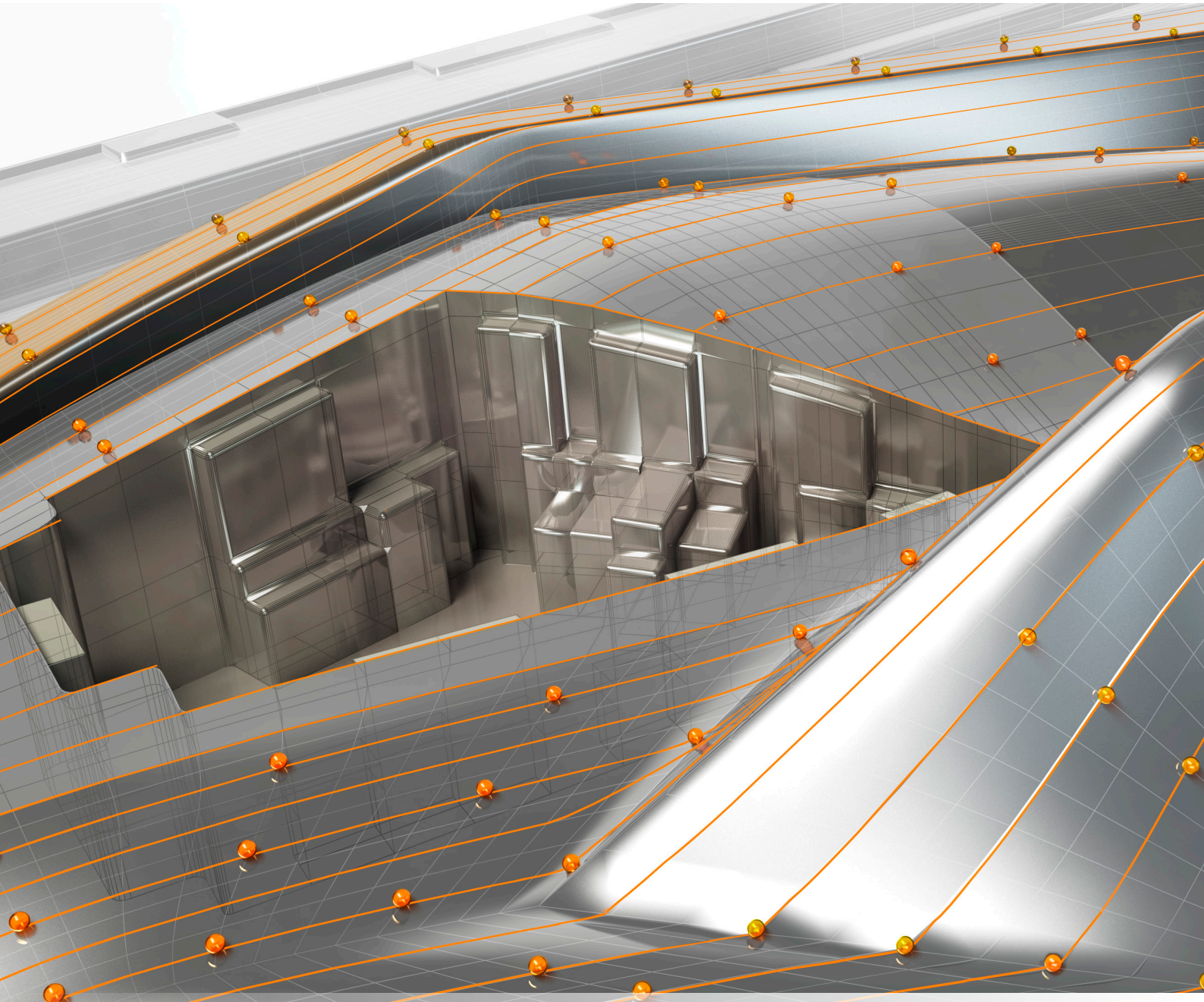
AUTODESK® **POWERINSPECT®**

Inspection for every environment

Training Course



AUTODESK.

Important Notice

Welcome to Autodesk.

This document is supplied as part of an Autodesk Training Course. It is not intended to be distance-learning material: rather as an aid for Tutors when presenting material to course delegates and as a subsequent aid memoir to those delegates.

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Contents

1. Introduction

PowerInspect is a CAD based inspection solution package for use with many types of inspection hardware, including manual and CNC CMMs, portable arms, optical measuring devices and CNC machine tools (OMV).

PowerInspect allows you to rapidly create a complete inspection, and automatically generate a high quality report using either HTML format or Microsoft Excel. The inspection reports can be customised to include pictorial, tabulated and statistical data, meeting the demands of design requirements and agreed international standards.

You are given instant results during the inspection, and the onscreen reaction and detailed graphical displays give immediate feedback for each measured point.

PowerInspect supports models from neutral formats such as IGES. If the relevant Exchange translators are purchased, PowerInspect can import data created by third party software packages, such as CATIA, Unigraphics, SolidWorks and IDEAS.

The user interface is intuitive and easy to use, and the learning curve is short, allowing the operator to get the most from the measuring device in the shortest possible time.

The examples given in these training notes are specific to the demo block model used in the training course. The techniques should be learnt and adapted to your own requirements.

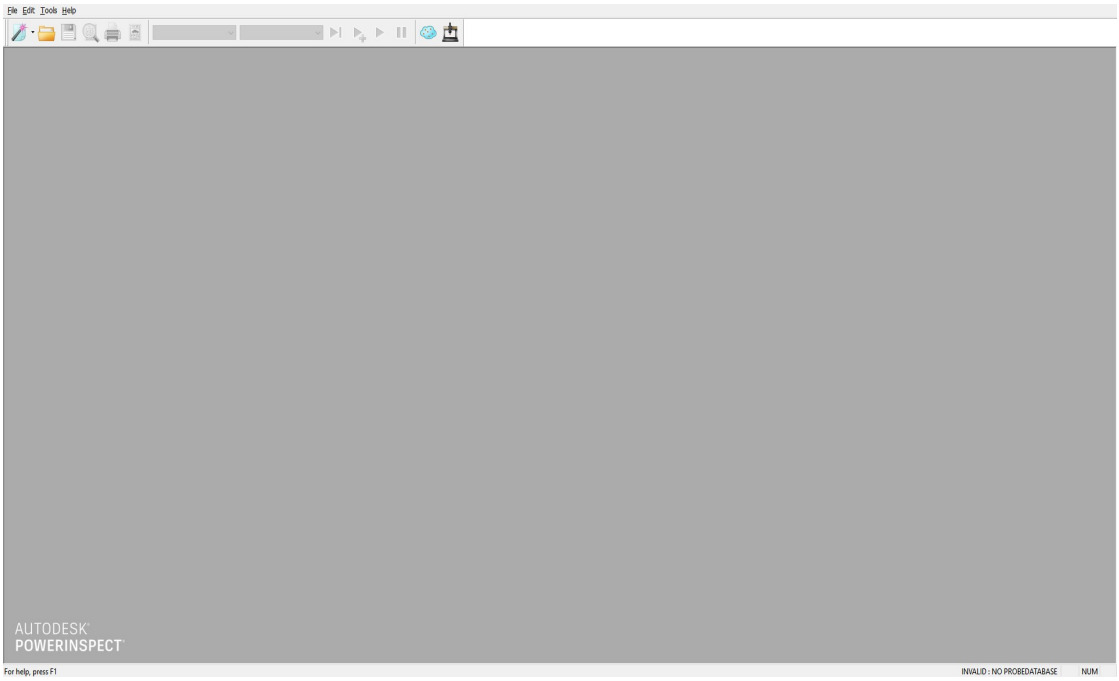
PowerInspect gives you:

- Comparison against all mainstream CAD formats.
- Rapid alignment, even for complex free form shapes.
- Simple and rapid measurements between features.
- The ability to inspect user-defined sections.
- The ability to inspect along the edge of a part.
- Full geometric inspection capabilities.
- Automatic creation of inspection features from CAD nominals.
- The ability to apply geometric dimensioning and tolerance (GD&T).
- Point, wireframe and surface export for measured entities.
- CAD manipulation, including surface offsetting (for example, the reverse side for sheet metal, or a spark gap for electrodes).
- Measurement without CAD data.

1.1 Opening PowerInspect

To open PowerInspect, double-click the desktop icon  .

The screen should look as follows:



The work environment remains empty until you start a new session. When a new session is started, further options become available in the Menu bar, and a graphics window, Sequence Tree and context-sensitive toolbar are generated.

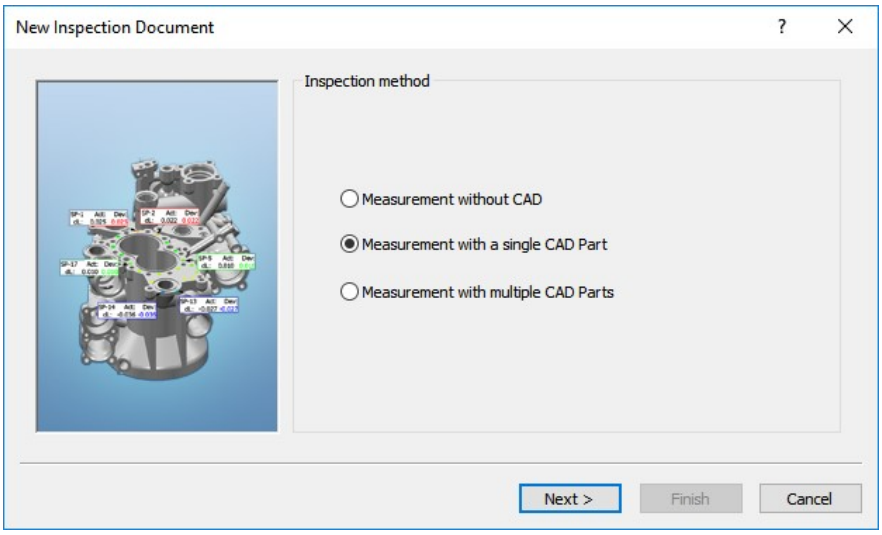
1.2 Creating a new session

A new session can be started from the **File** menu or from the **Main** toolbar.

- **New Session**  creates a new inspection document without loading a CAD model.
- **New Session Wizard**  allows you to choose whether you want to open one or more CAD models. When you start a new session using the **New Session Wizard**, you are guided through the process step by step.
- If a PowerInspect session (*.pwi) already exists, you can select **File > Open** from the **Main** menu. Alternatively, click **Open**  on the **Main** toolbar.

New Session Wizard

1 Click the **New Session Wizard**  button to open the **New Inspection Session** dialog.

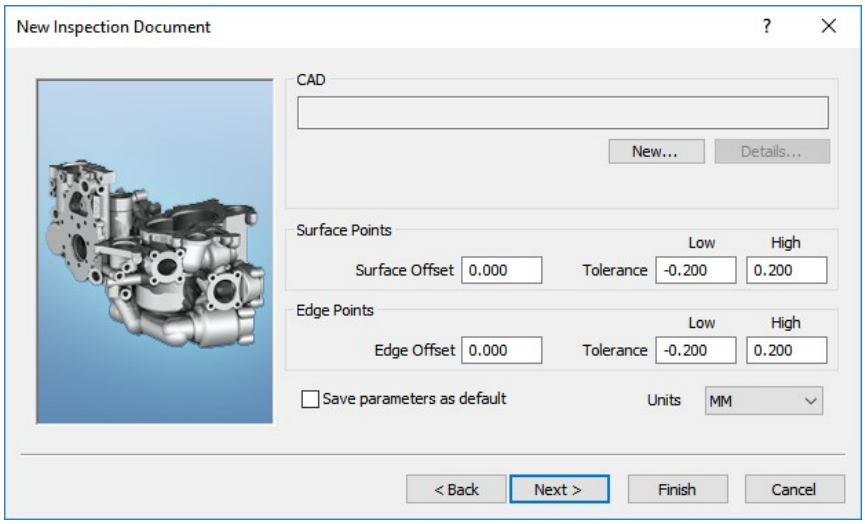


This dialog contains three options for creating an inspection session:

- **Measurement without CAD** creates an inspection session without opening a CAD model.

  This is the same as selecting **New Session**  from the **Main** toolbar.
- **Measurement with a single CAD Part** creates an inspection session using one CAD model.
- **Measurement with multiple CAD Parts** creates an inspection session using multiple CAD models.

2 Select **Measurement with a single CAD part** and click **Next** to display the second page of the wizard.



Use this dialog to browse to the CAD file that you want to open. You can also specify the Units you want to work in, and specify offsets for surface and edge points.

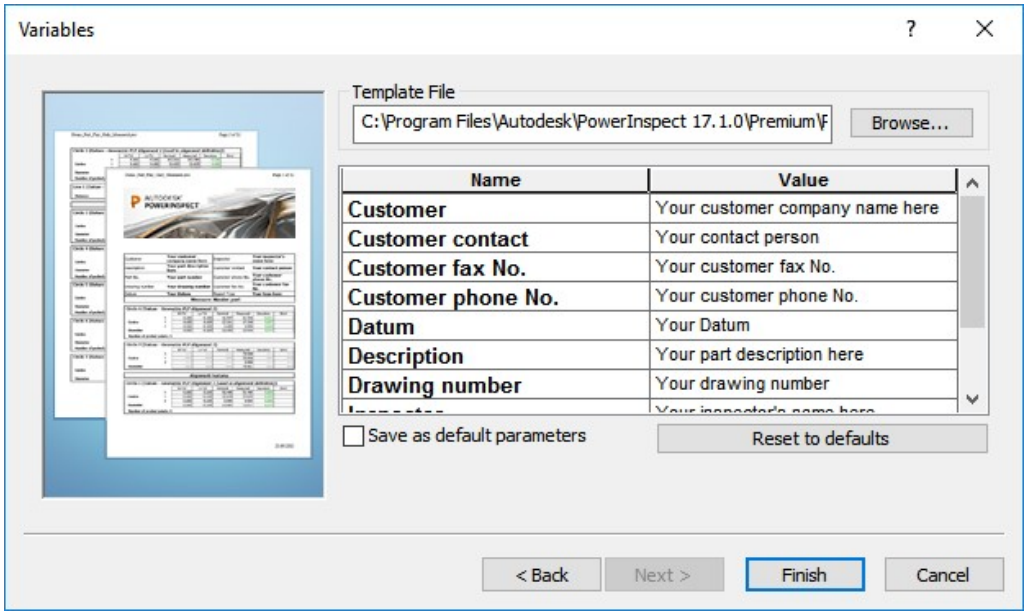
3 Click **New** to browse to the CAD file for the part you want to inspect.

4 In the **Open** dialog, select DemoBlock2008(CMM+Arm).dgk and click **Open**.

*After a CAD file has been selected, it can be transformed. Click **Details** to display the **CAD Details** dialog and select **Transformation Matrix** (for further details, see Chapter 2).*

5 Click **Next** to display the next page of the wizard, **Variables**.

The **Variables** dialog allows you to choose the template in which you want to report your measurements. Click **Browse** to navigate to the template file you want to use.



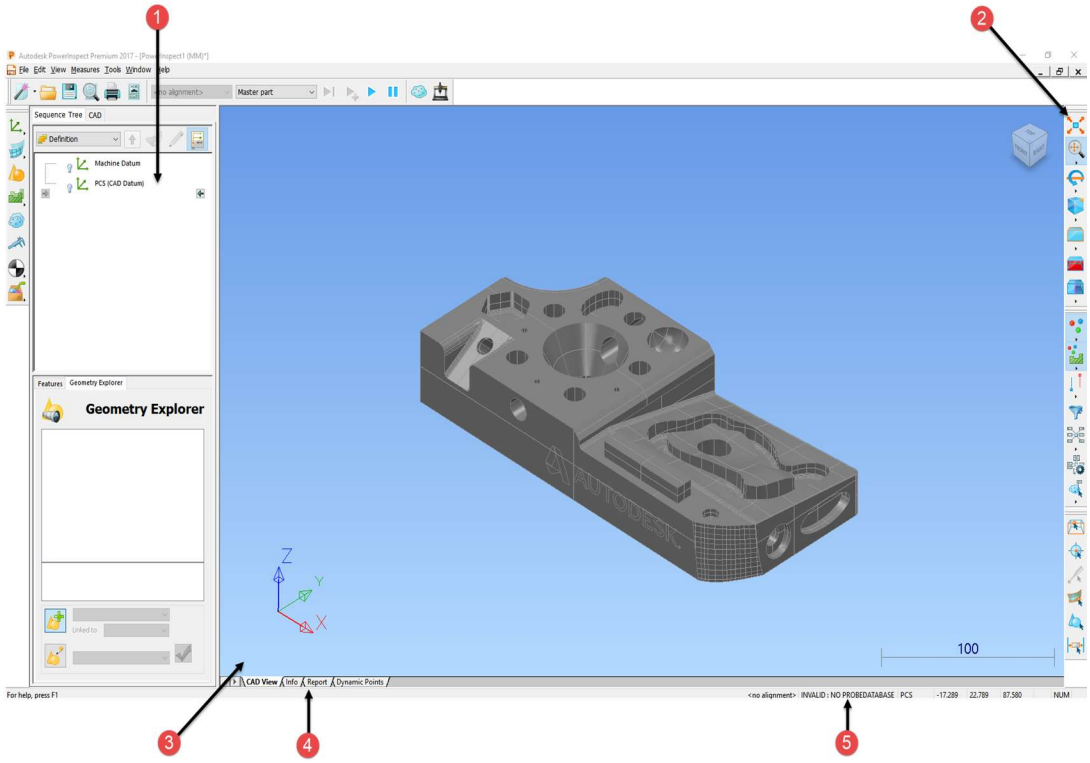
PowerInspect has the ability to create reports in two formats:

- **HTML format** reports are created directly inside the PowerInspect *.pwi file, and are accessible through the **Report** tab below the graphics window.
- **Microsoft Excel** reports allow backwards compatibility for older versions.
- Entries can be edited by clicking on the required field and editing the contents.

6 Click **Finish** to complete the process and begin the session.

The Graphics Window

When a session is open, a number menus and toolbars are displayed, as shown below.



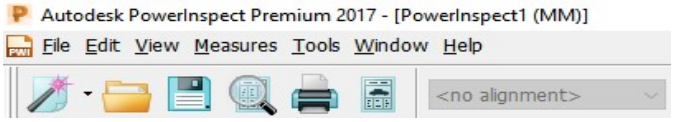
1. Sequence tree
2. Context-sensitive toolbars area
3. Graphics window
4. Area selection tabs
5. Probe status bar

The Graphics window includes the following tabs:

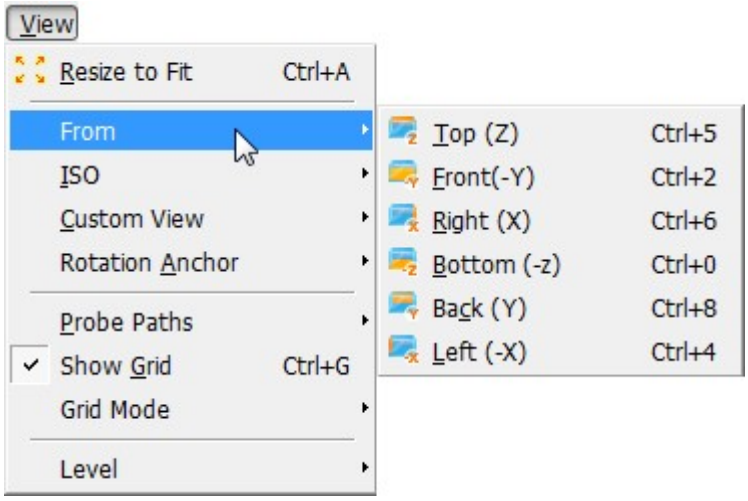
- **CAD View displays** the CAD model and all inspected results.
- **Info** displays information about a selected Sequence Tree item.
- **Report** displays the report in HTML format.
- **Dynamic Points** lists the points used to create a free form alignment or guided inspection.
- **Section** displays the section each time you create a Section group. This tab is only available if you have created Section group.
- **Point Cloud** displays the points in the selected point cloud, together with a graph and selected statistics.
- **Digitised Points** displays the points in the selected Digitised Curve item.

Menu bar

The **Menu** bar is located at the top of the main window.



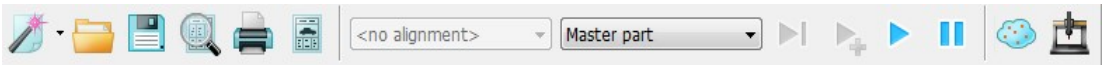
Click on a menu item to open it. If a menu item is unavailable, it does not apply to the currently active document. A small arrow next to a menu item indicates that there are more options available. Move the cursor over the menu item to display these options.



Click outside a menu to cancel any command.

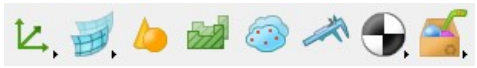
Main toolbar

The **Main** toolbar is displayed at the top of the graphics window under the Menu bar. The buttons allow easy access to the most frequently used PowerInspect menu items.



Item toolbar

The **Item** toolbar is used for the creation of alignments, geometric groups, inspection groups, and datums.

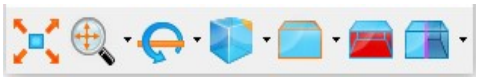


The toolbar changes according to the function chosen.

Click on an icon with a small arrow next to it to open that item's submenu. For example, clicking the **Miscellaneous**  button displays the following toolbar:

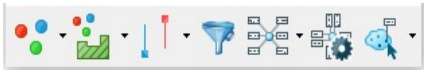
CAD View toolbar

The **CAD View** toolbar contains options to orientate and shade the CAD model.



View Options toolbar

The **View Options** toolbar enables you to display measurement data in the graphics window, and to control the information that is displayed.

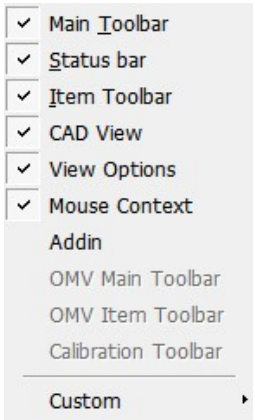


Mouse Context toolbar

The **Mouse Context** toolbar enables you to choose the function of the cursor.

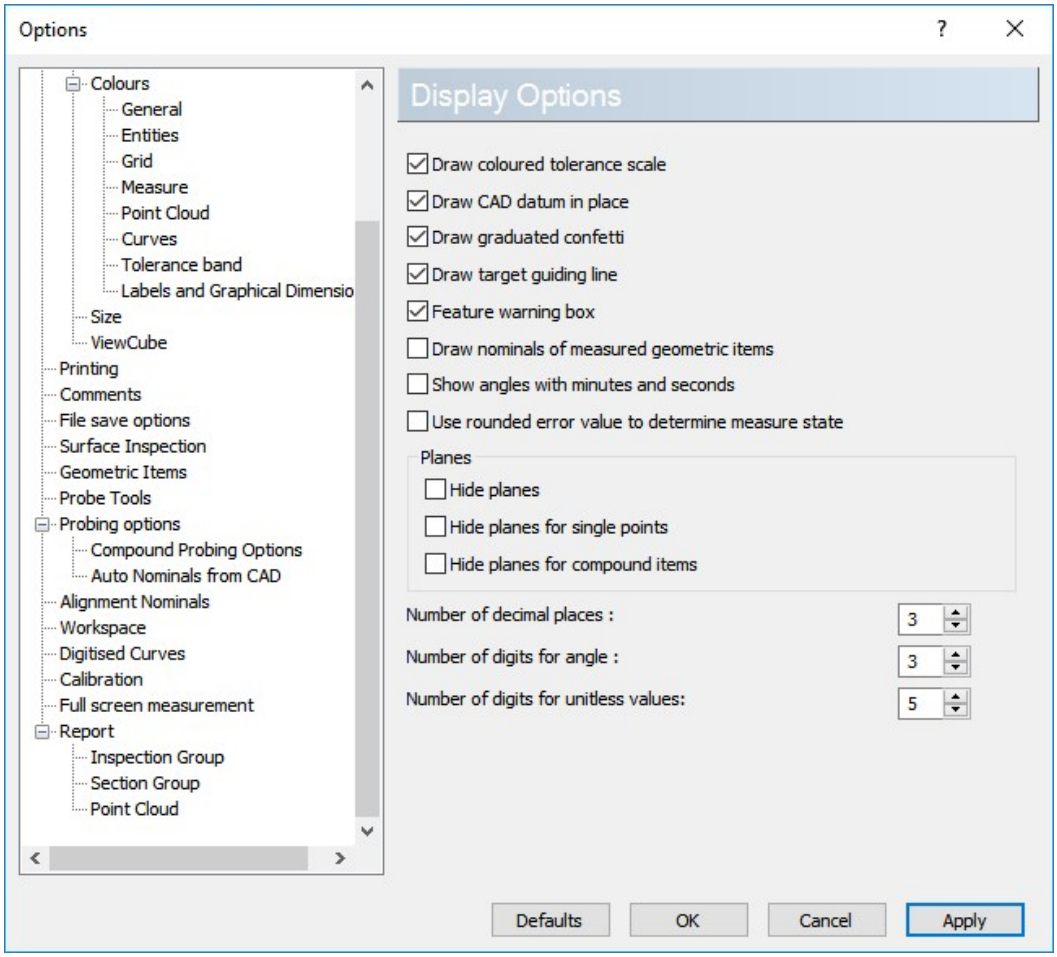
Toolbar visibility

Right-click in and empty area of the toolbar or menu bar to toggle the visibility of all toolbars.



Tools > Options

Select **Tools > Options** from the **Menu** bar to specify settings for various PowerInspect features.



Views and View Manipulation

After the new session has been created, you can view the CAD model in order to gain some visual information about the part. To do this, the various views and their manipulation need to be understood.

Before running through the views and view manipulation, the use of the mouse functions will be established.

Mouse buttons

Each of the three mouse buttons performs a different dynamic operation.

- **Left mouse button: Picking and selecting**

This button is used for selecting items from the menus, options within dialogs, and items in the graphics area.
It is also responsible for view manipulation depending on the setting used in the **Set Mouse Button 1 View Mode**  button.

- **Middle mouse button: Dynamics**

- Zooming in and out: Hold down the **Ctrl** key and the middle mouse button. Move the mouse up and down to zoom in and out. Alternatively, rotate the scroll wheel, if available.
- Pan around the model: Hold down the **Shift** key and the middle mouse button. Move the mouse in the required direction.
- Zoom box: Hold down the **Ctrl** and **Shift** keys, and drag a box around the area that you want to zoom into using the middle mouse button.
- Rotate mode: Hold down the middle mouse button and move the mouse to rotate the view.

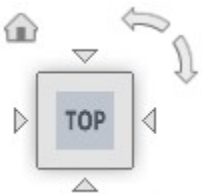
- **Right mouse button: Dynamics, Special Menus & PowerInspect Sequence Tree Options**

- Zooming in and out: Hold down the **Ctrl** key and the right mouse button. Move the mouse up and down to zoom in and out.
- Pan around the model: Hold down the **Shift** key and the right mouse button. Move the mouse in the required direction.
- Zoom Box: Hold down the **Alt** key and the right mouse button. Drag a box around the area that you want to zoom into.
- Rotate mode: Hold down the **Ctrl** and **Shift** keys, and use the right mouse button to rotate the view.

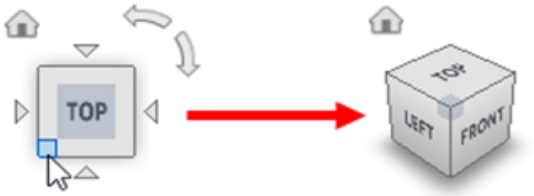
When the right mouse button is pressed on its own, a popup menu is displayed based on the item that the mouse is over, such as the Sequence Tree, or the toolbar menus.

Graphics Area

This is the large, visual display area to the right of the **Sequence Tree** where the model is displayed.
In the top right corner of the Graphics area is the **ViewCube** as shown below.



This is used to quickly select a range of standard viewing orientations.
For an **ISO1** view click the lower left corner of the **ViewCube**.



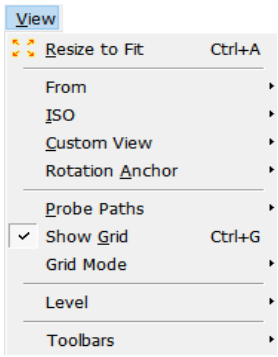
To return to the default **Top** view select the **Home** symbol to the top left of the **ViewCube**.

Another method would have been to select the central top area.

To enable 90 incremental rotation on the planer views select one of the circular arrows.

View menu

The **View** menu is accessed from the **Menu** bar and contains the **Resize to Fit** option, and a number of **View** submenus. Each of these submenus corresponds to a different group of views.



- **From** — these options allow you to alter the viewing angle of the part shown in the CAD view.
- **ISO** — these options allow you to alter the CAD view layout.
- **Custom View** — these options allow you to save the current CAD view for future use and to reload previously saved views.
- **Rotation Anchor** — these options allow you to specify how the model in the CAD view is rotated.
- **Probe Paths** — these options are only applicable for the CNC version of the software. They allow you to control which probe paths can be seen in the CAD view.
- **Grid Mode** — these options allow you to specify the coordinate system in which the grid is displayed.
- **Level** — These options allow you to group surfaces into levels and to control which levels are displayed in the CAD view
- The **View** flyout shares these functions.

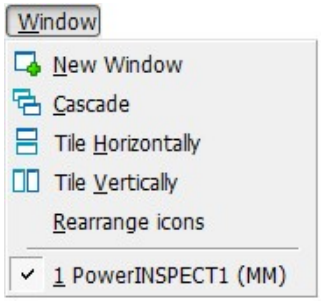
View flyout

Open the **View** flyout by clicking the arrow below the **Select View**  button on the **CAD View** toolbar. This flyout contains a series of buttons that correspond to the same icons in the view menu.

The **1**, **2**, **3**, and **4** buttons are used to select the **Custom Views**, which can be saved using the corresponding **Save**  icon. Rotations can also be triggered using the **Arrow** keys on the keyboard, by dynamically moving the cursor using the **Mouse Functions**, or by using the **Rotations**  flyout. The rotation axis is selected from the flyout and the view then rotated incrementally by clicking on the icon.

Window menu

If more than one session is open, you can view all sessions in a single screen, using the **Window** menu. The sessions can then be arranged according to your preference.



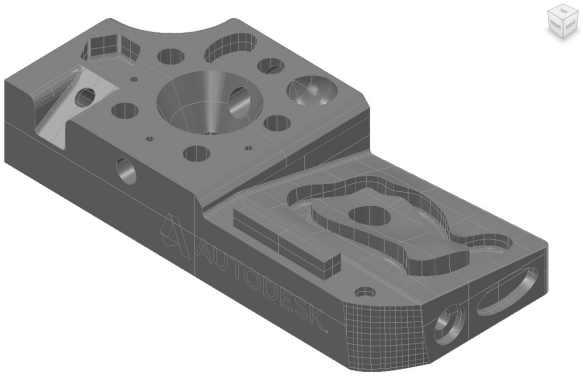
The CAD View

The CAD View is where all the inspection and geometric information is viewed. CAD models can be seen either with no shading, transparent shading, or solid shading, combined with or without wireframe.

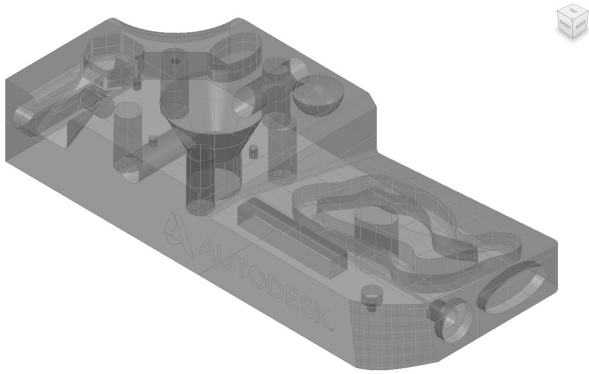
Click the **Shading Mode For Model**  button to open the **Shading Mode**  flyout and choose the shading you want to apply to the CAD model. Alternatively, use the shortcut keys to toggle the wireframe and shading.

- **w** — toggle wireframe on/off
- **s** — toggle shading on/off

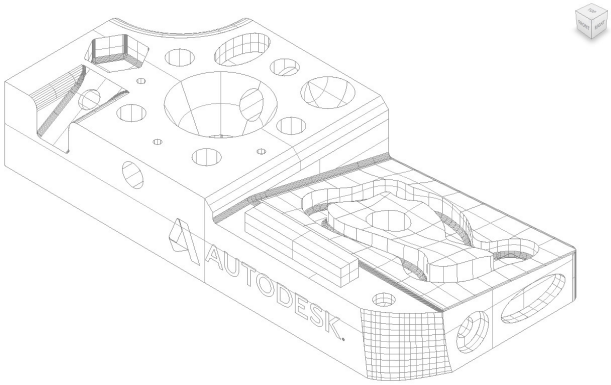
Solid Shading



Transparent Shading



Wireframe Shading



Surface normal direction

PowerInspect has a function which allows you to determine the surface normal direction.

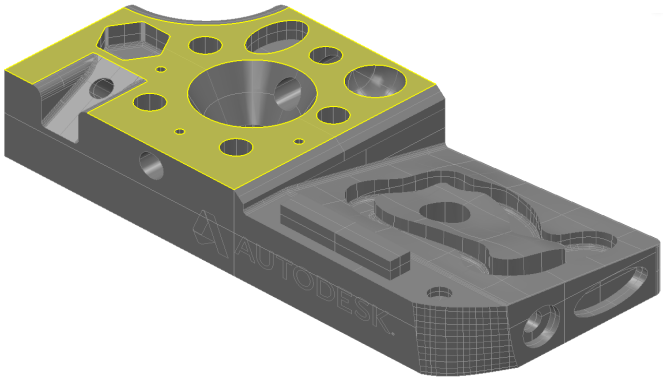
Using the **Highlight back-facing surfaces** button in combination with the shading, will show all reversed surfaces as bright red. This is useful for determining the direction needed for surface offsets.

To reverse a surface:

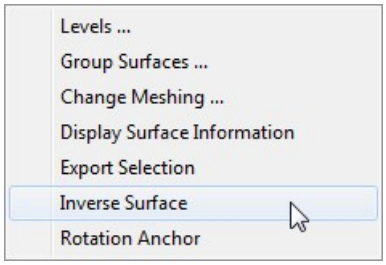
1 Click **Highlight back-facing surfaces** .

2 Click **Surface Selector** .

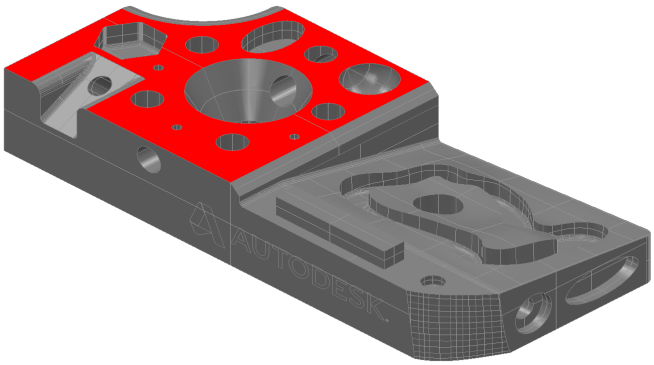
3 Click a surface to select it. The selected surface is highlighted yellow.



4 Right-click in the graphics window and select **Inverse Surface** from the popup menu.

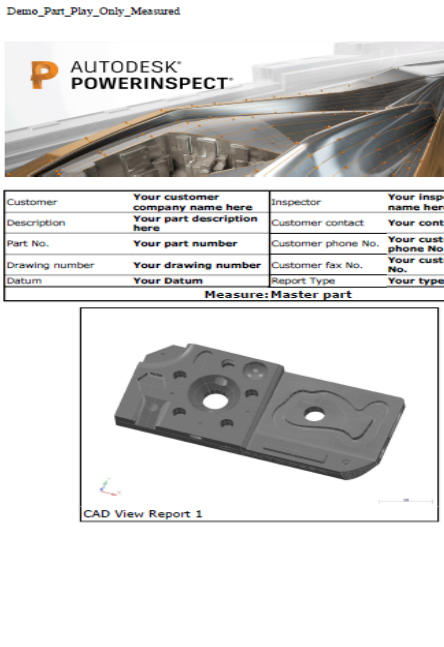


The selected surface changes colour to red to demonstrate that it has been reversed.



The Report Template

PowerInspect allows you to select the report template you want to output your data to. If using the HTML format, the report is integrated into the PowerInspect work environment, and can be accessed at any time using the **Report** tab. These reports can be customised to suit the customer's needs.



Demo_Part_Play_Only_Measured Page 1 of 12

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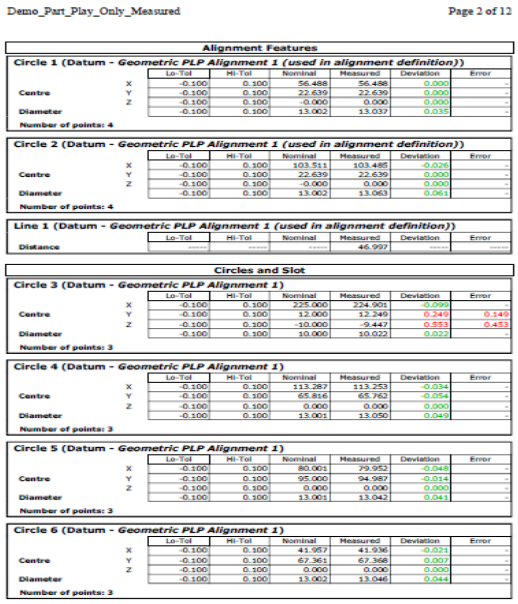
Customer	Your customer company name here	Inspector	Your inspector's name here
Description	Your part description here	Customer contact	Your contact person
Part No.	Your part number	Customer phone No.	Your customer phone No.
Drawing number	Your drawing number	Customer fax No.	Your customer fax No.
Datum	Your Datum	Report Type	Your type here

Measure: Master part

CAD View Report 1

27/07/2016



Demo_Part_Play_Only_Measured Page 2 of 12

Alignment Features

Circle 1 (Datum - *Geometric PLP Alignment 1 (used in alignment definition)*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	56.488	56.488	0.000	-
	Y	-0.100	0.100	22.639	22.639	0.000	-
	Z	-0.100	0.100	-0.000	0.000	0.000	-
Diameter		-0.100	0.100	13.002	13.037	0.035	-

Number of points: 4

Circle 2 (Datum - *Geometric PLP Alignment 1 (used in alignment definition)*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	103.511	103.485	-0.026	-
	Y	-0.100	0.100	22.639	22.639	0.000	-
	Z	-0.100	0.100	-0.000	0.000	0.000	-
Diameter		-0.100	0.100	13.002	13.063	0.061	-

Number of points: 4

Line 1 (Datum - *Geometric PLP Alignment 1 (used in alignment definition)*)

	Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Distance	-----	-----	-----	46.997	-----	-----

Circles and Slot

Circle 3 (Datum - *Geometric PLP Alignment 1*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	225.000	224.901	-0.099	-
	Y	-0.100	0.100	12.000	12.249	0.249	0.149
	Z	-0.100	0.100	-10.000	-9.447	0.553	0.453
Diameter		-0.100	0.100	10.000	10.022	0.022	-

Number of points: 3

Circle 4 (Datum - *Geometric PLP Alignment 1*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	113.287	113.253	-0.034	-
	Y	-0.100	0.100	65.816	65.762	-0.054	-
	Z	-0.100	0.100	0.000	0.000	0.000	-
Diameter		-0.100	0.100	13.001	13.050	0.049	-

Number of points: 3

Circle 5 (Datum - *Geometric PLP Alignment 1*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	80.001	79.952	-0.048	-
	Y	-0.100	0.100	95.000	94.987	-0.014	-
	Z	-0.100	0.100	0.000	0.000	0.000	-
Diameter		-0.100	0.100	13.001	13.042	0.041	-

Number of points: 3

Circle 6 (Datum - *Geometric PLP Alignment 1*)

		Lo-Tol	Hi-Tol	Nominal	Measured	Deviation	Error
Centre	X	-0.100	0.100	41.957	41.936	-0.021	-
	Y	-0.100	0.100	67.361	67.368	0.007	-
	Z	-0.100	0.100	0.000	0.000	0.000	-
Diameter		-0.100	0.100	13.002	13.046	0.044	-

Number of points: 3

27/07/2016

The Information Tab

The **Info** tab allows you to view specific items for positional data, tolerances, deviations and errors. This can be displayed by selecting the item to be investigated from the Sequence Tree, and then selecting the **Info** tab. The following is then seen:

Circle 1 (Probed Circle)

Information

Datum	<Active Alignment>

Links

Name	Link
Reference Plane	Plane 1

Parameters

Name	Start Angle	End Angle
Angle Quadrant	-270.005°	89.995°

Name	Value
Material Side	Hole (ID) - Only for CNC

Name	Used	Offset
Guided Measure	No	-1.000

Name	Value	
Fitting Algorithm		Least Square
Orientation	I	-0.83986
	J	0.54280
	K	0.00000

Name	Used	Value
Auto accept points	No	3

Properties

		Nominal	Lo-Tol	Hi-Tol	Actual	Deviation	Error
Centre	X	56.488	-0.100	0.100	56.488	0.000	-
	Y	22.639	-0.100	0.100	22.639	0.000	-
	Z	0.000	-0.100	0.100	0.000	0.000	-
Diameter		13.002	-0.100	0.100	13.037	0.035	-

	Value	Calculated
Offset/Thickness	0.000	No

	Maximum	Actual	Error
Circularity	0.100	0.006	-

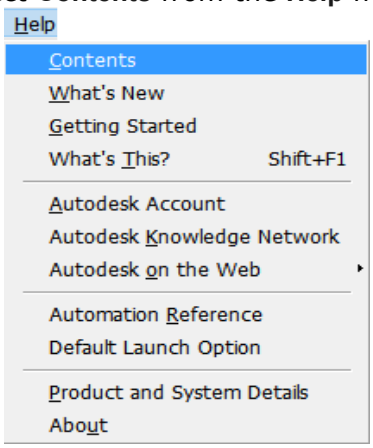
Exported Items

Name	Type	Description	Linked as
Circle 1::Centre	pwi_feature_Point	Circle 1::Centre	Line 1::Reference Point 1 Geometric PLP Alignment 1::Reference Point 1 Circle 7::Reference Points
Circle 1	pwi_feature_Circle	Circle 1	
Circle 1::Normal Axis	pwi_feature_Line	Circle 1::Normal Axis	

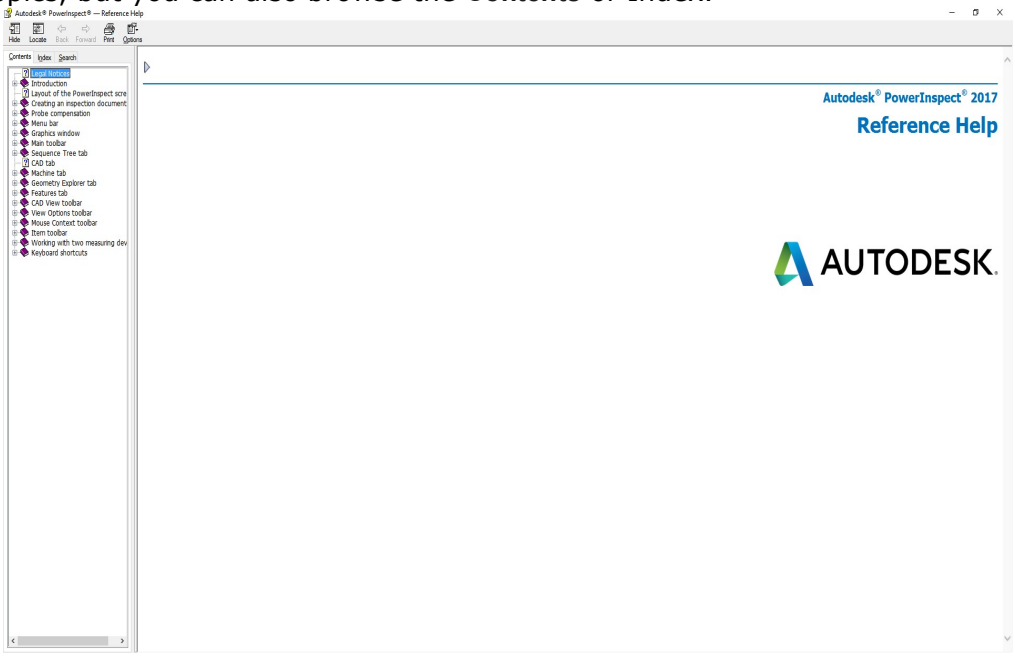
PowerInspect Help

PowerInspect comes with an online help which can be accessed from the **Help** menu. When faced with a problem, it is best practice to make the **Help** menu your first port of call.

To access the online help, select **Contents** from the **Help** menu.



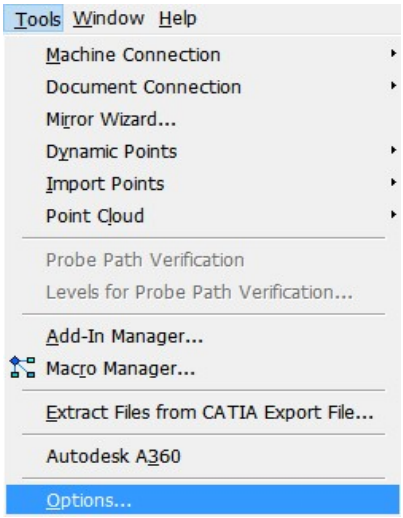
The most effective way to use the Reference Help is to use the **Search** tab to search for help topics, but you can also browse the **Contents** or Index.



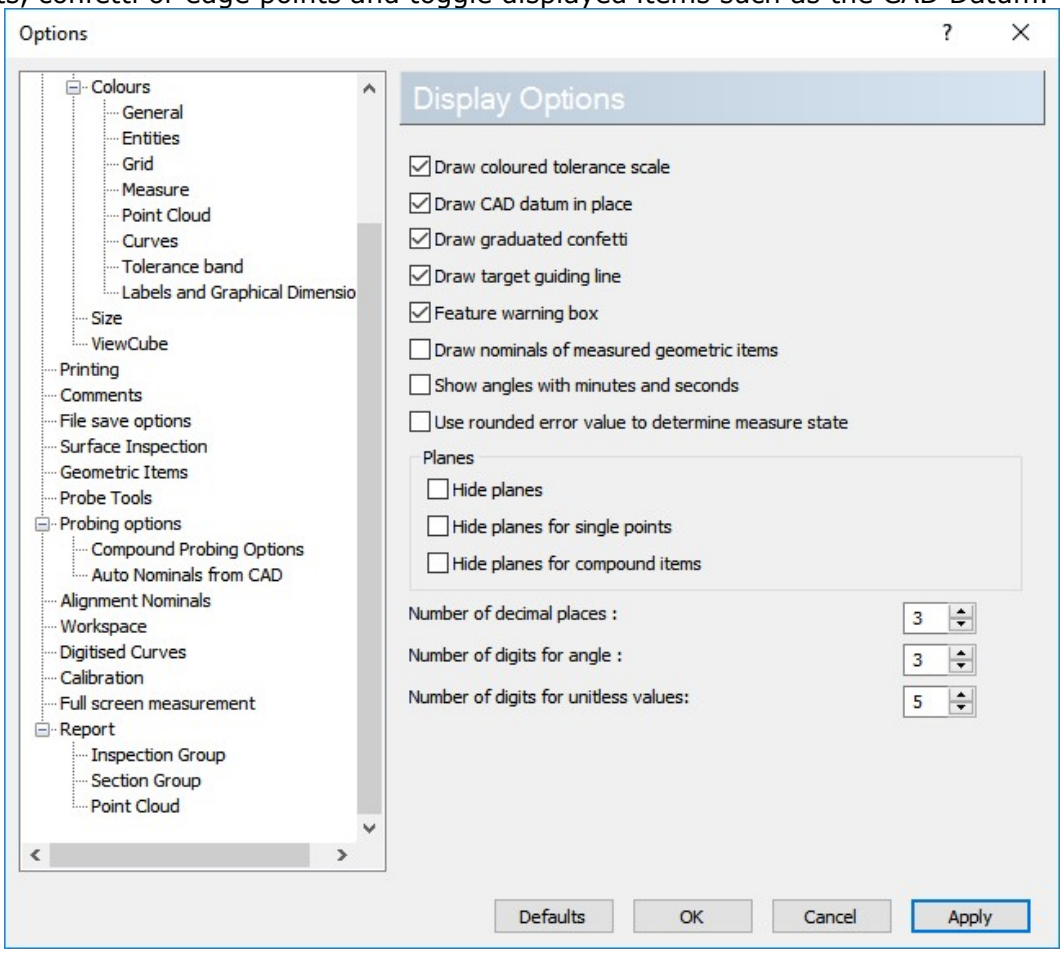
To target your search more effectively, enclose search terms within quotation marks. For example, to search for help on exporting point clouds, enter "Export point cloud" in the search box so that only topics containing this exact phrase are displayed.

Customising PowerInspect

Select **Tools > Options** to display the **Options** dialog and specify settings and defaults for PowerInspect.



Use the **Options** dialog to specify settings including, the colour of entities, the size of points, confetti or edge points and toggle displayed items such as the CAD Datum.

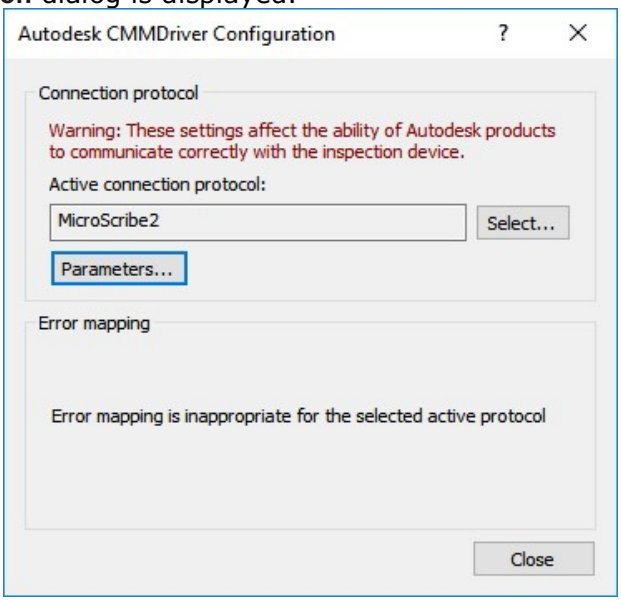


Device Connection

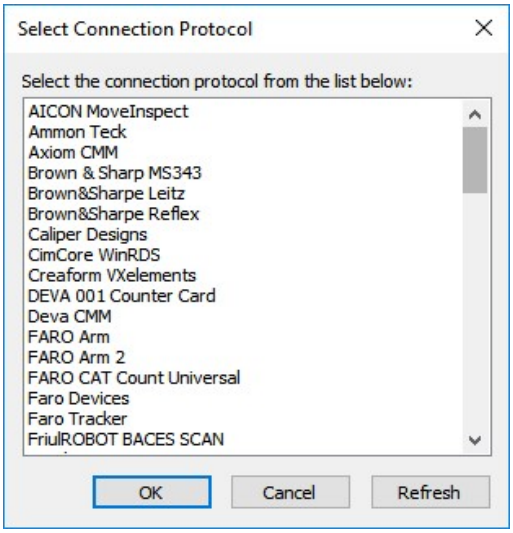
The first time you open PowerInspect, the **CMMDriver Configuration** dialog is displayed. Before loading up, an active connection needs to be made to an inspection device.

The **CMMDriver Configuration** dialog can also be opened manually from the **Main** menu.

1 From the Main menu, select **Tools > Machine Connection > Configure**. The **CMMDriverConfiguration** dialog is displayed.



2 Click **Select** to display the **Select Connection Protocol** dialog, which contains a list of connection protocols.



3 Select the required protocol for the system that PowerInspect is to connected to and click **OK**.

The correct connection protocol should now be set up.

Do not alter the connection protocol unless you are an experienced user. The original installation should have a connection already set up for your machine.

Starting Set-up

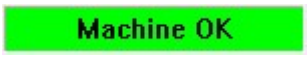
Click the **Toggle the Machine toolbar**  button on the **Main** toolbar to display the **Machine** toolbar.

Use this toolbar to:

- **Connect to**  the machine.
- **Home**  the machine.
- Open the **Tracking Box**  .
- **Change Probe Tool**  .
- View the connection status of the CMM.
- Making a Connection
- To connect a CMM to PowerInspect:

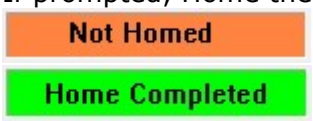
1 Click the **Toggle the Machine toolbar**  button on the **Main** toolbar to display the **Machine** toolbar.

2 Click the **Connection**  button. Alternatively, select **Tools > Machine Connection > Connection** from the **Main** menu.
If the connection is successful the status changes to **Machine OK**.



For a manual CMM, you are prompted to move the probe head to A = 0, B = 0.
You may also be prompted to move the CMM to the home position. In most cases, this means moving all axes to their positive end stops, although machines can vary.

3 If prompted, Home the CMM by clicking the **Home**  button.



There are a number of conditions displayed in the **Status** bar on the **Machine** toolbar:

- When no session is open, the **Status** bar shows the status of the connection between PowerInspect and the measuring device:
- 

PowerInspect is not connected to the machine.
- 

PowerInspect is connected to the machine.
- When one or more sessions are open, the **Status** bar shows the status of the connection between the active session and the measuring device:
- 

Neither PowerInspect, nor the active session is connected to the machine.
- 

PowerInspect is connected to the machine, but the active session is not connected.
- 

PowerInspect and the active session are connected to the machine.
- The **Status** bar can also display the following information messages:
- 

The machine is not homed.
- 

The homing procedure has just completed successfully.

The connection procedure varies according to the measuring device. You should follow the prompts given on your own machine. When connected, the probes can be changed and edited.

2. CAD Management

The manipulation of CAD data within PowerInspect is very important. Using the **CAD File Manager,** you can add or remove CAD files, transform CAD data in relation to datums and indicate levels to be included in the inspection.

Coordinate Systems

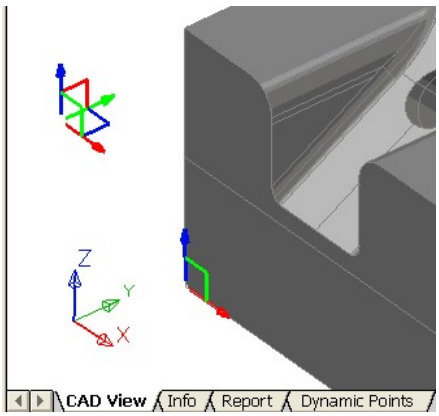
When you create an inspection session, PowerInspect automatically creates two coordinate systems:

The **Machine Datum** is the X 0, Y 0, and Z 0 of the 3D coordinate measurement machine.

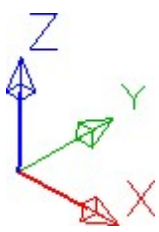
The **PCS Datum** (**Part Coordinate System Datum**), is the X 0, Y 0, and Z 0 of the CAD model. If the **PCS Datum** is in the wrong location, you can transform it using translations and rotations.

When the part is aligned with the CAD model, PowerInspect automatically converts the CMM movements from the **Machine Datum** coordinate system to the **PCS Datum** coordinate system.

By default, these datums are not displayed in the CAD view. To display them, click the light bulb icon next to their entries in the Sequence Tree. The icon changes to to indicate the datums are displayed. In this example, the **PCS Datum** is shown at the corner of the part; the **Machine Datum** is shown above and left of the **PCS Datum**.

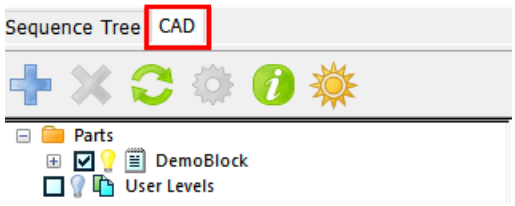


In addition to the datums, a graphical representation of the coordinate system of the active alignment (the **PCS Datum**, by default) is displayed at bottom left of the CAD view. It allows you to identify the axes of the active alignment when the active alignment is not visible on screen.



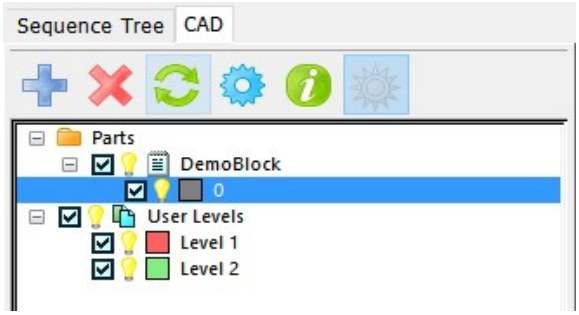
CAD File Manager

To display the **CAD File Manager**, select the **CAD** tab.



When you select the tab, the window displays the **Parts** folder and the **User Levels** area. Click the ⊞ icon to open folder and display the sub-items.

The following example shows the open *Demo Block*. It contains one part, with three levels.



The coloured box indicates the colour with which each level is displayed in the **CAD View**. Change the colour by left-clicking the box, and assigning a new colour from the palette.

The light bulbs indicate whether the level is displayed or not in the **CAD View**, and the check box ☑ includes or excludes the level from the **CAD Context**. These can all be switched on or off by left-clicking on them.

When the **CAD File Manager** is opened, a toolbar is displayed at the top of the tab with a number of functions that can be used to manipulate the CAD data.

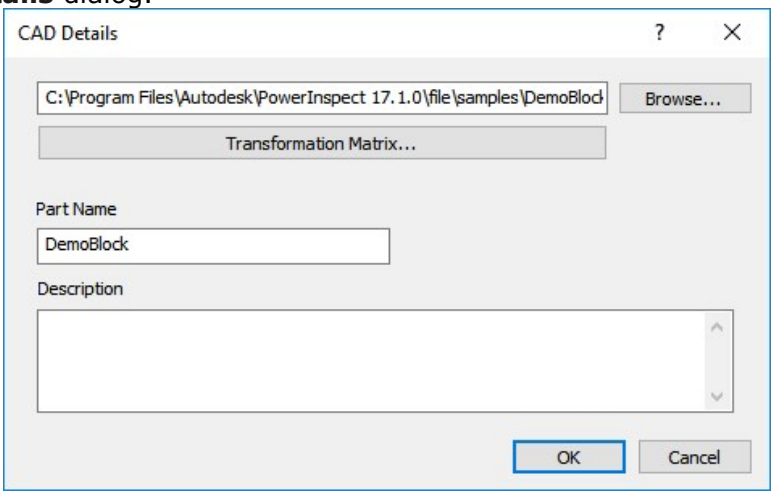
This toolbar enables you to:

- add or remove CAD files.
- reset the user levels.
- edit the CAD details.
- show or hide CAD files and the detailed view.
- highlight CAD details.

Click the **CAD Details**  button to open the **CAD Details** dialog. This dialog enables you to locate CAD files and perform transformations on them.

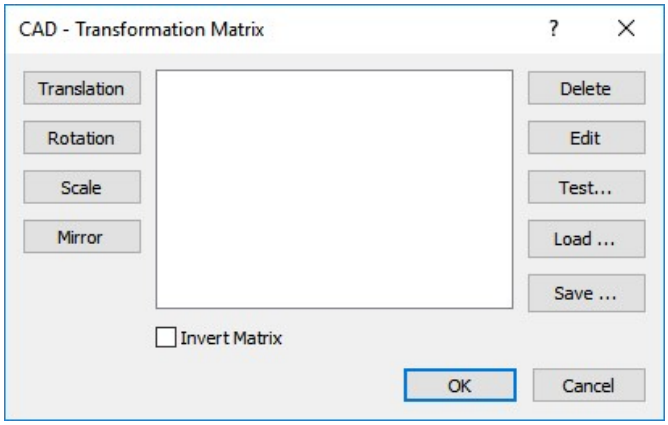
Transformations are used to orientate parts within the **CAD View**.

To transform any CAD data, you must first specify the part by using the **Browse...** button in the **CAD Details** dialog.



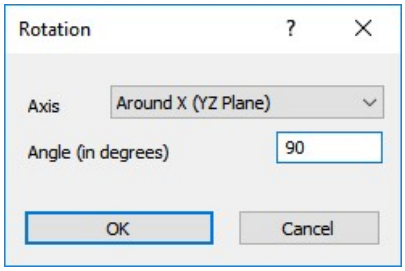
PowerInspect cannot change the CAD model. It can create, hide and edit levels but these cannot be saved to the CAD model and are only visible within PowerInspect.

When the part is specified, click **Transformation Matrix...** to open the **Transformation Matrix** dialog.

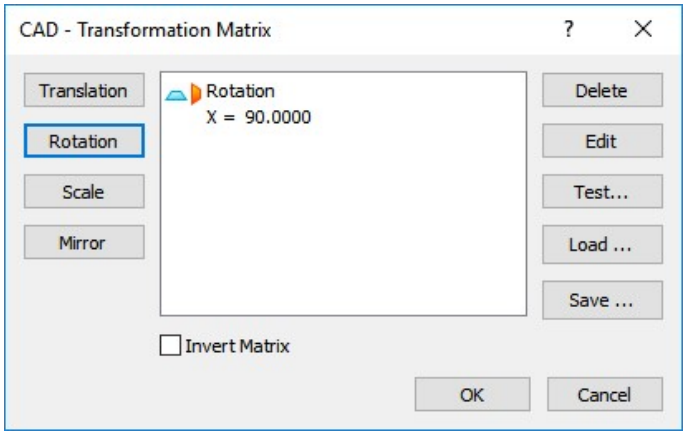


Use this dialog to specify any transformation (**Rotation**, **Translation**, **Mirror** and **Scale**), by clicking on the appropriate button and entering the desired values.

For example, clicking **Rotation** opens the **Rotation** dialog. In this dialog, you can specify the rotational axis (for example, the X axis) and the angle by which the part is rotated (for example, 90 degrees).

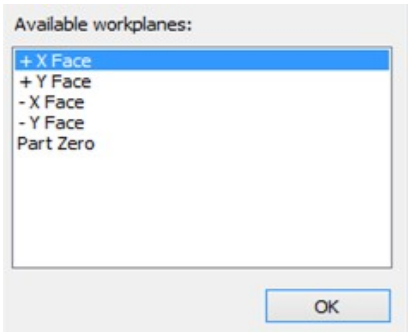


Click **OK** to close the **Rotation** dialog. The **Transformation Matrix** dialog is updated with the new value.

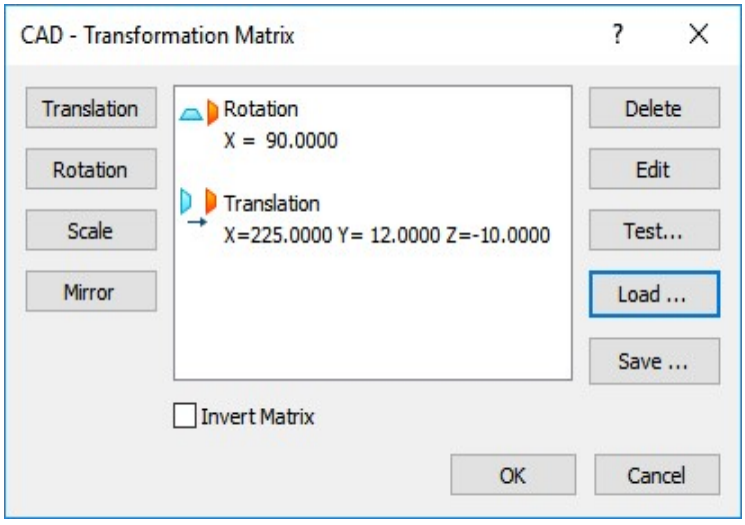


Click **Load** to read and load workplanes from multiple CAD formats.

When a CAD model is selected, the **Select Workplane** dialog lists all the workplanes associated to the model.



When you select a workplane, its transformations are displayed in the **Transformation Matrix** dialog.

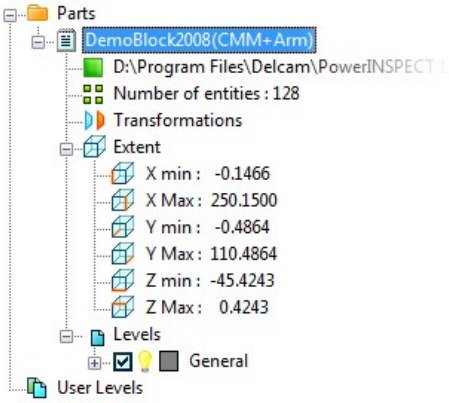


Click **OK** to apply the transformations to the model relative to the datum.

These transformations can be saved, deleted, edited and tested all within this same dialog. To edit or delete a transformation, left-click its entry in the central area of the **Transformation Matrix** dialog.

Summary and Detailed CAD View

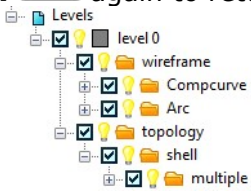
Click the **Summary/Detailed View** button on the **CAD File Manager** toolbar to expand the CAD information area and display more information about the part files.



The information displayed includes any transformations, the levels, part locations and the number of entities that make up the part.

The extent values indicate the dimensions of the part in relation to the datum origin.

Click **Summary/Detailed View** again to return to the data summary.

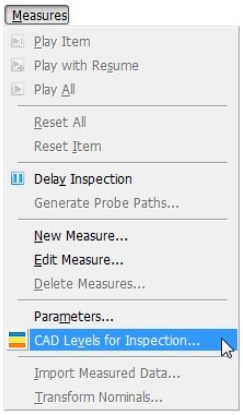


When the **Levels** section is expanded, the sub-levels reveal the individual surfaces.

As before, the light bulbs **hide** or **show** the individual level or surface. The check box **includes** or **excludes** the level from the CAD context. These icons can be switched *ON* or *OFF* by left clicking on them.

The levels can also be managed from the **Menu** bar.

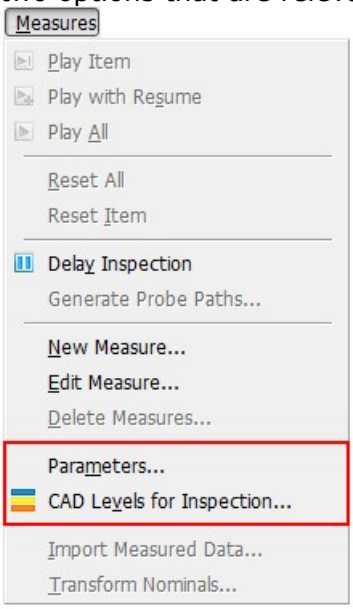
Select **Measures > CAD Levels for Inspection** from the **Menu** bar to display the levels section as shown above.



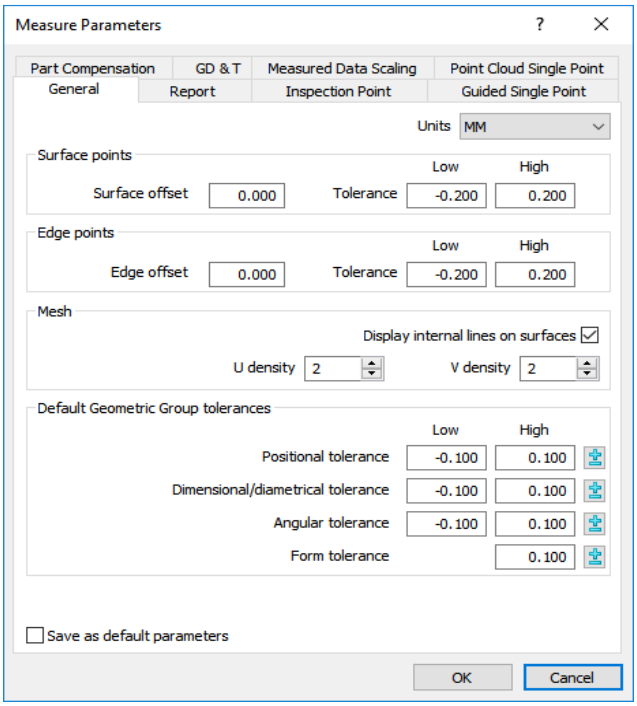
CAD Menu Options

As well as the **CAD File Manager**, CAD edits can be made using some of the options on the **Menu** bar.

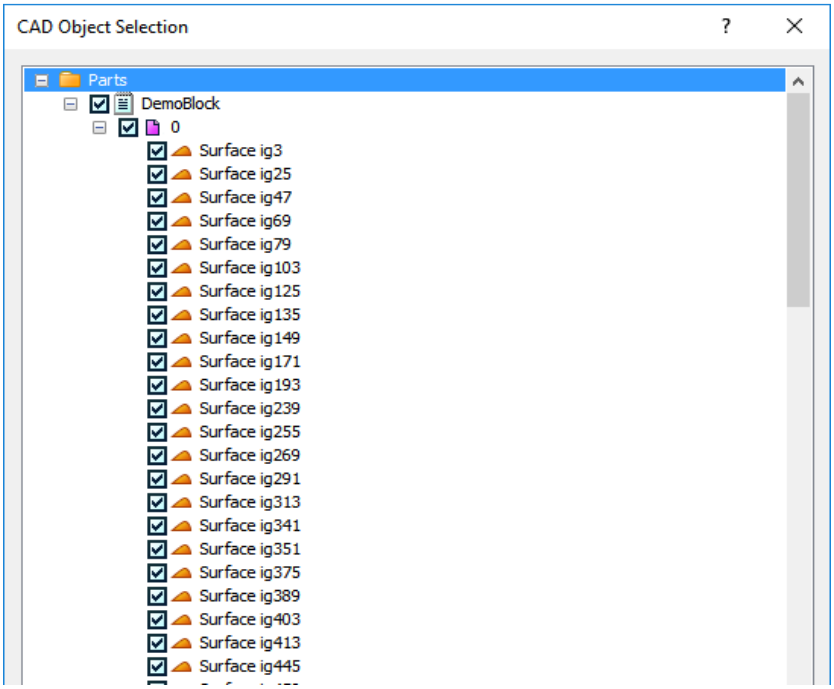
The **Measures** menu contains two options that are relevant to the CAD management.



Select **Parameters** to open the **Measure Parameters** dialog. This dialog contains a series of tabs that relate to different areas of PowerInspect.



Select **CAD Level for Inspection** to open the **CAD Object Selection** dialog. This dialog allows you to select the levels and surfaces you want to make available for inspection.



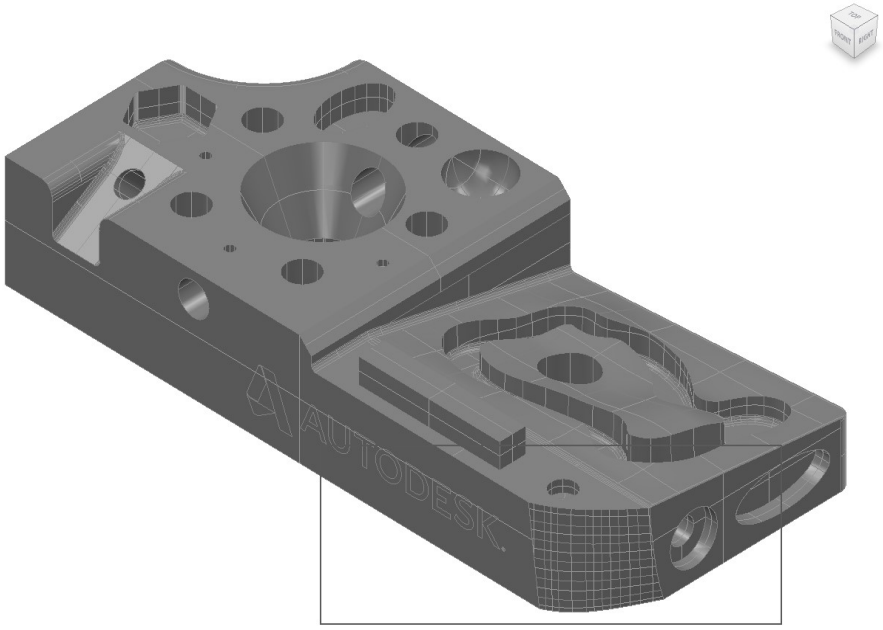
Level Edits

CAD data within PowerInspect can be transferred to other or new levels.

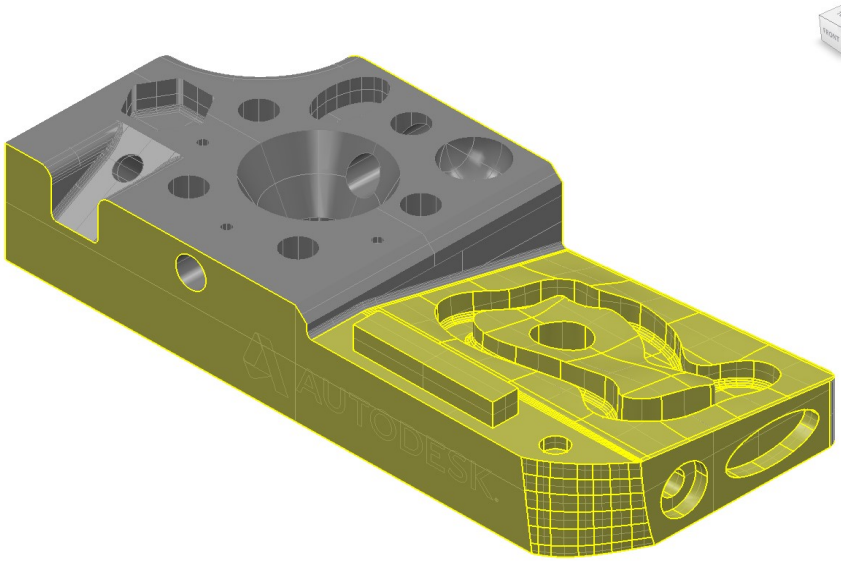
Click the **Surface Selector** button in the **Mouse Context** toolbar to specify which surfaces you want to move, and to which level.

To create a new level:

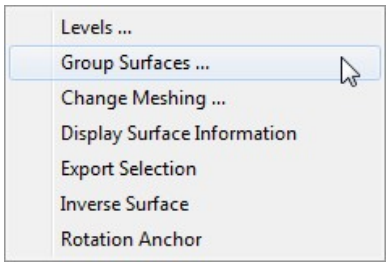
1. Click the **Surface Selector** button on the **Mouse Context** toolbar.
2. Use the left mouse button to box-select the surfaces that you want to add to a level.



When you release the mouse button, the selected surfaces are highlighted. Alternatively, you can select individual surfaces by left-clicking the model.

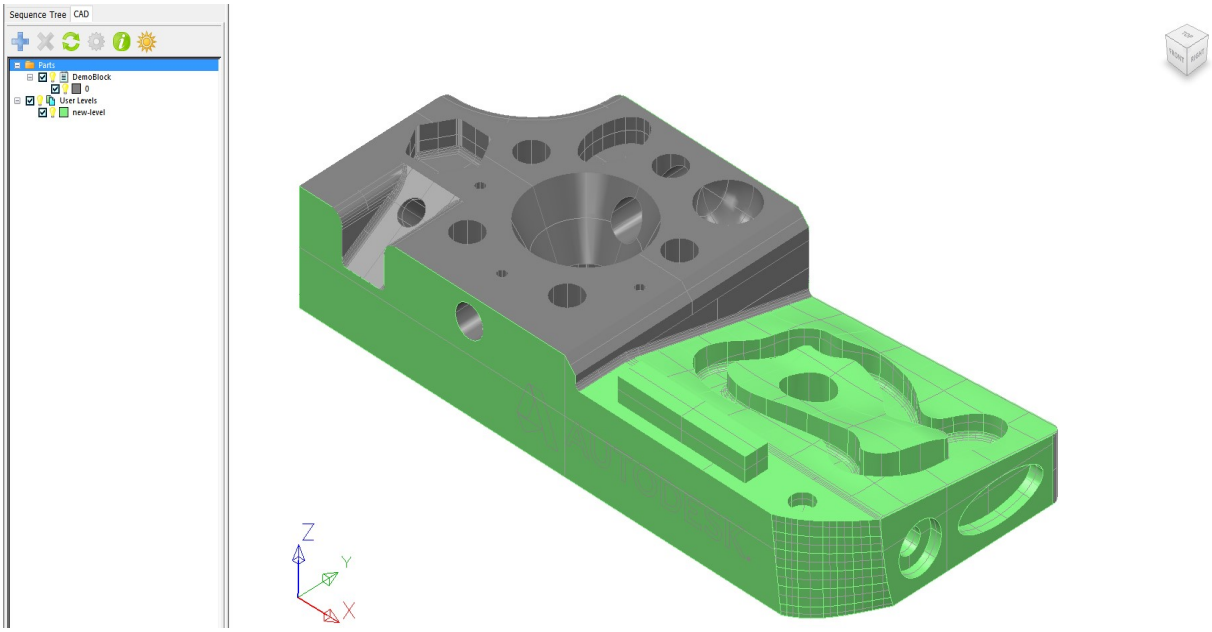


3 Right-click anywhere on the highlighted selection and select **Group Surfaces** from the local menu.

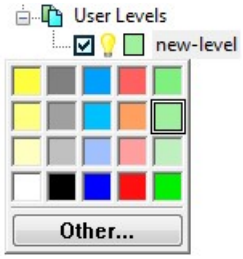


The **Level Manager** dialog is displayed.

4 To add the selected surfaces to an existing level, select the level in the **Available levels** list, and select **OK**.
To add the selected surfaces to a new level, enter a **Name** for the level, and click **OK**. The level appears in the **User levels** area of the **CAD File Manager**.



This technique can be useful if there is a large quantity of data in a view. Each layer has an associated colour, which is used when colour shading the model. To change the shading colour of a level, left-click the small coloured box to the left of the level's name, and select a new colour from the palette.



3. Alignments

Alignment of the part or component allows PowerInspect to match the relative positions and orientations of the CAD and/or Machine datum. An alignment is normally the first item or task in the inspection sequence. The type of alignment strategy selected is dictated by the component's shape and which features, if any, can be used.

Alignments are accessed by selecting the **Alignment**  flyout from the **Item** toolbar.

The most widely used alignment strategies are outlined in this chapter.

Geometric PLP Alignment

A **Geometric PLP Alignment**  is an alignment based upon the relationship of a physical **P**lane, **L**ine and **P**oint (PLP), and **CAD Defined Coordinates**, which define the part in the **X**, **Y** and **Z**. It is considered to be a more accurate method of alignment (compared to the **Free Form Alignment** – see later) because it works directly from CAD Nominal Values, and can be an easier method to understand.

Generating a Geometric PLP Alignment

For this example the DemoBlock2008 CAD file is going to be used.

1. Click **New Document Wizard**  to open the **New Inspection Session** dialog.
2. Select **Measurement with a single CAD Part** and click **Next**.
3. Click **New** and browse to **DemoBlock2008(CMM+Arm).dgk**.
4. Leave the **Offset** and **Tolerance** settings unchanged, and click **Next**.
5. In the **Variables** dialog, browse for any **HTML Report Template** (**Excel** could be used, but for this example **HTML** is to be used).
6. Click **Finish**.

The new session is now ready for inspection.

Choosing the Geometric features

Before the alignment can begin, a decision needs to be made on where the Plane, Line and Point are going to be aligned to, and what will determine these elements.

The Plane

With any alignment, the results are better if a greater area spread is used to define the aligning elements. In this example, the CMM bed (or table) will be used to define the plane, because the model has a flat base, and all the planes in the model are small.

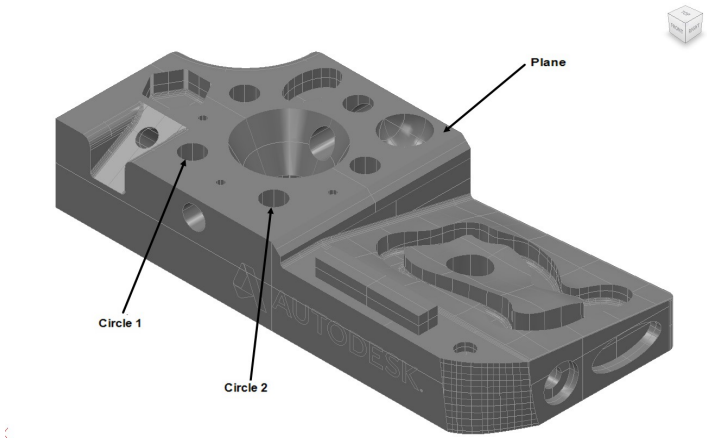
The Line

Lines can be defined from square edges (probed lines), or from the connection or intersection of measured features. In this example, the line is to be defined in the X-direction, using the centres of two measured circles on the part.

The line direction in a Geometric PLP Alignment must be axial.

The Point

Because the circles will have already been defined, the centre position of one of these circles will be used for the point position.



The two circles chosen are indicated, and the plane will be the base area. The line will be between the centres of both circles, and the point will be taken from the centre of circle 1.

Both circles and line will be projected onto the base (plane).

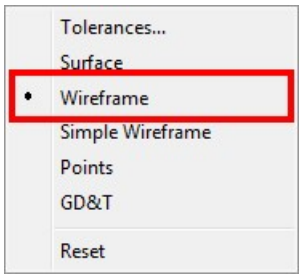
Defining the Geometric Items

When you have chosen the features to use for the alignment, you must now define the geometric items to probe.

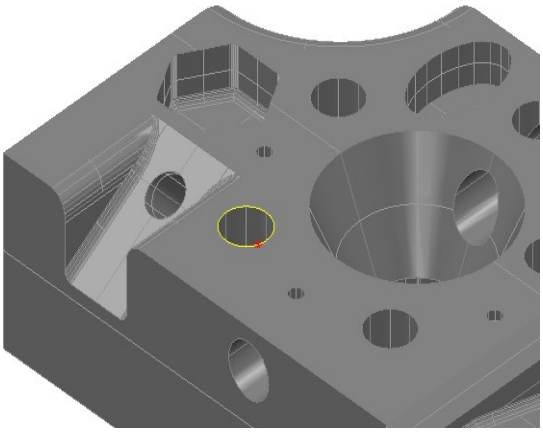
1 Click the **Delay Measure** button on the **Main** toolbar. Alternatively, select **Measures > Delay Inspection**.

Delay Measure allows you to defer the measurement until all the items are specified. Otherwise, PowerInspect will automatically move to the probing process.

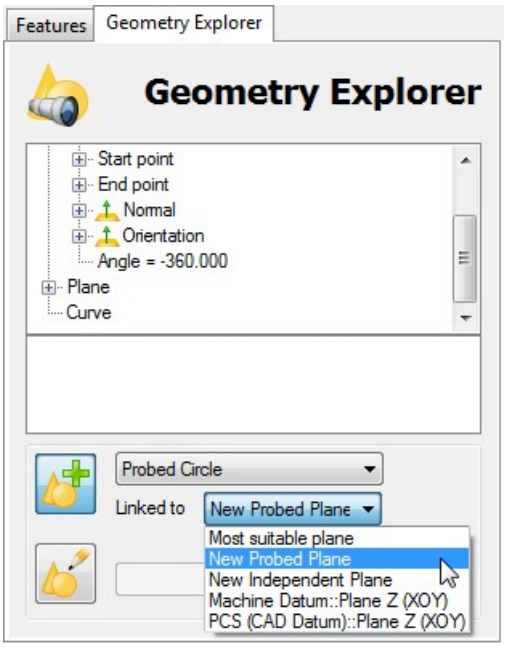
2 Select **Wireframe Checker**, right-click an empty area of the graphics window and select **Wireframe** from the popup menu.



3 Move the cursor over the first circle, as shown below. The circumference is displayed in yellow to indicate it can be selected. Click to select it.

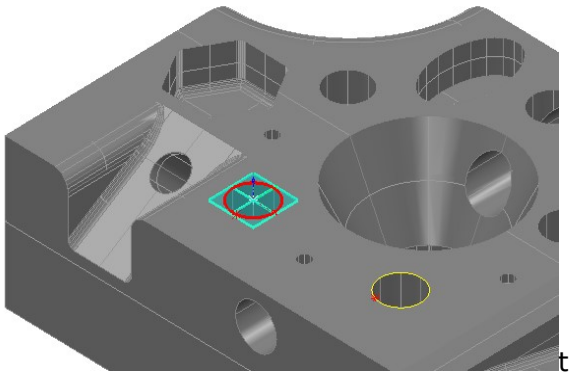


4 In the **Geometry Explorer** tab, select **New Probed Plane** from the drop-down list and click  .



PowerInspect creates a geometric group and adds the new probed plane and the probed circle items to the inspection sequence.

5 Use the cursor to select the second circle, as shown below.



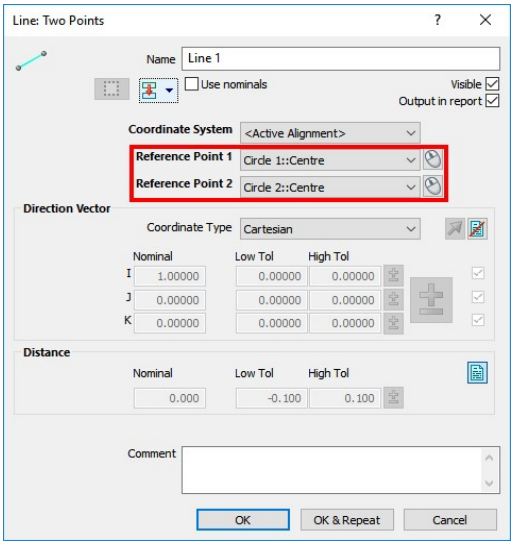
t

6 In the Geometry explorer, select **Plane 1** from the **Linked to** drop-down list and click  to add a second circle to the Inspection sequence.

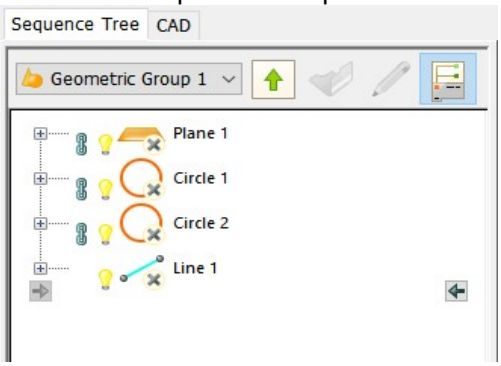
7 Click the **Lines**  button to display the **Lines** flyout menu.

8 Click  to display the **Line: Two Points** dialog. Ensure that the reference points are set as follows:

- **Reference Point 1 — Circle 1::Centre**
- **Reference Point 2 — Circle 2::Centre**



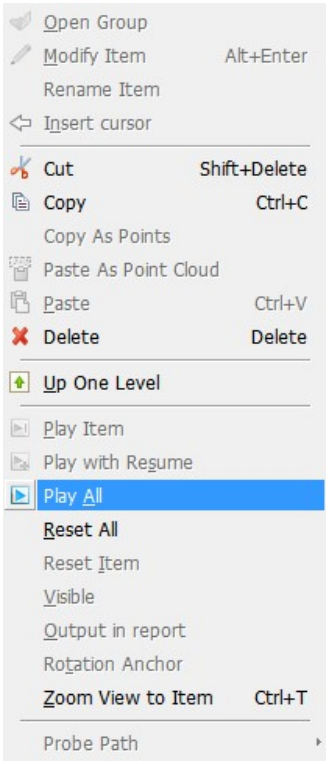
9 Click **OK** to add the line item to the inspection sequence.



Probing the Geometric Items

When the geometric items have been created, and the part is secured to the CMM bed, the probing process can begin.

1 Click **Play All**  on the **Main** toolbar.
Alternatively, right-click any item in the Inspection sequence to display the popup menu, and select the **Play All** option.



 *Selecting the **Play Item***  *option plays only the selected item. Selecting **Play All** plays all unmeasured items.*

2 PowerInspect changes to **Full Screen Measure Mode** and displays the **Feature Probing** dialog, which prompts you to start probing the indicated item.

Full Screen Options Toolbar

In **Full Screen Measure Mode**, a new toolbar is displayed at the right of the screen. This is called the **Full Screen Options** toolbar and contains the functions that are common to all measurement modes.

 **Toggle CAD Context Display** selects the CAD levels that can be probed.

 **Edit CAD Levels Used For Inspection** allows you to choose the levels used.

 **Toggle Previous Measure Display** switches on/off all previously measured items in the view.

 **Change Probe** opens the **Probe Database Editor**, allowing you to change probes and positions.

 **Tracking Box** opens the **Tracking Box**, which displays the probe position in terms of CMM or CAD coordinates.

 **Help** opens the reference help.

Feature Probing Dialogs

When in full screen measure mode, the **Feature Probing** dialog is displayed, which contains options that you can use any stage during the probing process. The options available depend on the items being probed. For geometric items, the following options are available:

 **Reset All** resets all the points taken for the currently active item.

 **Remove Last** allows you to delete the details of the last probed point so that it can be probed again. This is useful when a mistake has been made and the point needs to be re-probed.

 **OK** allows you to save your points when you have finished probing and are satisfied the points are accurate.

 **Cancel** exits the probing mode without saving any of the probed points for the currently active item (items that have been successfully probed and accepted remain measured).

 **Tracking** displays the point about to be probed in a view perpendicular to the point's normal.

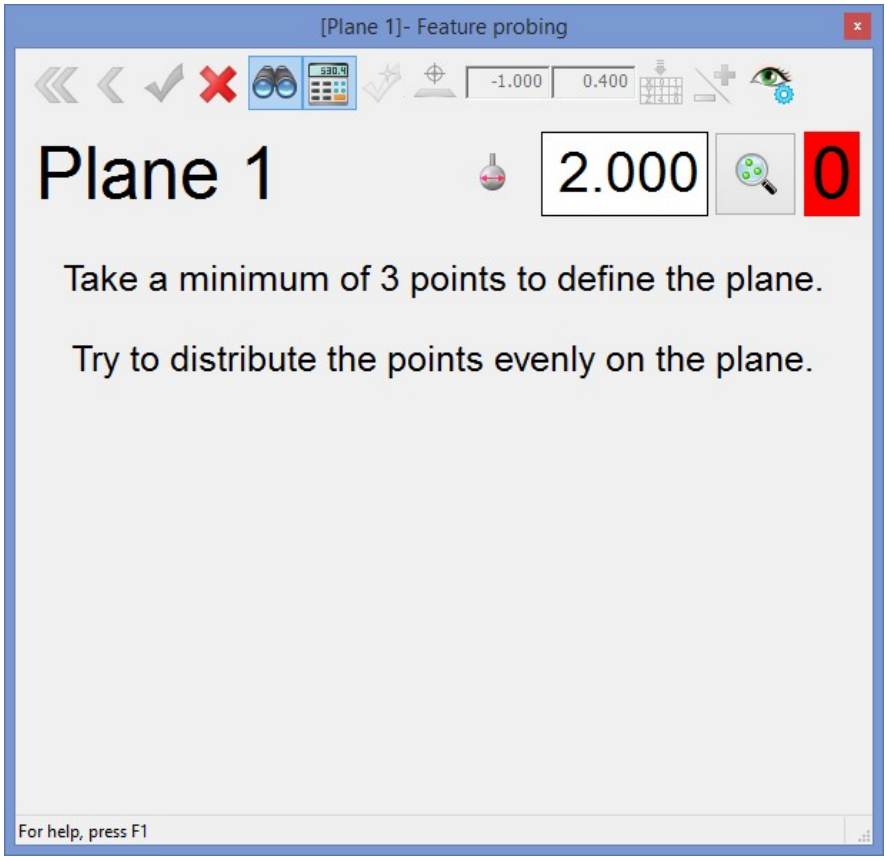
 When selected, **Auto -Calculate** calculates the feature while probing. When deselected, the feature is calculated when accepted.

 When selected, **Auto-Accept** automatically saves items as soon as you have probed the required number of points. When deselected, you must save your measurements using the **OK** button.

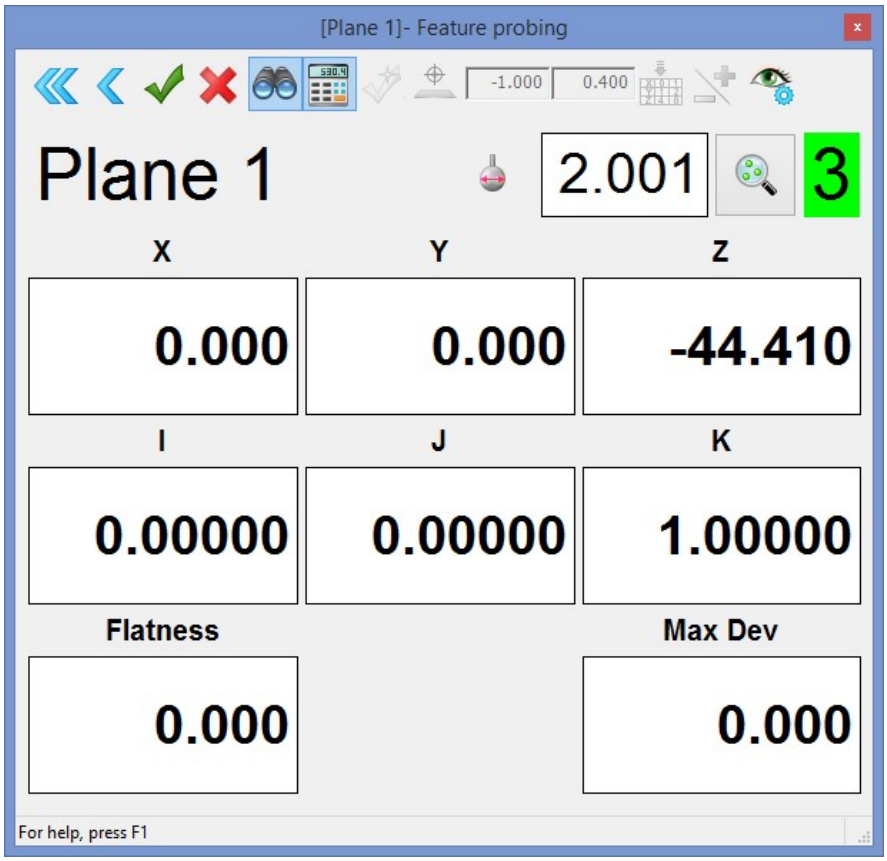
 When selected, **Automatically extract nominals from CAD** loads the nominal data and enables comparison with the CAD during probing. You must have previously measured the active alignment to use this option.

Feature Probing

After you have clicked **Play All**, the first **Feature Probing** dialog is displayed and asks you to take a minimum number of points to define that feature. In this instance PowerInspect is asking you to probe three points to measure the plane.



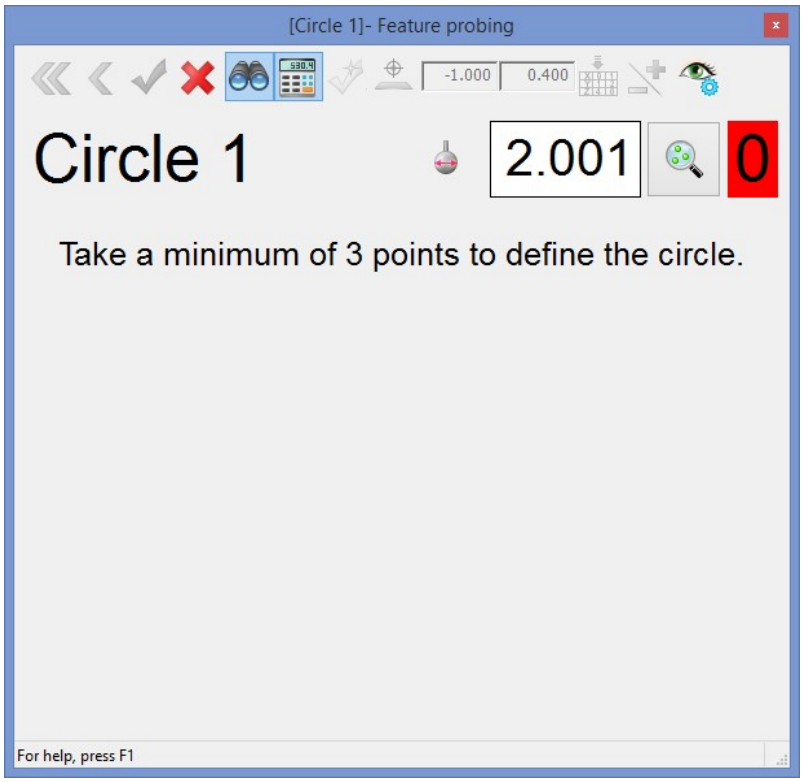
1 Probe three points around the part, using the CMM bed as the planar surface.
As you probe each point, the counter at the top-right of the dialog displays the number of points taken. When you have probed the minimum number of points, the background colour changes from red to green.



2 When you have probed at least three points, click  to save them.

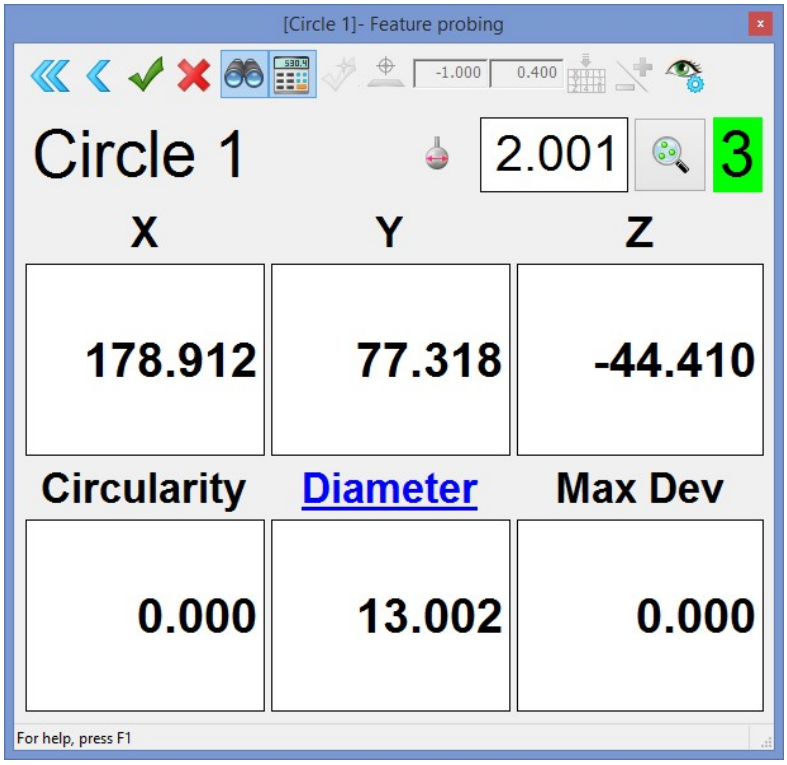
 *If **Auto-Accept**  is enabled, PowerInspect automatically saves your measurements when you have probed the minimum number of points.*

PowerInspect plays the next item in the inspection sequence. The next **Feature Probing** dialog is displayed asking you to take a minimum number of points to measure Circle 1.



However, rather than the minimum three points, four points will be taken for the circles, at the North, East, South and West positions. This makes it easier for you to probe an even spread of points.

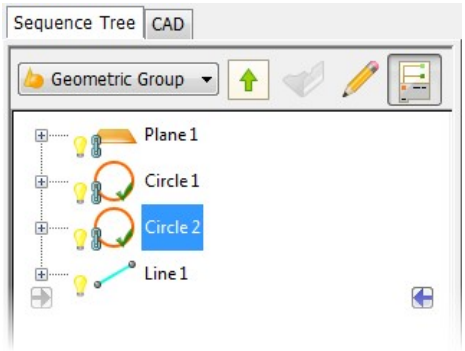
3 Probe four points inside **Circle 1**, which is highlighted on CAD model on screen.



4 When the points have been probed select  to save them.

5 Repeat the process for **Circle 2**.

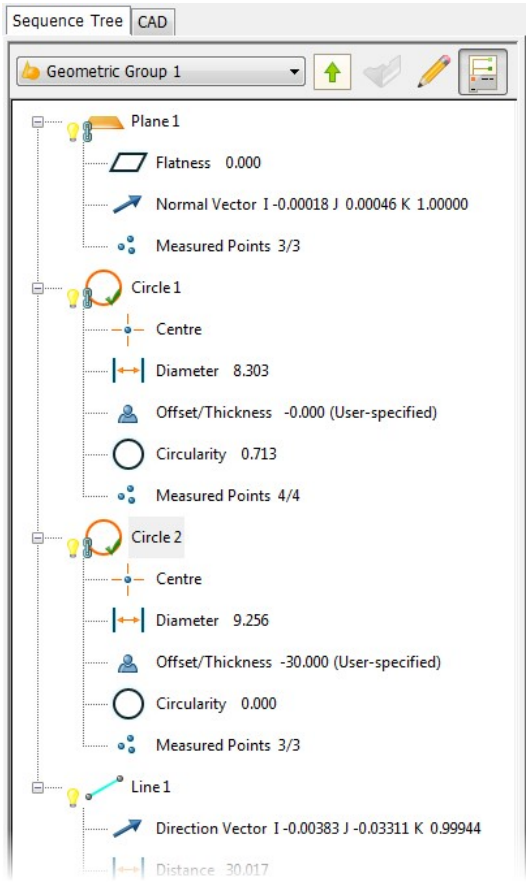
When all the items have been probed, the Inspection sequence is updated, and all the items lose their crosses.



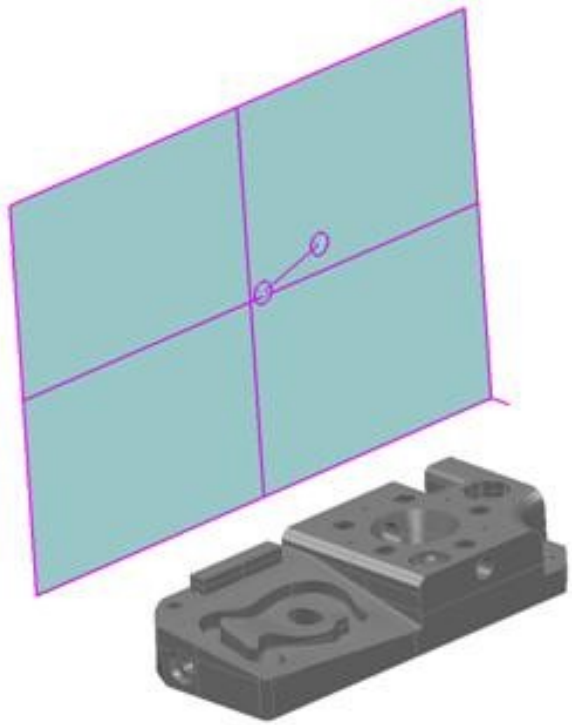
The line was not probed but its red cross icon has gone because the centres of the two circles determine the line's position. These have been probed, and therefore satisfy the measurement conditions.

 *The point required for the alignment will be specified as the centre of **Circle 1**.*

Click on the **Expand** ⊞ icon next to each item to show basic information about it.



The unaligned items can be seen in the CAD View with purple wireframe and blue shading. For example:



If the probed items cannot be seen, press the ***Resize to Fit***  *button to resize the screen to fit the model, because the items may lie some distance away from the model at this point.*

You must now create an alignment to align the probed positions with the CAD model.

Generating the Geometric PLP Alignment

To create the Geometric PLP Alignment, some CAD nominals need to be known. These nominals create the relationship between the probed positions and the CAD model, and orientate them in terms of the **X**, **Y** and **Z** coordinates.

The deciding factors will be the plane height and normal orientation, the line direction, and the point position.

Because Wireframe Checker was used to select the geometry, the nominal data for the features is already known.

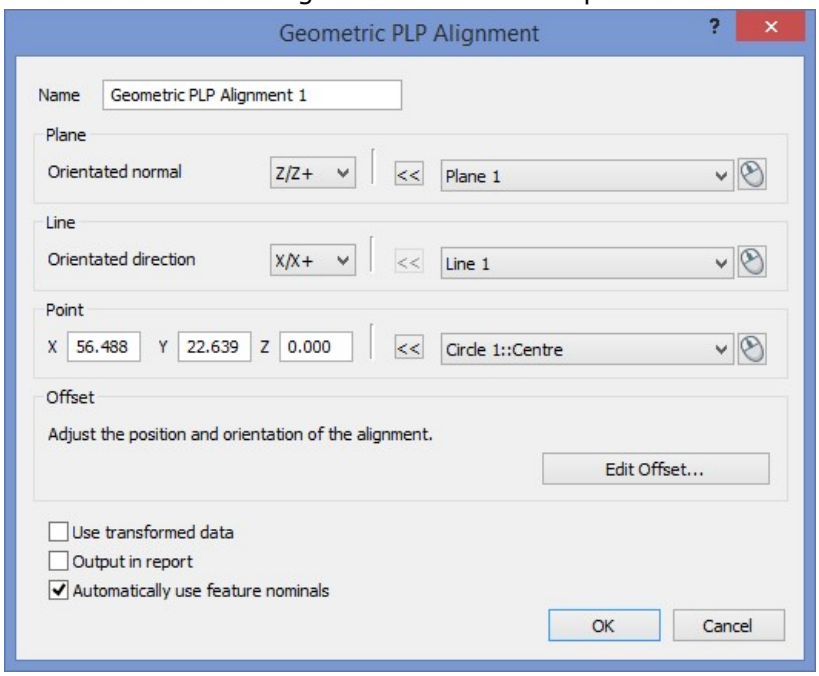
 *If these values are unknown, then the* ***Geometry Explorer***  *may need to be used to select the Feature locations.*

A Geometric PLP Alignment can now be created using these values:

1 Select  to close **Geometric Group 1**.

2 From the **Alignments**  flyout on the **Item** toolbar, select **Geometric PLP**

3 **Alignment**  .
The **Geometric PLP Alignment** dialog appears and prompts you to specify the **Plane**, **Line** and **Point**. A **Name** for the alignment can also be specified here.



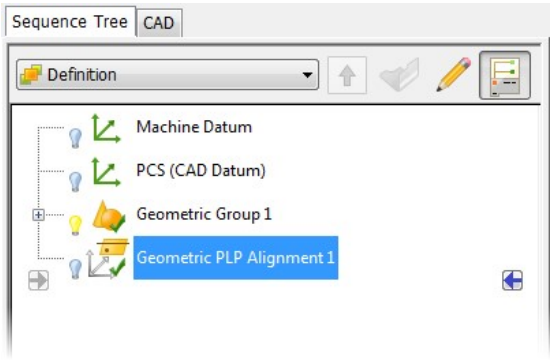
4 In the **Plane** area, select **Plane 1** from the drop-down list, and set the **Orientated Normal** to the **Z/Z+**.

5 In the **Line** area, select **Line 1** from the drop-down list, and set the **Orientated Direction** to the **X/X+**.

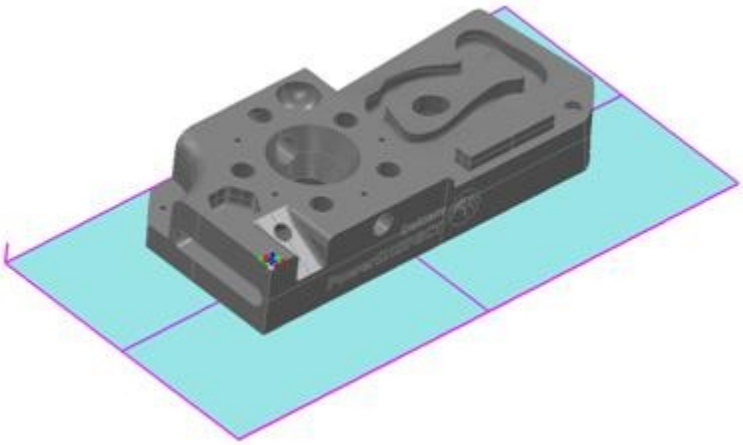
6 In the **Point** area, select **Circle 1: Centre** from the drop-down list. If the **Point Coordinate** nominal values are not loaded, click  to load the data.

7 Click **OK** to close the dialog and create the alignment item.

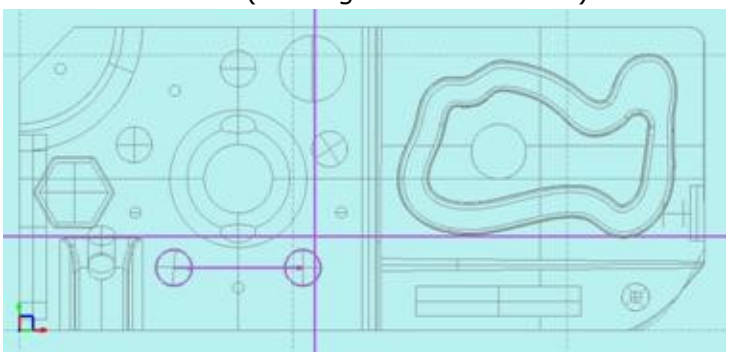
All the criteria for the Geometric PLP Alignment have been satisfied, and the alignment can be seen in the Sequence Tree.



The **CAD View** is updated, with the **Geometric Alignment** positioned correctly.

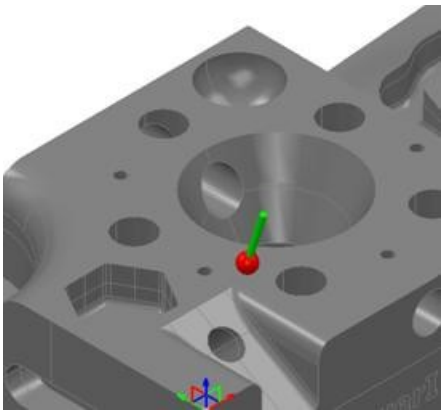


This view of the model from above (looking down the Z axis) shows the alignment.



The alignment can be seen more closely in the zoomed view of the bottom-right corner (below).

To test if the alignment is correct (if probe tracking is available on the system), move the probe around key points on the part and check the Graphics window to see if the representation of the probe is in the same place (see below). Any variation suggests an incorrect alignment, a failure in the probing of the items or an incorrect part in relation to the CAD.



8 Click  to save the inspection session. Name the file **GeometricPLP.pwi** and save it in any convenient location (e.g. **C:\Temp**).

Free Form Alignment

Free Form Alignment  is considered to be relatively inaccurate because it relies on the skill of the user, but it is sometimes the only option if there are no clearly definable features for which CAD values are known (such as flat planes, circular forms etc.).

Generating a Free Form Alignment

For this example, the **DemoBlock2008(CMM+Arm)** file is going to be used. There are a number of geometric features on this part, but for the purposes of this exercise, these will be ignored and it will be assumed that there are no features.

1. Click  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part**, and click **Next**.
2. Browse for **DemoBlock2008(CMM+Arm).dgk** and click **Open**.
3. Leave the offsets and tolerances unchanged, and click **Next**. The **Variables** page is displayed.
4. Browse for an **HTML Report Template**. The report variables are displayed.
5. Click **Finish**.

The new session is now ready for inspection.

Choosing the target positions

When performing a **Free Form Alignment**, you need to carefully study the CAD model and physical part before deciding where to take the probed positions. For a **Free Form Alignment** the part needs to be held in position in all three axes (**X**, **Y** and **Z**). This holding should ensure that the part cannot move in a space through any translation or rotation.

Taking the **CMM Bed/Table** as the surface on which to 'pin' the part, you need to choose a realistic minimum of six or seven points (this can be as low as four but the alignment is less accurate). In this case, six points will be used: **3** in the **Z** direction, **2** in the **Y** direction and **1** in the **X** direction. It is important to place these points in positions that will be easily found on the physical surface, such as in line with other parts of geometry, or key surface points. These will act as guides for the probing positions.

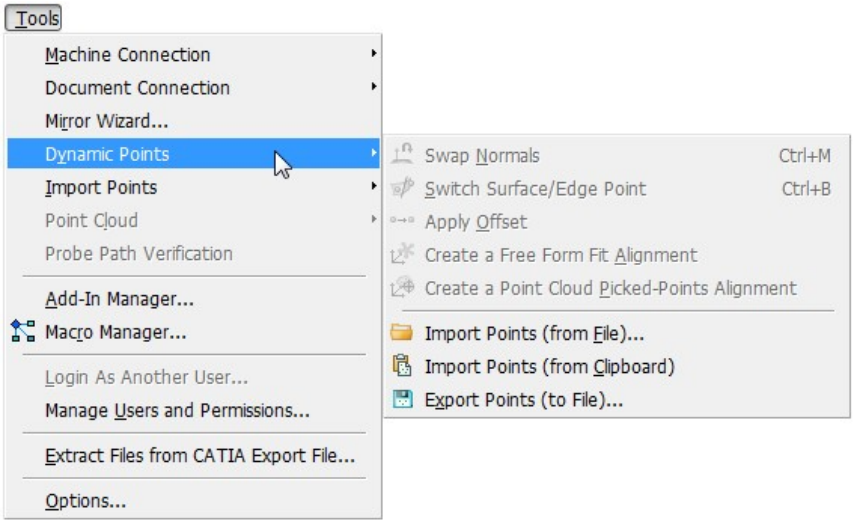
When you have selected the positions, it is time to create a **Free Form Alignment**:

1 From the **Alignments**  toolbar in the **Item** toolbar, select the **Free Form Alignment**  button.

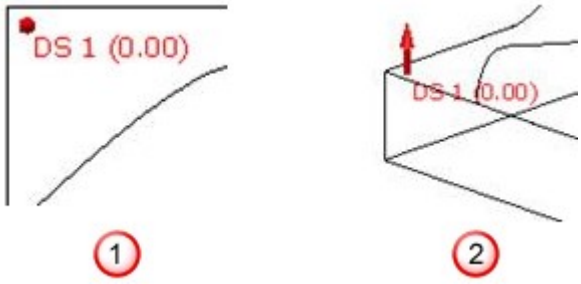
The **Dynamic Point** toolbar is displayed on the right of the screen.

This toolbar allows you to create the free-form Target Points, and to manipulate their positions, normal directions and offset values.

*The **Dynamic Point** toolbar can also be accessed from the **CAD View** toolbar using the **Dynamic Points Editor**  button. Alternatively, you can access Dynamic Points editing options through the **Tool > Dynamic Points** option from the Main menu.*



In the **CAD view**, the cursor has now changed to a **Target Sight**  . This enables you to select the probing points on the CAD surface. Select points by double clicking on the surface of the model. The representation of the point can vary according to the view. For example:

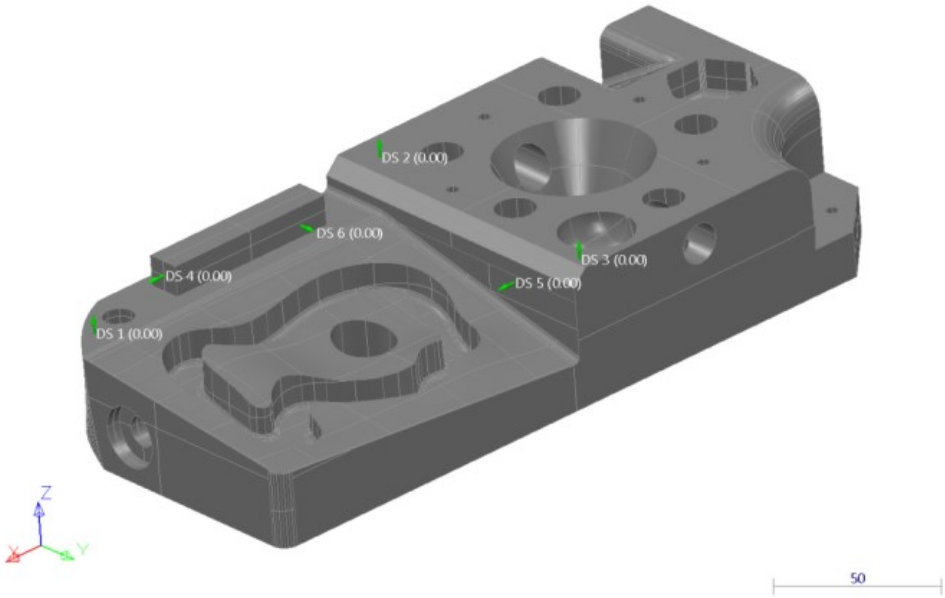


1 View straight down

2 Isometric view

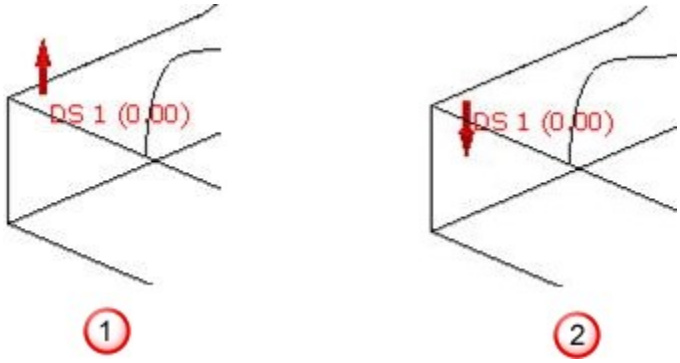
2 Choose six key points (**3** in **Z**, **2** in **Y** and **1** in **X**) on the part surface that can be easily found. Use key features to aim for such as lines, indents, corners and even projected aiming positions on the part.
The points chosen on the part now need to be re-produced on the **CAD** file within PowerInspect.

3 Replicate the points chosen on the part within the **CAD View**. Do this by double-clicking the **Target** cursor on the chosen surfaces to define the 6 points for alignment.
With the **Target Points** selected, the CAD View should look something like this:



When you have created the points, you can change their position and orientation, or delete and recreate them.
Hold the Target Cursor over any of the chosen points and the cursor changes to a hand  . This allows you to select and move the point around the surfaces using **Dynamic Points Selection**, which can help you find positions that are more easily probed (lining up with feature, for example).
The **Dynamic Points Selection** also lets you choose the points to be manipulated using the other **Dynamic Points Editor** toolbar functions.

Using the **Swap Normals**  button allows you to reverse the direction of the points.

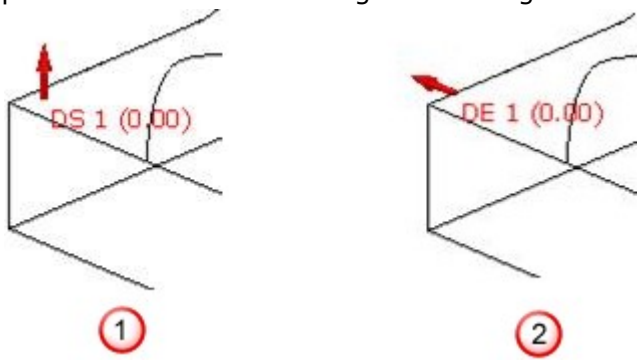


1 Normal To Surface

2 Normal Reversed

This can be useful if the user-defined point has been created in the wrong direction, or if the probed point is in an inaccessible position, for example, the underside of a surface.

Using the **Switch Surface/Edge Point**  button allows you to flip the surface point onto the edge if the point chosen is close enough to the edge.



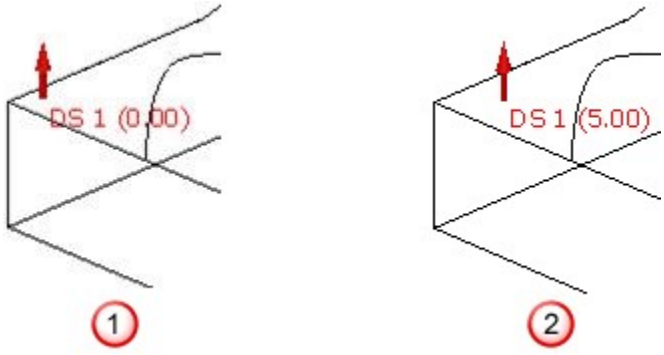
1 Surface Point

2 Edge Point

This is used when edge points are required rather than surface points.

 *The point label changes from an S (Surface) to an E (Edge).*

Using the **Apply Offset**  button allows you to set an offset value for the probed position.



1 Without Offset

1 With Offset (value in brackets)

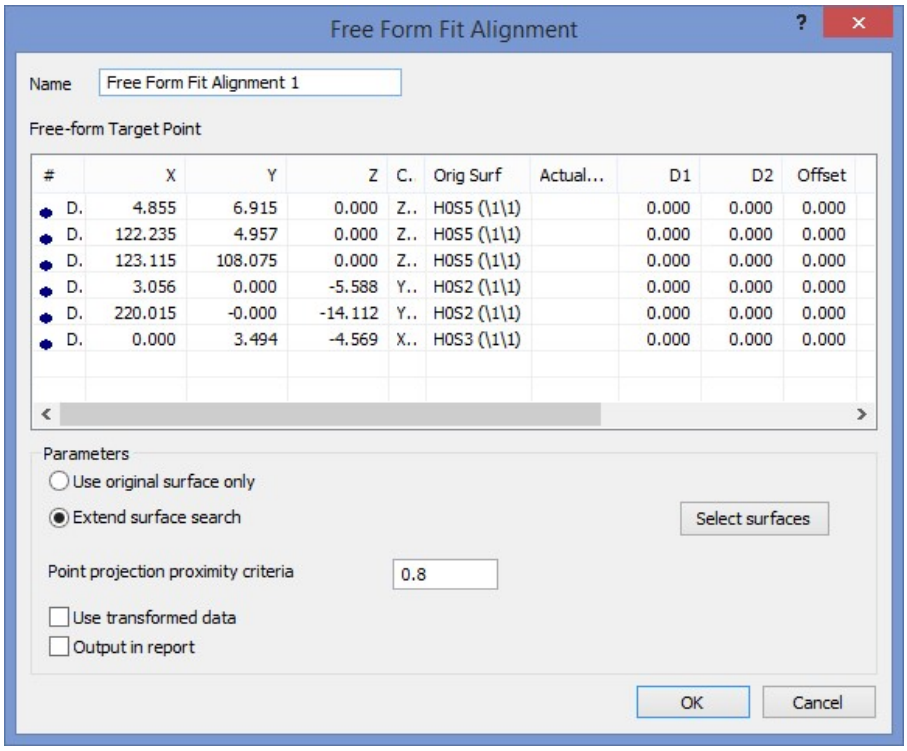
This is useful for inspecting pressings, whereby a known uniform thickness of material allows user to inspect the underside of a pressing, taking into account that thickness.

Once the **Target Points** have been selected and manipulated, the Free Form Alignment can be created ready for probing.

4 Select **Delay Measure** button from the **Main** toolbar.

5 Click the **Create Free Form Fit Alignment** button from the **Dynamic Point** toolbar.

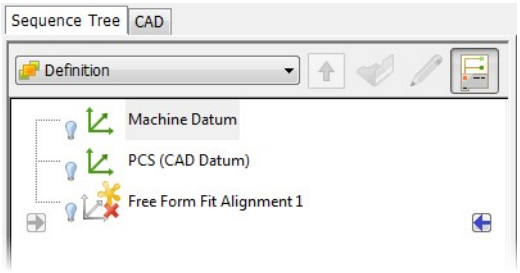
The **Free Form Fit Alignment** dialog appears indicating the **Positional Coordinates** of each chosen point, the **Surfaces** they lie on, and the **Offset** values.



Within this dialog, you can specify the **Point Projection Proximity Criteria**, which define the area of search around the probe.

6 Increase the **Point projection proximity criteria** to **3 mm** and click **OK**.

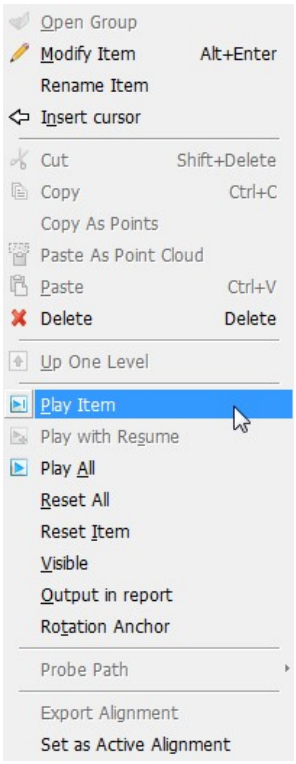
A Free Form Fit Alignment item is created in the inspection sequence.



This is the **Free Form Alignment** created from the **Target Points**, and will be used for probing the part.

*The cross icon next to **Free Form Fit Alignment 1** signifies that the alignment has not yet been probed or played.*

7 Right-click **Free Form Fit Alignment 1** to display the context menu.

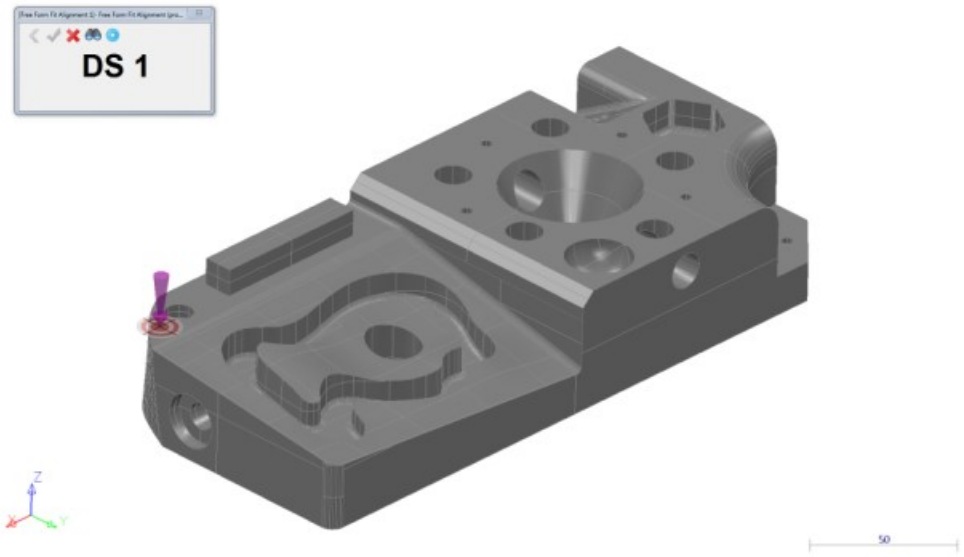


8 Select **Play Item**.

Alternatively, click the **Play Item** button on the **Main** toolbar.
The **Probing Screen** is displayed allowing you to probe the pre-defined target points.

Probing the Part

When you click **Play Item**, you enter **Full Screen Measure Mode** and you can probe the part. In this mode, a **Probing** dialog is displayed, with the target number shown (e.g. **DS 1**) and a series of probing option buttons. These buttons are the same as those displayed when probing the **Geometric PLP Alignment**, except that the **Reset All** has gone, and the **Parameters** button (which shows the **CAD Context** and **Proximity Criteria** for the probed points) is shown.

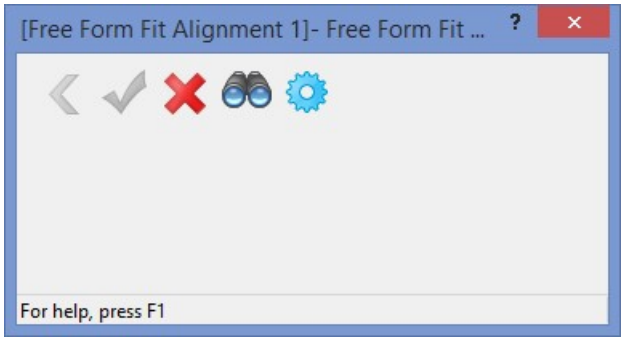


The **Dynamic Point Target**  is displayed, and this is where the skill of the user comes in, because you must probe as close to the targets as possible. The targets appear in sequence order and as each one is probed the next one is displayed on the screen.

1 Probe the 6 points to complete the alignment.

Always check the screen at each point to make sure the correct position is being probed. To achieve a more accurate result, aim for within +/- 3mm of the target point.

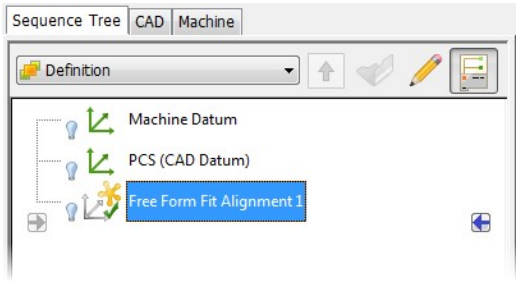
The probing dialog indicates that there are no more points to take when you have probed all the defined target points.



2 Select  to accept the alignment. PowerInspect calculates the alignment.

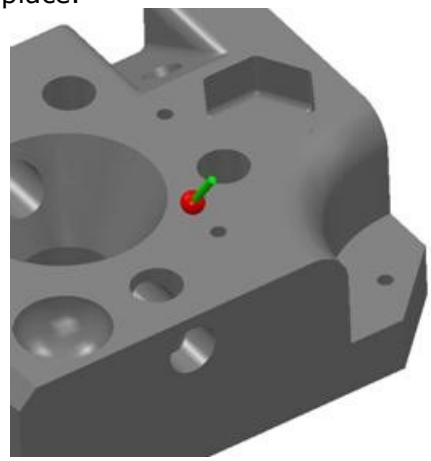
When the calculation is complete, PowerInspect exits the **Full Screen Measure Mode**. The less time this takes, the better the alignment tends to be.

In the Inspection sequence, the cross ✖ on the Free Form Fit Alignment item has changed to ✓ too indicate it is now measured.



The **Demoblock2008(CMM+Arm)** part has now been aligned using the **Free Form Alignment** method.

To check this alignment (if probe tracking is available on the system), move the probe around key points on the part and check the **Graphics Window** to see if the representation of the probe is in the same place.



If not, it may be necessary to either re-probe the part, or check the probe configuration for error mapping.

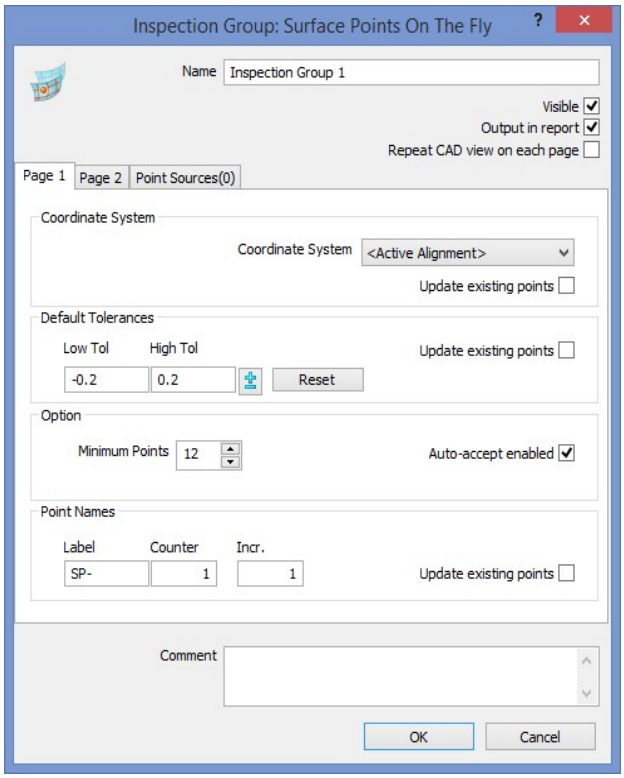
Generating a Surface Inspection Group

When the part is correctly aligned to the CAD file, the next stage is to create a surface inspection group. This group will be used to check the accuracy of the **Free Form Alignment**, and to improve the alignment using the **Best Fit Analyser**, so it is not necessary to include this initial group in the **Report**.

Surface inspection will be covered in greater detail in a later chapter.

1 Make sure the **Delay Measure** button is selected.

2 Create a **Surface Inspection Group** using **On the Fly Surface Points** from the **Inspection Groups** flyout found on the **Item** toolbar.

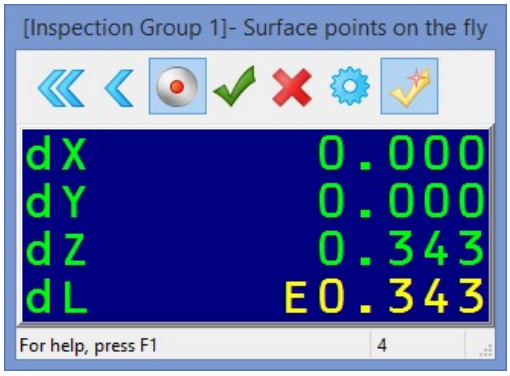


3 In the **Surface Inspection Group** dialog, deselect the **Output in Report** check box and leave the other settings unchanged.

4 Select the **Auto-accept enabled** check box and then set the **Points** to **12**. Leave the other settings unchanged and click **OK**.

5 Right-click the **Surface Inspection Group** icon in the inspection sequence and select **Play Item**.

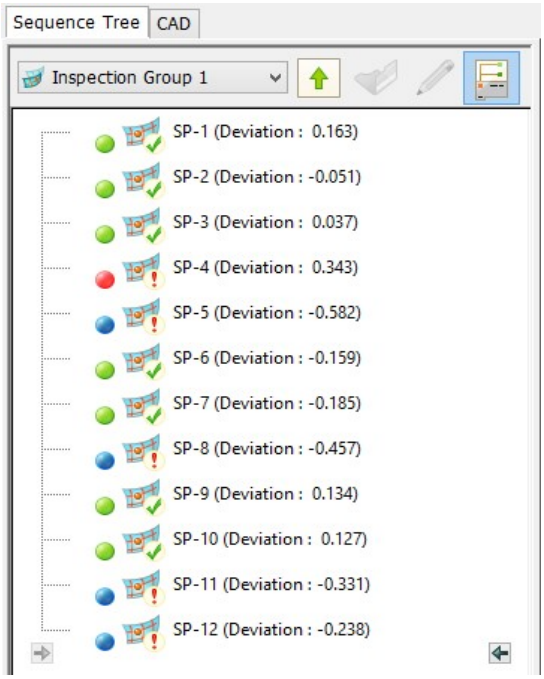
6 Probe **12** surface points on the part, **4** in **Z**, **4** in **X** and **4** in **Y** keeping a wide spread. As you probe the points, the **Surface Points on the Fly** dialog displays the deviation of the measured points.



If no points are being registered, it may be necessary to increase the proximity criteria to help detect them.

7 When you have probed 12 points the exclamation mark icon in the inspection sequence, is no longer displayed on the **Surface Inspection Group** icon.

8 Double-click on the **Surface Inspection Group** to open the group and verify that **12** points have been probed.



If the points have a wide deviation, it may be necessary to re-probe the alignment. The **Best Fit Analyser** may be able to correct this, but the key is in recognising the whether or not a **Best Fit** will work.
For example, if positive deviation can be seen on one side of the part, and negative deviation on the other side (each of the same value), this suggests a slight translation. Similarly, opposites around an axis indicate a rotation. If this sort of pattern can be seen, a **Best Fit** will improve the results.

Generating a Line of Best Fit

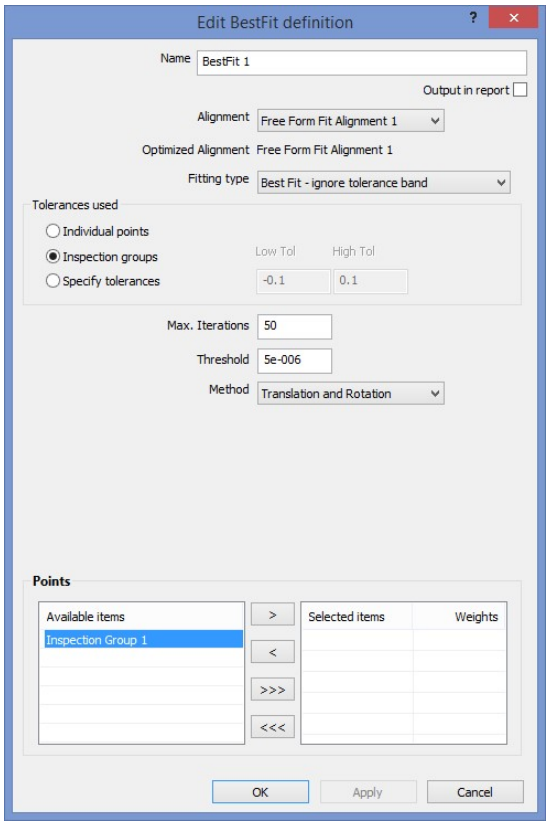
Because of the dependence on the skill of the user, a **Free Form Alignment** may need some fine-tuning to generate a better alignment. PowerInspect contains a Best Fit optimisation option, which allows you to generate a line of best fit between the probed points, and to align the CAD model and the part more accurately.

*The **Best Fit Optimisation** option should only be used on the Free Form alignment. Attempts to use it on an alignment based on geometric data may cause a distortion of the results. In addition, Best Fit optimisation should be used only once on each Free Form inspection.*

For this example a Best Fit optimisation is applied to the newly created surface inspection group:

1 Select **Up One Level** in the Sequence Tree to close the Surface Inspection Group.

2 From the **Alignments** toolbar, select **Best Fit**. The **Edit BestFit definition** dialog is displayed.

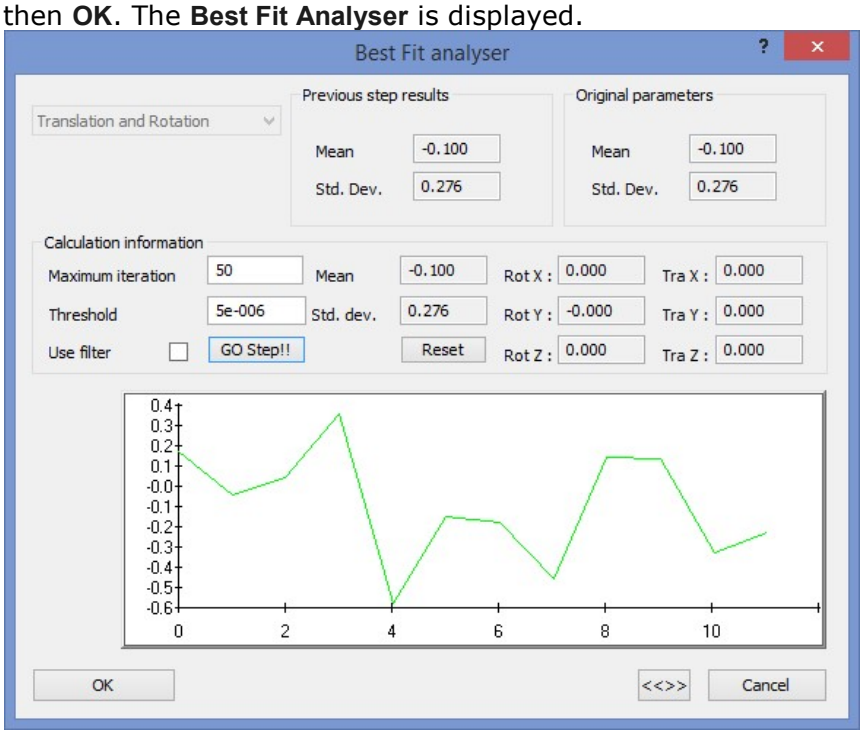


This dialog allows you to **Name** the item, to specify the fitting **Method**, and to select items to be used to calculate the best fit.

3 In the **Method** drop-down list, select **Translation and Rotation**.

4 Click to highlight **Inspection Group 1** from the **Available Items** list, and select it using the > button.

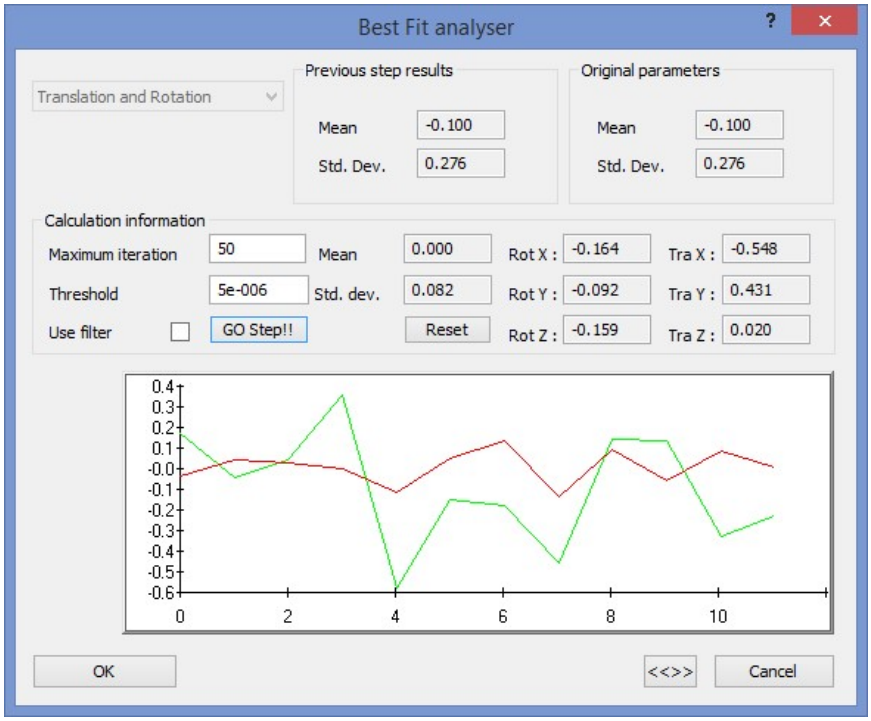
5 Click **Apply**, then **OK**. The **Best Fit Analyser** is displayed.



*If **Delay Measure** is selected, right-click on the **Best Fit Icon** in the Inspection sequence and choose **Play Item** to start the **Best Fit Analyser.***

A graphical display of the current points' deviation is given (green line) along with the information used for the calculation of the Best Fit.

6 Leave the settings unchanged and select the **Go Step!** button.
The **Best Fit Optimisation** is applied to the data and the new deviation graph (red line) fits closer to zero.



7 Select **OK** to accept the optimisation.
The **Best Fit** has brought the results closer to zero, creating a more accurate alignment.

Some results may be anomalous and may need modification or further inspection.

Clicking the  button will switch the view between original data with best fit and just the best fit. The scale changes accordingly.

8 Re-open the Surface Inspection Group to verify that the deviation results have improved.

With the alignment created, and best fit applied, more surface inspection groups can now be taken. Since all items are created with respect to the Best Fit, further best fitting is not required. However, should the results of the alignment not be accurate, it may be necessary to re-align the part.

RPS Alignment

The **R**eference **P**oint **S**ystem (**RPS**) alignment is a technique developed by Volkswagen and has now become an industry standard for inspection. As an alignment technique, its method lies somewhere between a **Free Form** alignment and a **Geometric** alignment, combining the benefits of both, with the ability to accept geometric positional data and surface points.

For the purposes of this training course, the use of geometric data will be emphasised, utilising the **Geometry Explorer** to find positional points in the CAD model.

Generating an RPS Alignment

Using the methods previously described in the course for starting a new session:

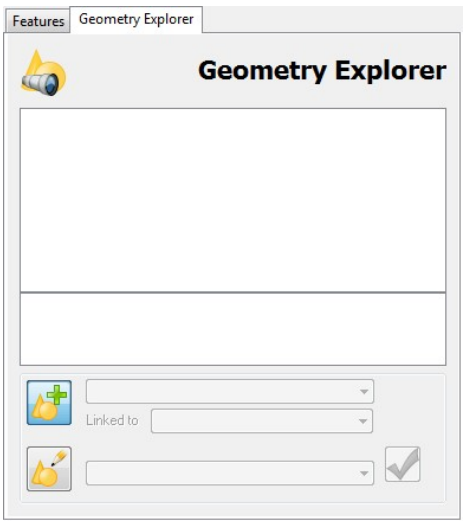
1. Click  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD Part** and click **Next**.
2. Browse for **Demoblock2008(CMM+Arm).dgk**.
3. Leave the **Offset** and **Tolerance** settings unchanged, and select **Next**.
4. In the **Variables** dialog, browse for any **HTML Report Template** (**Excel** could be used, but for this example **HTML** is to be used).
5. Click **Finish**.

Choosing the Geometric Elements

As with the previous methods, you must decide which geometric features will determine the alignment. In this example, the part will be aligned using a series of probed circles, each with their own locally probed plane. The centres of these circles will then be used to determine the **RPS** alignment. The Geometry Explorer will be used to create these circles.

The Geometry Explorer

The blank **Geometry Explorer** is shown below. As each feature is selected in the CAD view, its details appear in the tab.

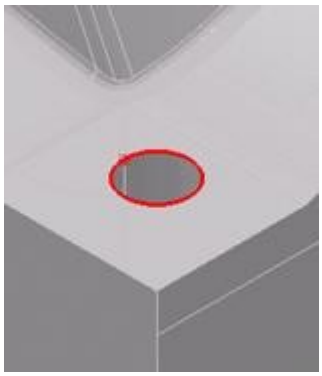


The **Geometry Explorer** can be used in two ways: as a measuring device, providing CAD nominals for geometric alignments; and as a selection tool, selecting the geometric items you want to probe.

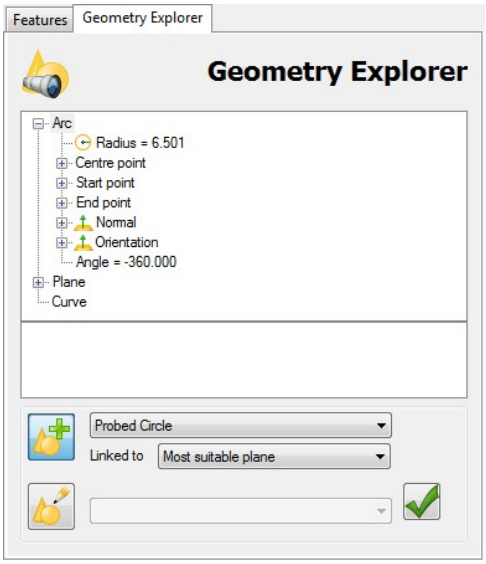
*The **Geometry Explorer** and **Features** tabs can be hidden by toggling*  *at the top of the Sequence Tree.*

To select an item, select the **Wireframe Checker**  button in the **Mouse Context** toolbar, then hold the cursor over the feature in the CAD model until its wireframe is highlighted yellow, then left-click to select it.

The chosen item is highlighted in red, with its details displayed in the **Geometry Explorer**.



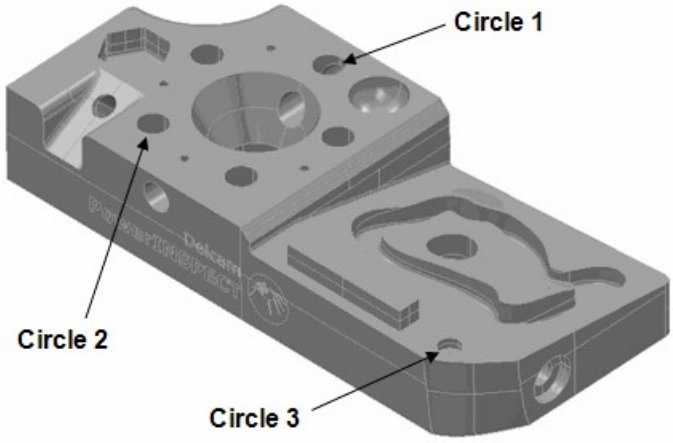
The details are displayed in the top portion of the explorer, and by moving down the levels (using the ⊞ icon), you can view and copy specific values, like the **Arc Centre Point** for example.



These values can then be used to determine CAD nominals for a **Geometric PLP Alignment**.

In this chapter, the **Geometry Explorer** is used to select the geometric items to be probed. The part is to be aligned using a series of probed circles, each with their own locally probed plane. The circular holes are located at each corner of the part:

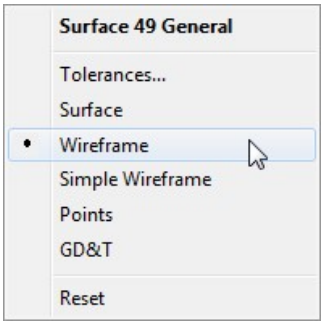
1 Click  to orientate the view so that all the circles can be seen.



2 Select the **Delay Measure**  button.

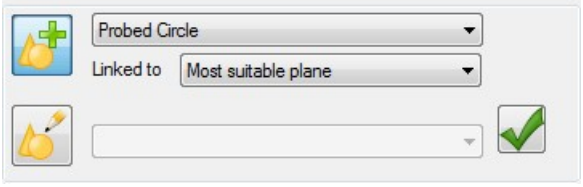
3 Select **Wireframe Checker**  .

4 Right-click in the CAD view and select **Wireframe** from the context menu.



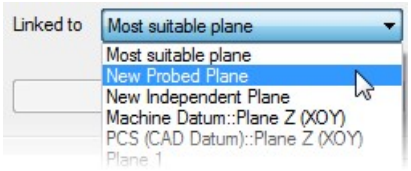
5 Move the cursor over circle 1. When it is highlighted in yellow, click to select it. The colour changes to red.

6 In the **Linked to** drop-down list of the **Geometry Explorer**, select **Most suitable plane**.

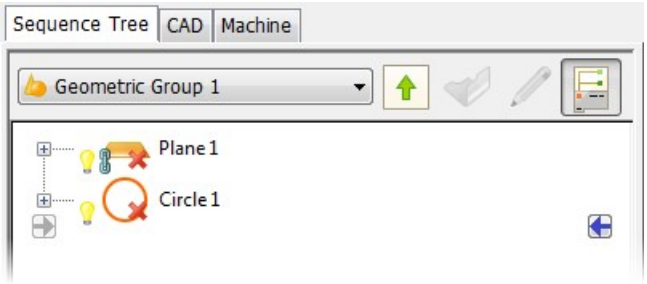


7 Click  to accept the feature.

If an individual plane (or the correct plane) is not assigned to a feature, you can create a new plane by selecting ***New Probed Plane*** *before accepting the feature.*



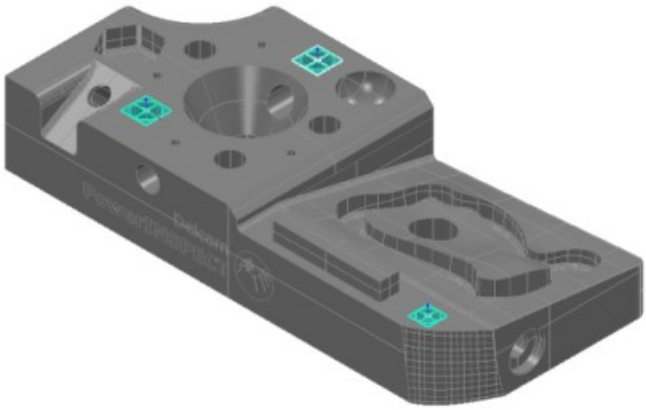
PowerInspect creates a Plane and Circle to be probed. This can be seen in the second portion of the **Geometry Explorer** and in the inspection sequence.



The crosses ✖ *next to the* ***Geometric Items*** *indicate that the items are not yet measured.*

In the CAD View the created Local Plane and Circle can be seen.

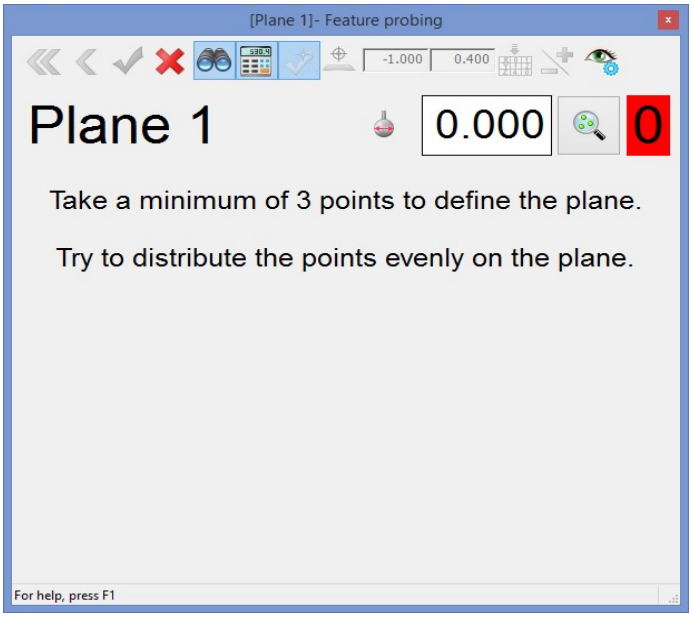
8 Repeat this process for the other two circles.



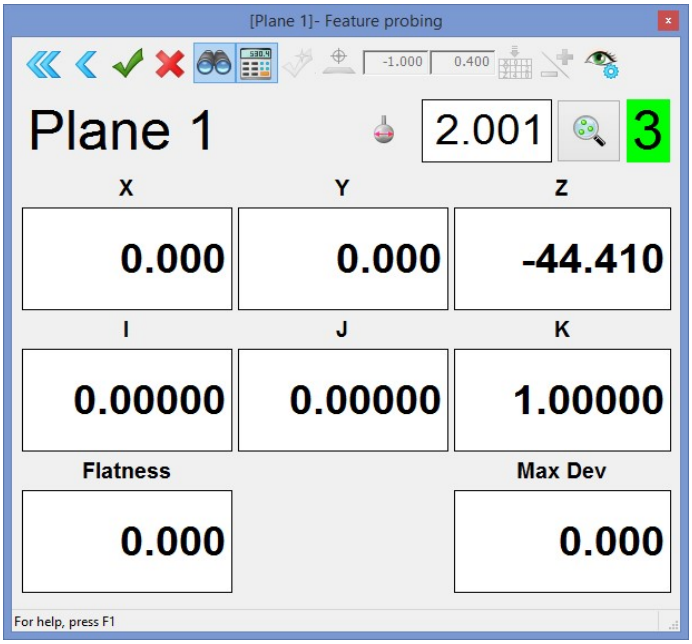
The Sequence Tree displays the six items (three Planes and three Circles), which can also be seen in the CAD View. When the part is securely fixed to the CMM bed, these items are ready to be probed.

9 Right-click any item in the inspection sequence and select **Play All** from the context menu.

Alternatively, click the **Play All**  button on the **Main** toolbar.
As with the **Geometric PLP** alignment, the first **Feature Probing** dialog appears asking you to take a minimum number of points to define that feature.



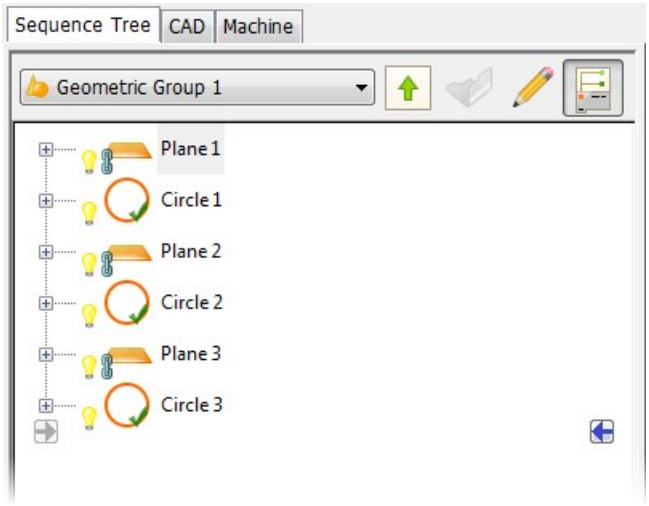
10 Probe three points around the Plane shown on the screen.



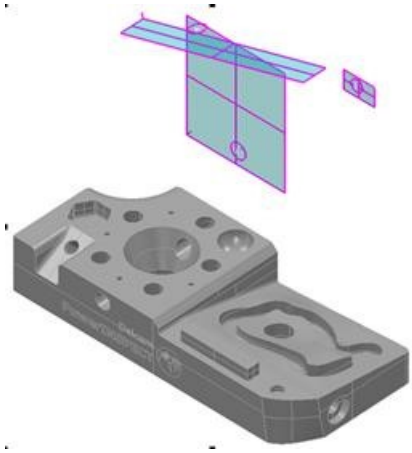
11 When the points have been probed, click  . The next item is displayed.

12 Continue the probing process until all items have been probed, making sure to probe them in the order that they are shown.

When all the items have been probed, the inspection sequence is updated.



The **CAD View** now contains the probed planes and circles, which require an alignment.



If the probed items can't be seen, click  *to resize the screen to fit, because the items may lie some distance away at this stage.*

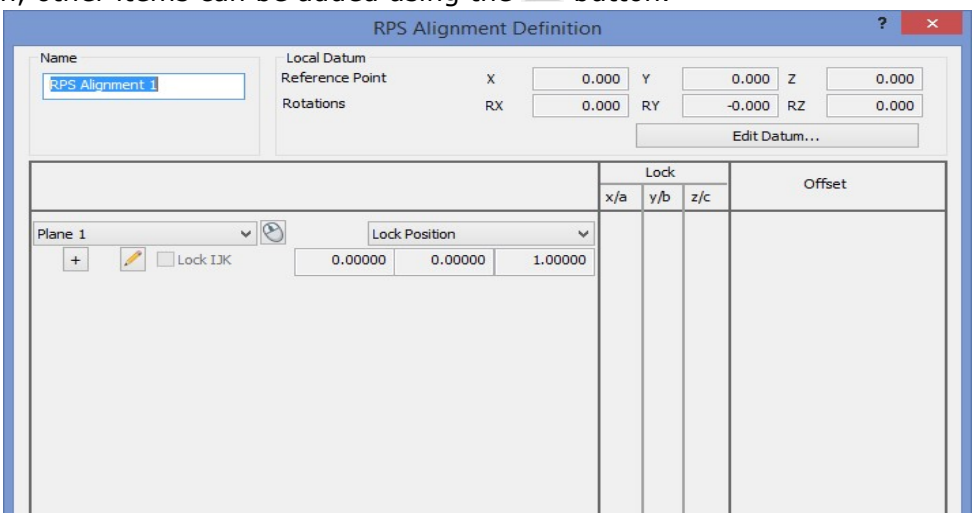
Generating an RPS Alignment

An **RPS** alignment will now be used to align the CAD model to the part.

1 Click  to close **Geometric Group 1**.

2 From the **Alignments**  toolbar in the **Item** toolbar, select the **RPS Alignment** button  . The **RPS Alignment Definition** dialog is displayed.

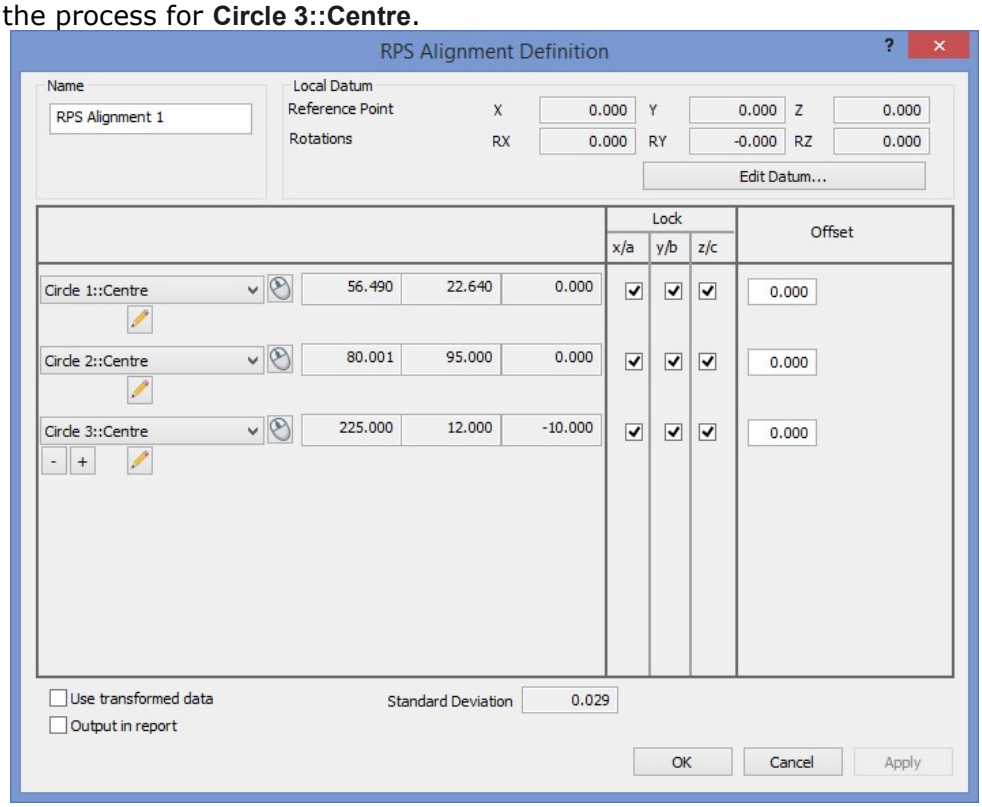
The items used for the alignment are selected in the drop-down menu ①. After an item is chosen, other items can be added using the + button.



3 Leave the **Name** and **Local Datum** unchanged, and select **Circle 1:: Centre** to specify the first position from the pull-down menu.

4 Select the + icon, to display a new drop-down menu. From the new menu, select **Circle 2::Centre** to specify the second position.

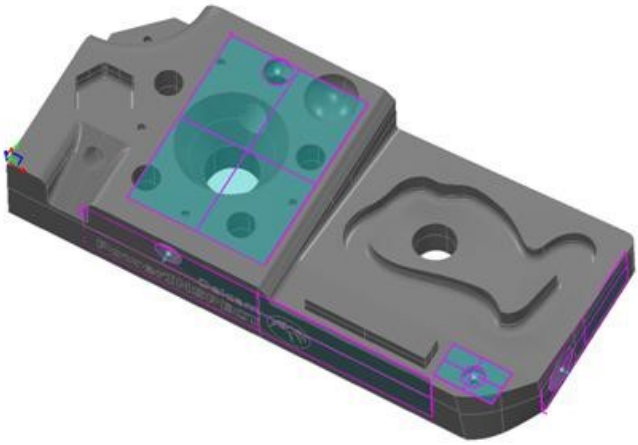
5 Repeat the process for **Circle 3::Centre**.



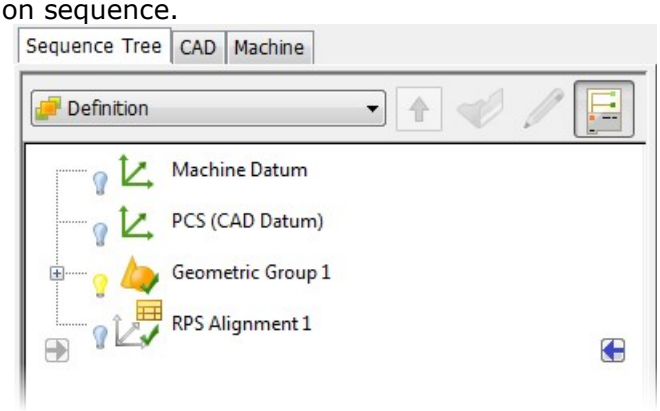
*The check boxes allow you to **lock** and **unlock** axes relative to particular items. This enables you to create a 3-2-1 alignment.*

6 Click **Apply** and then **OK**.
If all check boxes are selected, the **RPS** alignment is over constrained and behaves similar to a best fit.

PowerInspect aligns the probed positions to the CAD points given, satisfying **X**, **Y** and **Z** coordinates.

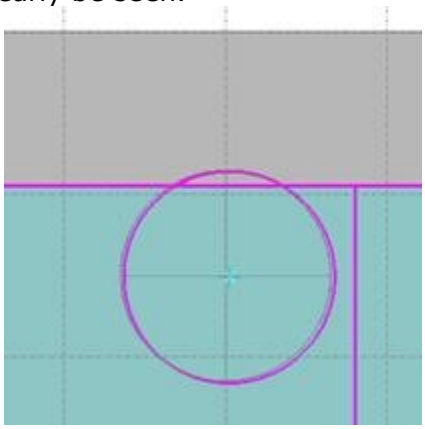


The CAD View is updated, with the **Probed Items** having been aligned to the **CAD Data**. All the criteria for the **RPS Alignment** have now been satisfied, and the alignment can be seen in the inspection sequence.



To check the alignment has worked, choose a variety of views and zoom into the aligned items to see if they match up.

For example, in this picture, a view looking down the Z-axis  zoomed into one corner, the alignment can clearly be seen.



To test the alignment is correct, move the probe around key points on the part and check the **Graphics** Window to see if the representation of the probe is in the same place.

If the alignment is correct, Inspection Groups can then be produced. Inspection will be discussed in the next chapter.

Best Fit From Points Alignment

The **Best Fit From Points** alignment  can be used to align from three or more points for which the CAD coordinates are known. PowerInspect uses these points that have been probed as geometric items to carry out a best fit to the rest of the CAD data. This method allows you to modify the alignment later, if necessary, by amending the details of the geometric items.

Before you use this alignment, you must create geometric items that provide at least three points. Points include centres of geometric items, such as the centre of a circle or sphere. You must also know the nominal position of the points: if you do not have this information, you can extract the coordinates using the **Geometry Explorer** tab. This method is shown below.

*A **Best Fit** alignment can produce the same results as a **Three Spheres** alignment if three spheres are used to supply the three points. A **Three Spheres** alignment uses tooling balls; it is predominately used in the aerospace industry.*

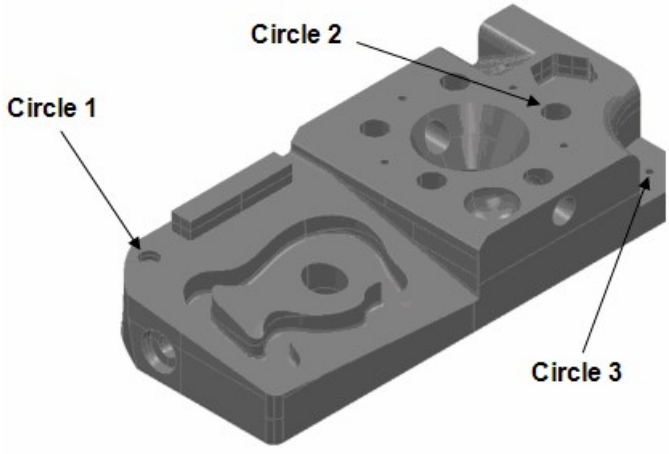
Generating a Best Fit From Points Alignment

1. Click  to create a **New Document using the Wizard**. In the Wizard, select **Measurement with a single CAD part** and click **Next**.
2. Browse for **Demoblock2008(CMM+Arm).dgk** and open the file.
3. Leave the offsets and tolerances unchanged, and click **Next**. The **Variables** page is displayed.
4. Browse for an **HTML Report Template**. The report variables are displayed.
5. Click **Finish**.

Choosing the Geometric Items

As with the previous methods, you must decide which geometric items will determine the alignment. In this example, the part is to be aligned using a series of probed circles, each with their own locally probed plane. The centres of these **Circles** are then used to determine the Best Fit points.

As in the previous example, the **Geometry Explorer** is used in conjunction with the **Wireframe Checker**  to select the items.



1 Click  to orientate the view so that all the circles can be seen.

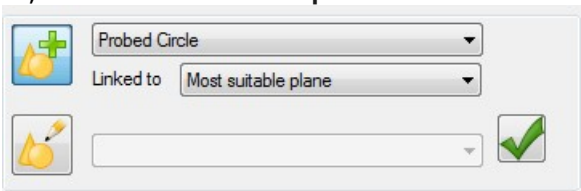
2 Select **Delay Measure**  .

3 Select **Wireframe Checker**  .

4 Right-click in the CAD view and select **Wireframe** from the context menu.

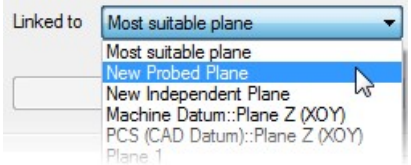
5 Select the top wireframe of Circle 1.

6 In the **Geometry Explorer**, select **Most suitable plane** in the **Linked to** drop-down list.

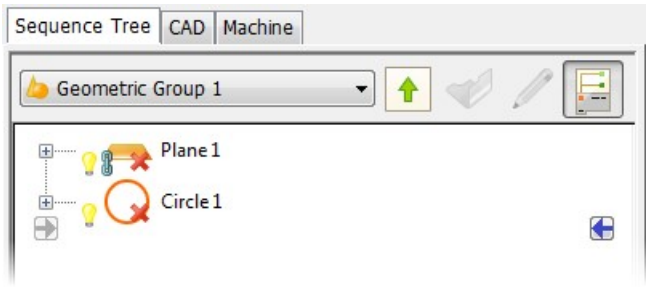


7 Click  to accept the feature.

*If an individual plane (or the correct plane) is not assigned to a feature, you can create a new plane by selecting **New Probed Plane** before accepting the feature.*

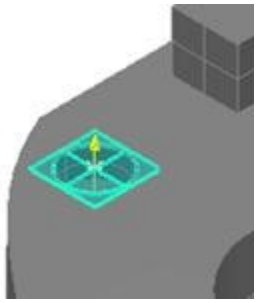


PowerInspect creates a plane and circle to be probed. This can be seen in the **Geometry Explorer** and in the Sequence Tree.



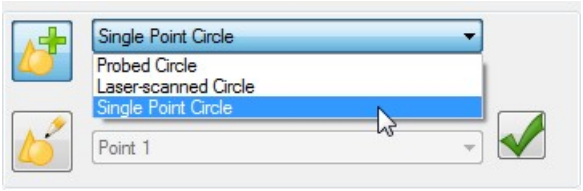
*The exclamation marks ✖ next to the **Geometric Items** indicate that they are not yet measured.*

Looking at the **CAD View**, the created Local Plane and Circle can also be seen.



8 Repeat the selection process for **Circle 2**.

9 Select the **Circle 3** wireframe. In the **Geometry Explorer**, change the **Circle** option to **Single Point Circle** before clicking .



The Sequence Tree and the CAD view display the six items (three Planes, two Circles and a Point). With the part securely fixed to the CMM bed, these items are now ready to be probed.

10 Right-click any item in the inspection sequence and select **Play All** from the context menu.

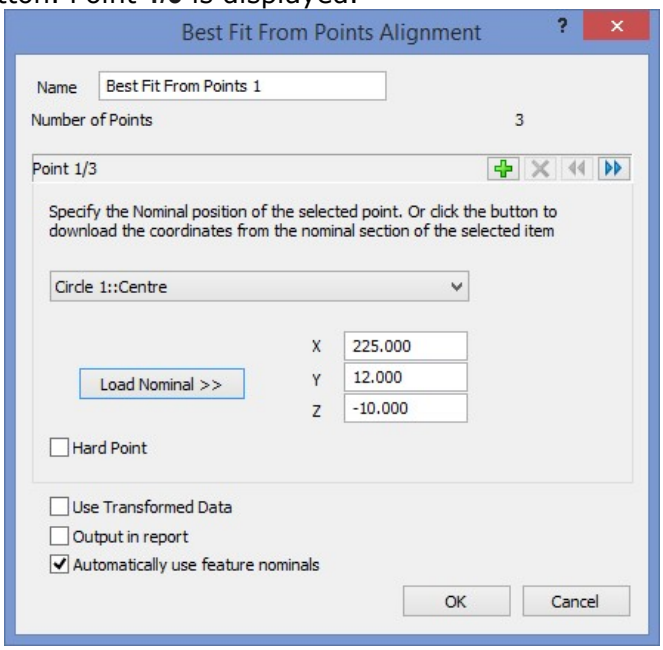
Alternatively, click the **Play All** button from the **Main** toolbar.

11 Probe all features as directed.

Generating a Best Fit From Points Alignment

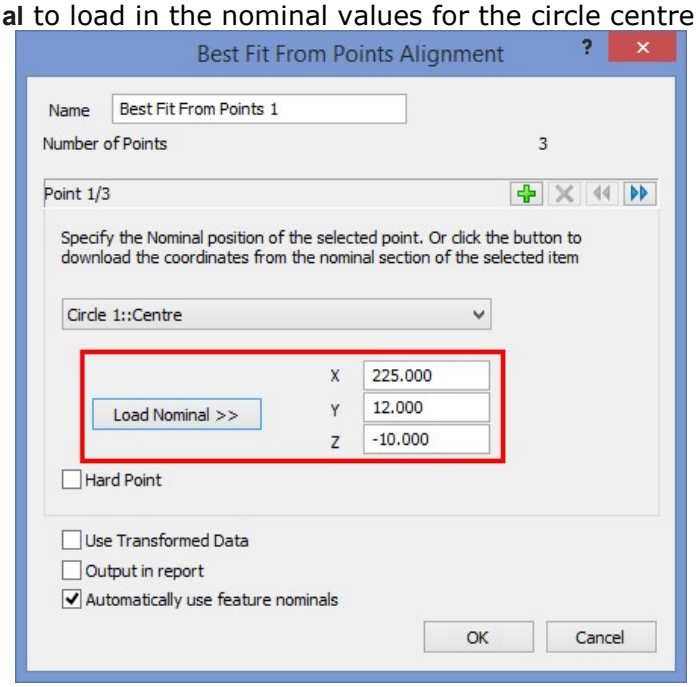
1 Select  to close Geometric Group 1.

2 From the **Alignments**  toolbar, in the **Item** toolbar, select the **Best Fit From Points Alignment**  button. Point **1/3** is displayed.



3 Select **Circle 1: Centre** from the drop-down list.
You can type the **X**, **Y** and **Z** Coordinates for **Circle 1: Centre**. Alternatively, because the nominal values have been extracted from the CAD model using **Geometry Explorer**, you can enter them using the **Load Nominal** button.

4 Select **Load Nominal** to load in the nominal values for the circle centre.



*You can make the first point a hard point by selecting the **Hard Point** check box. This means that PowerInspect will match the geometric point to the CAD nominal exactly rather than trying to best fit it with the other features. This option will not be used in this example.*

5 Select to display **Point 2/3**.

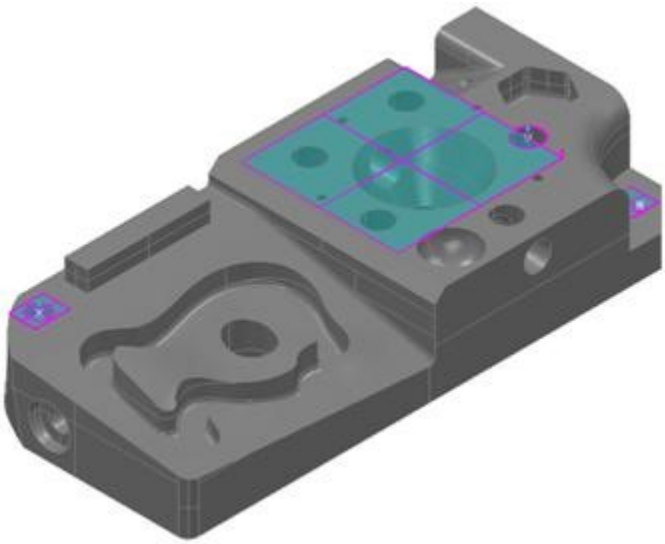
6 Select **Circle 2:Centre** and then select **Load Nominal**.

7 Select to display **Point 3/3**.

8 Select **Point 1:Centre** and then select **Load Nominal**.

9 Select **OK** to create the **Best Fit** alignment.

*A minimum of three points are required for the alignment. You can add more points by selecting **New** .*



4. Datums

Datums (origins) are reference points on a part. Additional datums can be created within an inspection sequence to reference features whose dimensional properties are relative to the datum.

Before you create a datum, you will need an aligned part. Before starting the following examples, load the **Demoblock2008(CMM+Arm).dgk** part and align it using the **Geometric PLP aligment**, described in chapter 3.

Datum toolbar

The **Datum** toolbar is available from both the **Item** and **Geometric** toolbars.

There are six methods for creating a datum in PowerInspect:

 Create a datum.

 Create a datum by shifting the origin of an existing datum to explicit CAD coordinates, or to a point feature.

 Create a datum by rotating an existing datum around one of its axes by a specified angle.

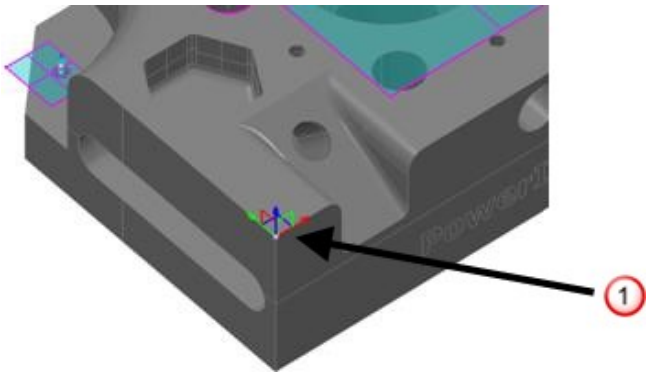
 Create a datum by aligning an axis of an existing datum to the vector of a specified 3D item.

 Create a datum by aligning an axis of an existing datum to the vector of a specified 2D item.

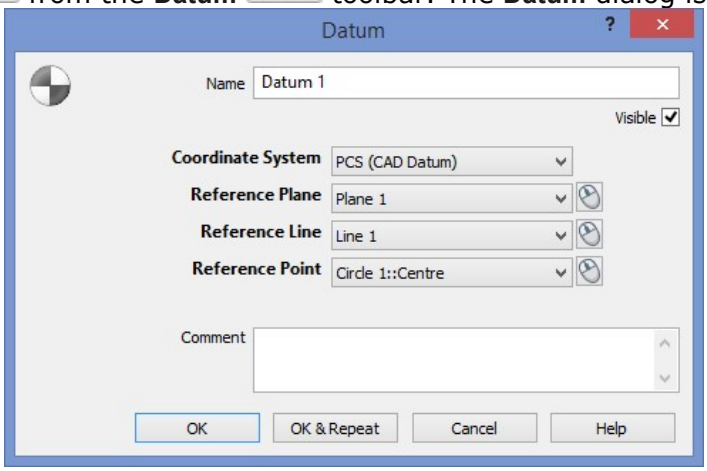
 Create a datum by applying a transformation matrix to an existing datum.

Creating a datum

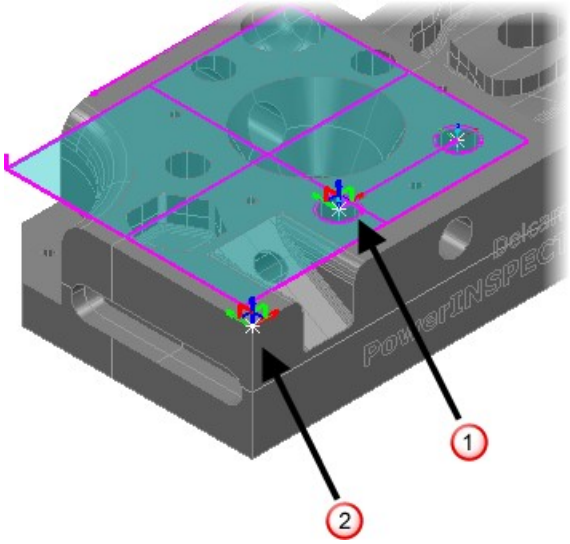
1 Select the light bulb icon of the **PCS (CAD Datum)** in the Sequence Tree to display the default datum (1).



2 Select **Datum**  from the **Datum**  toolbar. The **Datum** dialog is displayed.



3 Complete the dialog as shown above and then select **OK**.



A new datum, **Datum 1** (1), is created at the centre of Circle 1.

The original datum, **PCS** (2), is still shown.

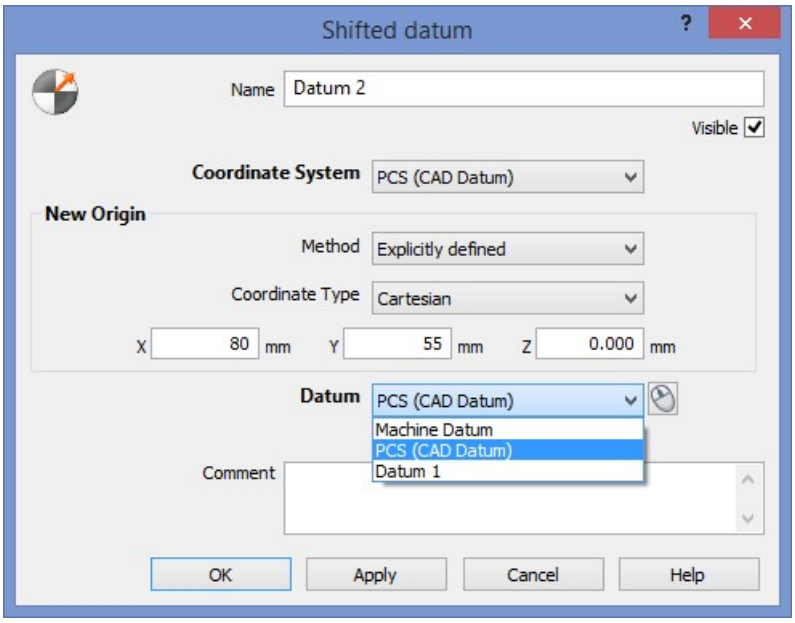
Creating a shifted datum

There are two methods you can use to create a shifted datum.

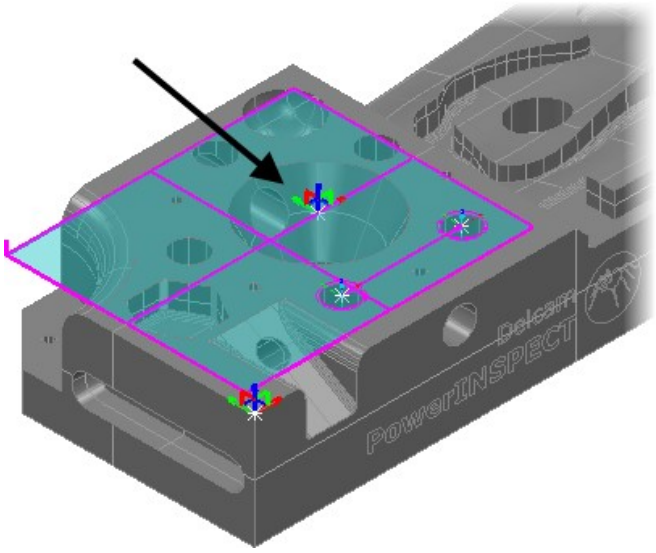
To explicitly define the datum position:

1 Choose **Shifted datum**  from the **Datums** toolbar.

2 Enter the coordinates as shown in the dialog:



3 Select **OK** to create the new Datum. Datum 2 is shown in the CAD view.

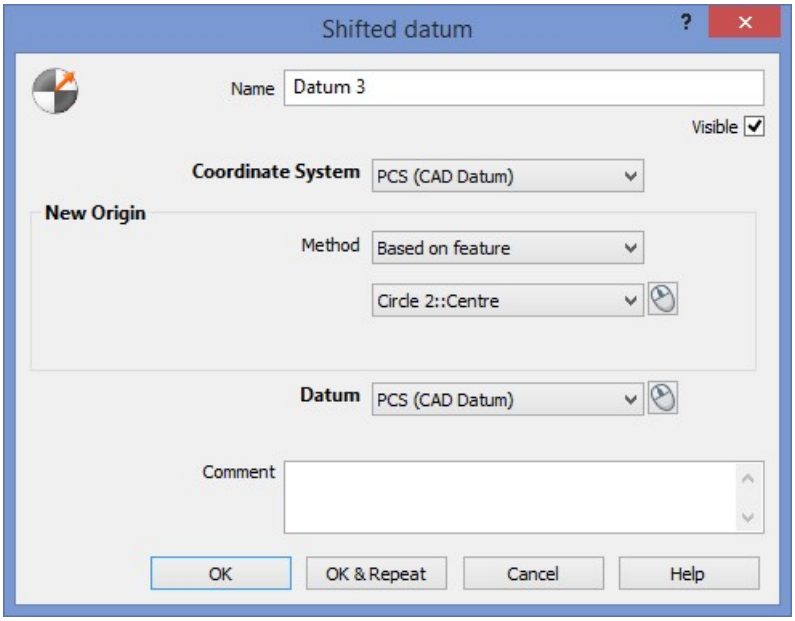


To base the datum position on a feature

1 Choose **Shifted datum**  from the **Datums** toolbar.

2 Select **Based on feature** from the **Method** drop-down list.

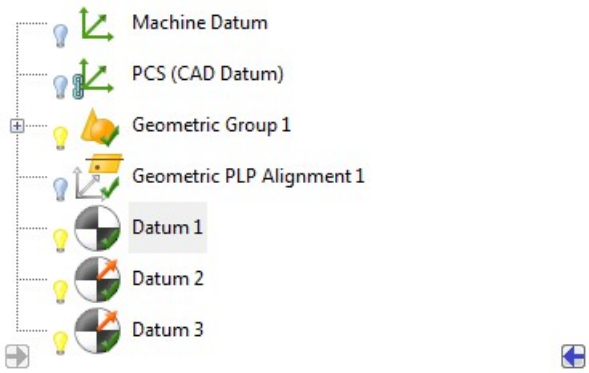
3 Select **Circle 2::Centre** as the feature.



Ensure **PCS (CAD Datum)** is selected as the Datum required to shift.

4 Select **OK** to create a new datum.

All datums created are shown in the Sequence Tree:

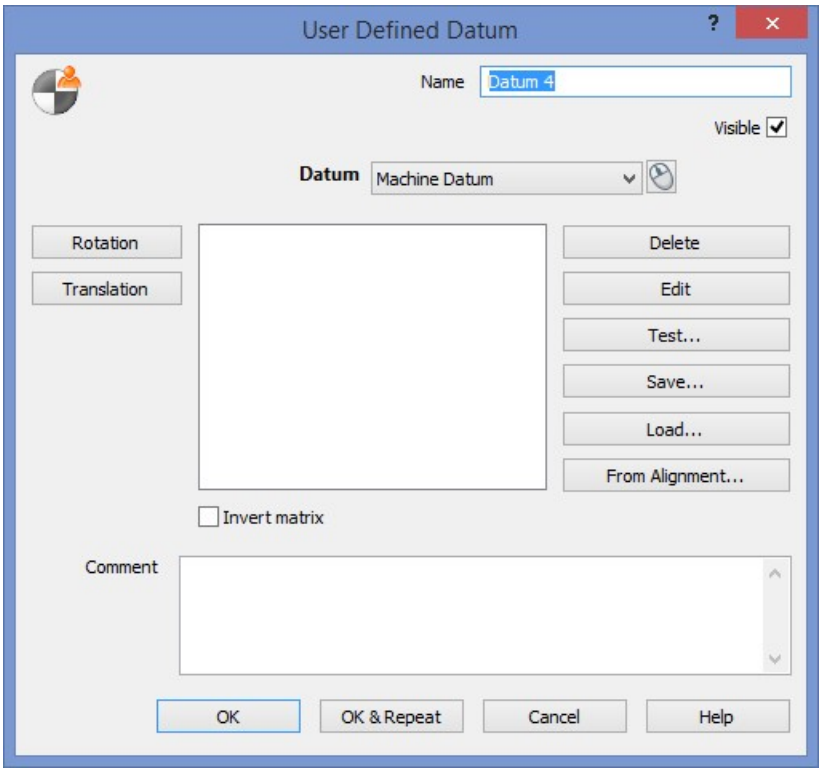


Creating a user-defined Datum

Use the user-defined datum button to create a datum by applying a transformation matrix to an existing datum.

To create a user-defined datum

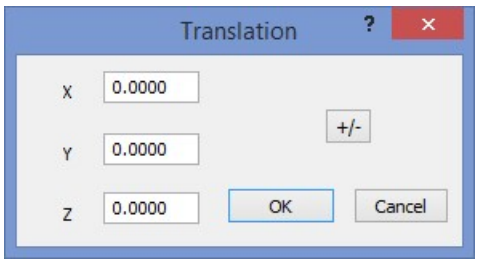
1 Click  on the **Datum** flyout. The **User-defined Datum** dialog is displayed.



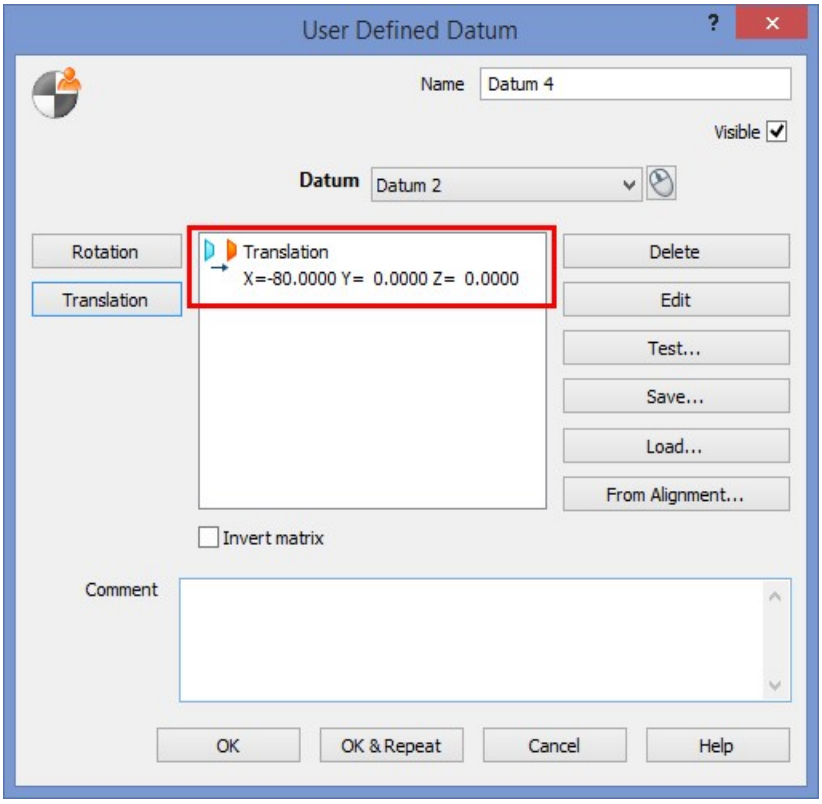
2 In the **Datum** drop-down list, select the datum to which you want to apply the transformation.

Alternatively, click  to select a datum from the CAD view using the cursor.

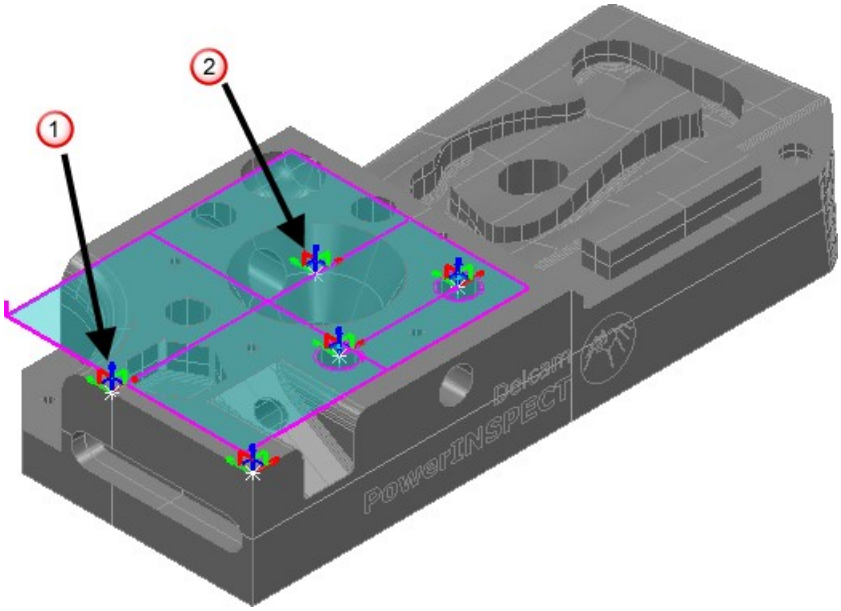
3 Select Datum 2 and click **Translation**. The Translation dialog is displayed.



4 In the **X** box, enter **-80** and click **OK**. The **User-defined Datum** dialog is updated.

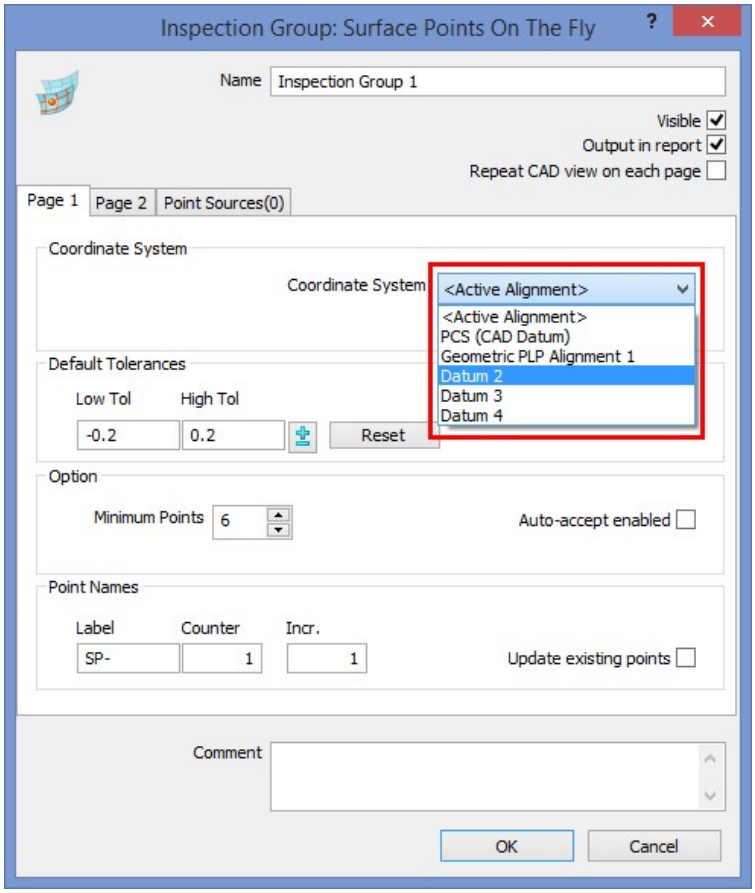


5 Click **Apply**, then **OK** to close the dialog. A new datum (1) has be created on the CAD model at the specified translation from Datum 2 (2).



Using Datums as local coordinate systems

Datums can be specified as the local coordinate system for individual inspection groups, section groups, point cloud groups, and point clouds using the **Coordinate system** list in each item's definition dialog. This enables you to create reports where points are measured relative to any point on the part.



When the **Coordinate system** of a group is changed from a datum, it has the following effects:

- Nominal target points move on the CAD model. This may cause them to become invalid when you change the definition of the datum, or when you change measures.
- Measured points remain in the same place on the CAD model, but are reported using the new coordinate system.
- For section groups, coordinates are reported in the new coordinate system.

5. Inspections

This chapter describes the main inspection techniques available in PowerInspect. After creating a suitable alignment, the part can be inspected against the CAD data. This measures the accuracy of the part and highlights any errors (deviations outside the specified tolerances), assuming the CAD data is correct.

The tolerance values, which are used by inspection groups, can be changed globally (though the use of the **Start-up Wizard**, or **Measures > Parameters** menu option), or individually in each inspection group.

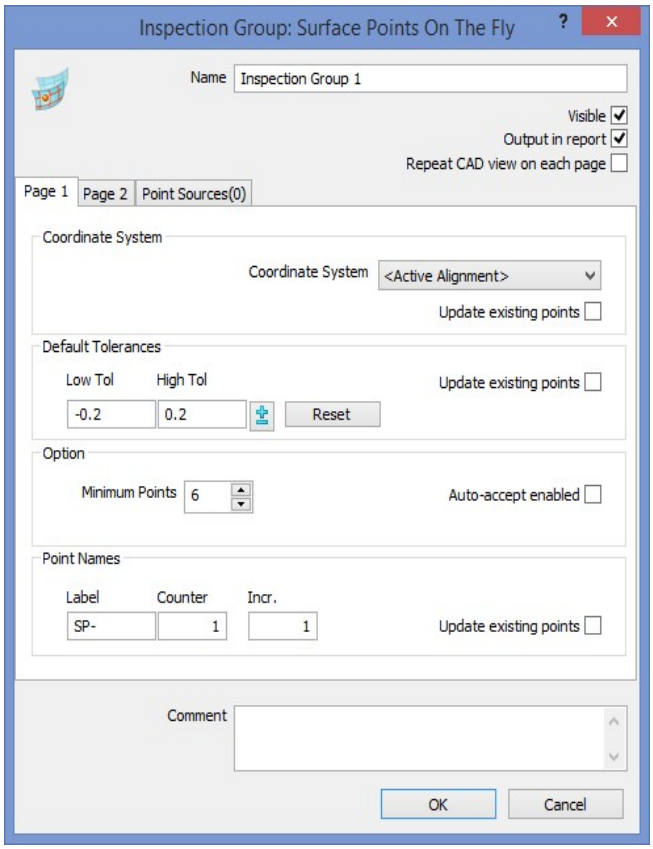
Surface Inspection Group – Points on the fly

1. Click **New Document Wizard**  to open the **New Inspection Session** wizard.
2. Select **Measurement with a single CAD part** and click **Next**.
3. Click **New** and browse to **DemoBlock2008(CMM+Arm).dgk**.
4. Leave the offsets and tolerances unchanged, and click **Next**. The **Variables** page is displayed.
5. Browse for an **HTML Report Template**. The report variables are displayed.
6. Click **Finish**.
7. Create an alignment using one of the previously learnt methods.

The new session is now ready for inspection.

1 Deselect the light bulb for **Geometric Group 1**  . This removes the contents of the group from the CAD View, making it less cluttered when taking the surface inspection.

2 Click the **Surface Inspection Group**  and then select **On the fly Surface Points**  .



With this dialog, you can determine the name of the group, choose whether to take surface points, edge points, or hem edge points, and choose whether to take points on the fly or guided points. You can also specify the minimum number of points to be taken, as well as setting counter values, labels and adding comments.
This dialog also allows you to change the local offsets and tolerances for the group; to specify whether the group is included in the Report; to rename the points in the group; and to change the point counter and its increments.
In this example, the dialog is used to create a group in which points are taken on the fly.

3 Select the **Output in Report** check box, leave the other settings unchanged, and then click **OK**.

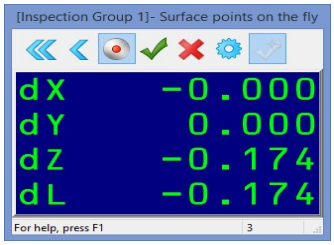
If **Delay Measure** is selected, you can display the **Surface points on the fly** dialog by.

- right-clicking the inspection group  icon in the Sequence Tree and selecting **Play Item** from the context menu.

or

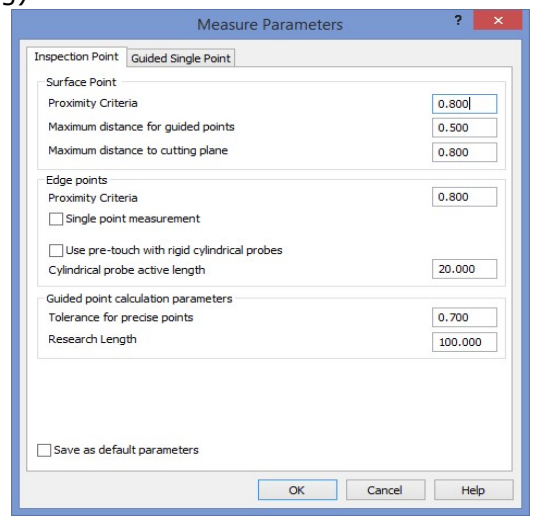
- selecting the inspection group in the Sequence Tree and clicking **Play Item**  on the **Main** toolbar

The **Surface points on the fly** dialog is displayed. A minimum number of points (set in the **Surface Inspection Group** dialog – in this case **6**) need to be taken to satisfy the inspection.
As the points are probed the **Surface Points on the Fly** dialog displays the deviation of the measured points. These are displayed in all three axes, **dX**, **dY** and **dZ**, as well as a resultant deviation, **dL**.

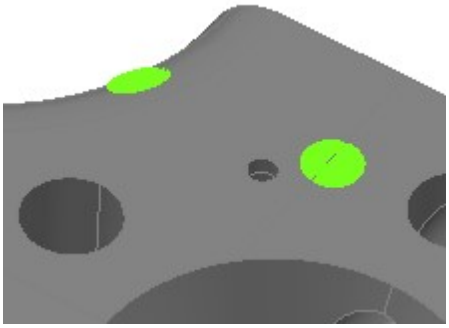


This dialog has similar options to that of the **Feature Probing** dialog with addition of a **Suspend Recording**  button, which allows you to see the points without recording them.

There is also has a **Parameters**  button. Click this button to display the **Measure Parameters** dialog, which contains a summary of the parameters used for inspection points and guided points (such as proximity criteria, which may need to be increased if points are not registering).



4 Probe any number of points on the part surface, making sure to satisfy at least the minimum number of points set in the **Surface Inspection Group** dialog.



5 When you have taken enough points, click  .

6 Select the **Report** tab to view the surface inspection group results. All the items chosen to be output to the report are displayed; scroll down the report to view the inspection group results.

To export the report so that it can be viewed in a web browser,

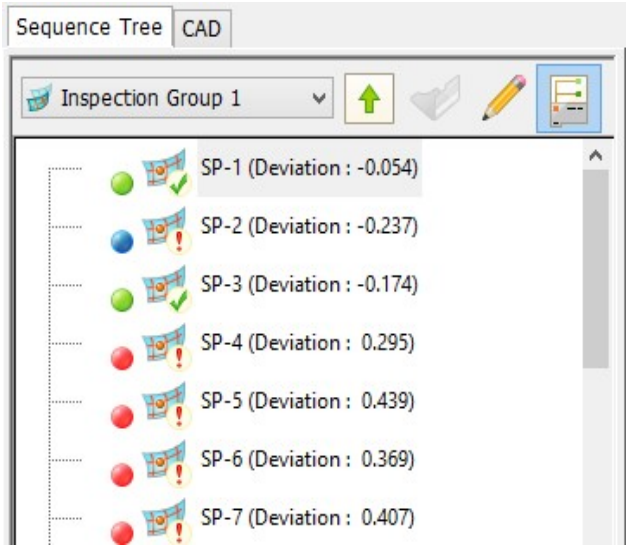
*click **Export Report***  *on the **Main** toolbar, and save the report as Report.mht.*

7 The results can also be viewed from within the CAD View and the Sequence Tree. To view the points that have been taken, select Inspection Group 1 in the Sequence Tree and click the **Open Group**  button, or select **Open Group** from the right-click popup menu.

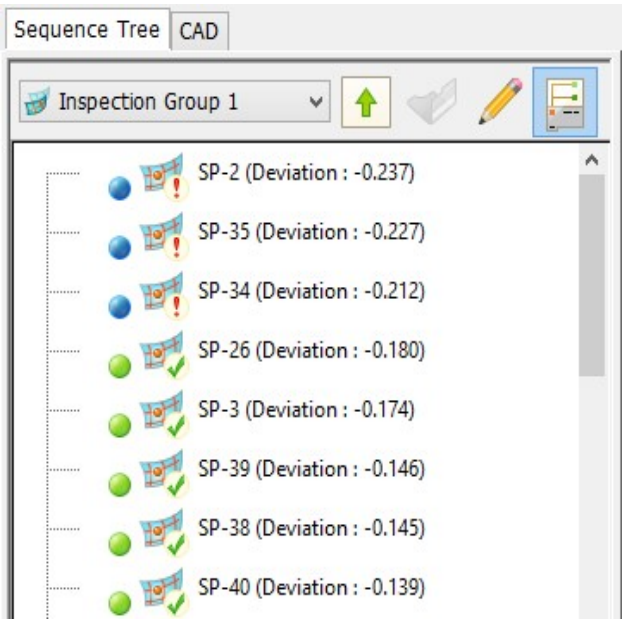
*Using the ⊞ button to open the group displays the points in the Sequence Tree, but does not display the statistical data in the **Print Preview**. To view this, the group must be opened.*

*When the group is open, the points can be individually highlighted, deleted or modified (For further details, see Chapter 9 **Modifying Elements**). Double-click on a point to open the **Guided Point** dialog which displays its position, deviations and so on.*

8 By default, an open group is sorted by the number label of the points taken (SP-1, SP2, SP3, etc….). Right-click anywhere inside the group and select **Sort by Deviation** from the popup menu.



This sorts the points by the amount by which they deviate from zero, starting at the lowest and working up to the highest deviations.



Surface Inspection Group – Edge Points

You can use PowerInspect to inspect edge points.

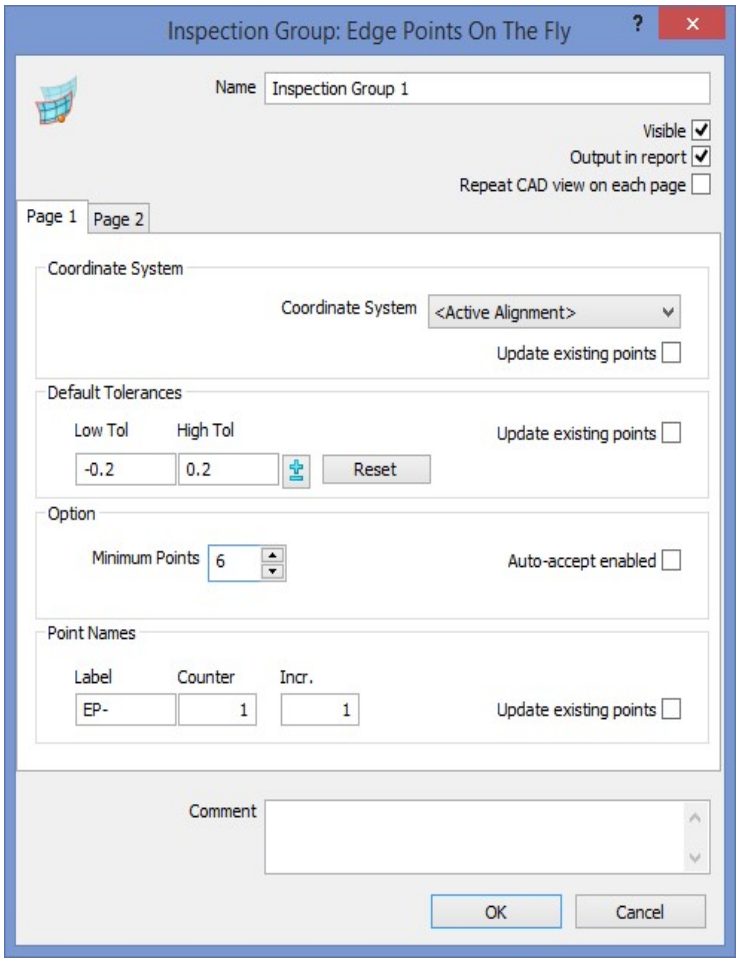
1 Click  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part**, and click **Next**.

2 Browse for **Demoblock2008(CMM+Arm).dgk** and click **Finish** to open the file.

3 Create an alignment, using one of the previously learned methods.

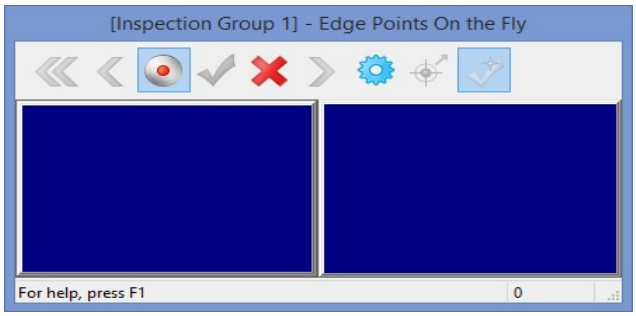
4 Click **Delay Measure**  .

5 Click the **Surface Inspection Group**  and then select **On the fly Edge Points**  .



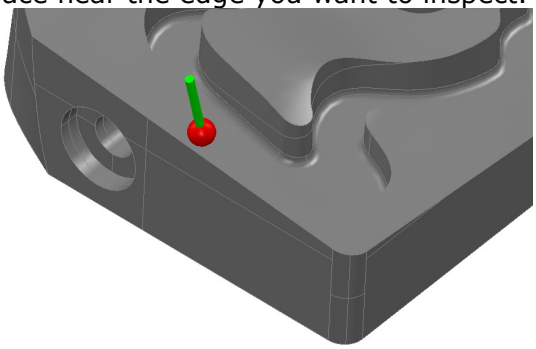
6 Ensure **Output in Report** is selected, leave the other settings unchanged, and then click **OK**.

7 Select the **Surface Inspection Group**  icon that has now been added to the Sequence Tree and click **Play Item**  .

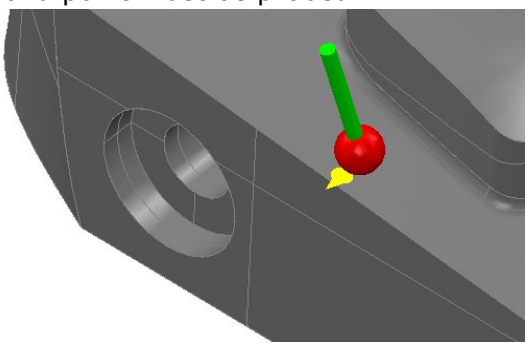


The **Edge Points On the Fly** dialog is displayed. A minimum number of points (in this case **6**) must be taken to satisfy the inspection. This number can be defined on the **Surface Inspection Group** dialog.
Using this method, PowerInspect requires two probed points to define the edge to be inspected.

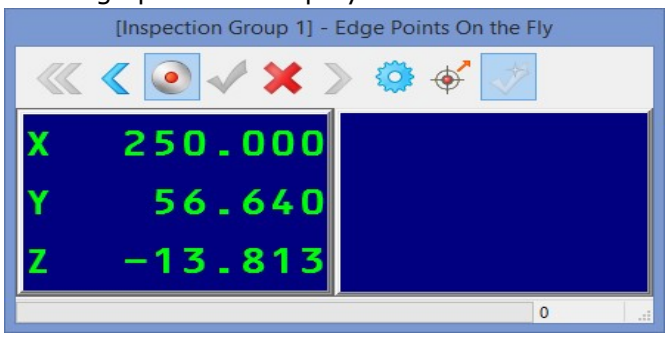
8 Probe a point on the surface near the edge you want to inspect.



PowerInspect displays a yellow mark to represent the point and a yellow triangle marker to indicate where the second point must be probed.

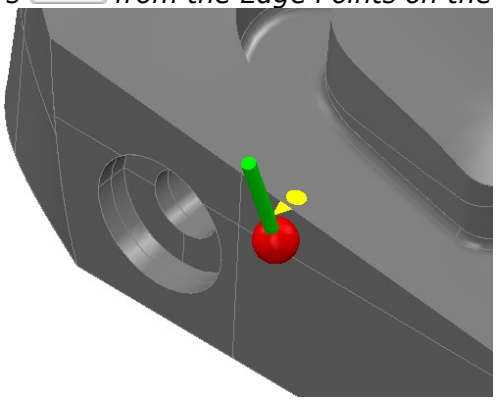


The co-ordinates of the edge point are displayed on the left of the dialog.



9 Probe the edge indicated by the marker to record the edge point.

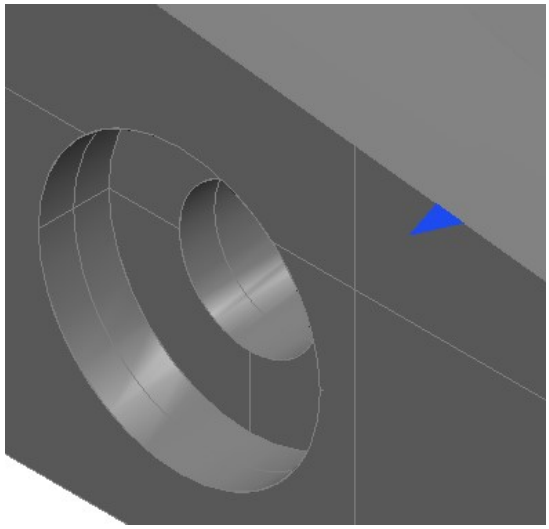
The point must be probed precisely. The proximity criteria can be changed by selecting Parameters from the Edge Points on the Fly dialog.



When the point is probed, the deviation is indicated on the right of the dialog.

10 Repeat the process above to take five further edge points.

11 When probing is complete, click .
PowerInspect displays the edge points on the model as triangles. The colour scheme used by the triangles to represent the deviation is the same as that used for **Confetti**. You can customise the display by displaying labels, for example.

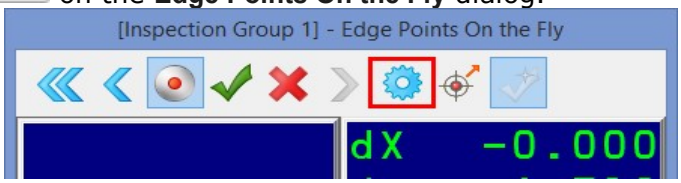


Edge points - Single point option

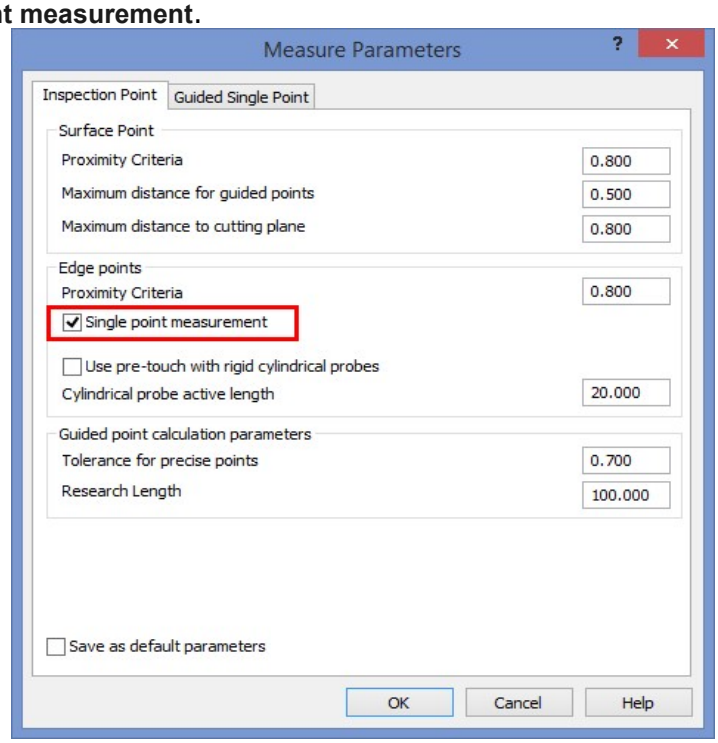
Use the single point option to create edge points using one touch. This method is faster, but potentially less accurate.

1 Follow points 1 to 8 in the Edge Points On The Fly section.

2 Click **Parameters**  on the **Edge Points On the Fly** dialog.

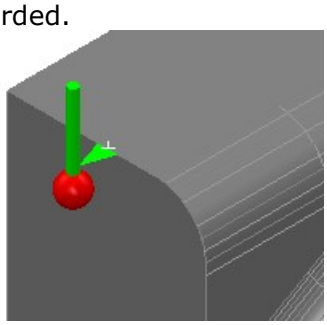


3 Select **Single point measurement**.

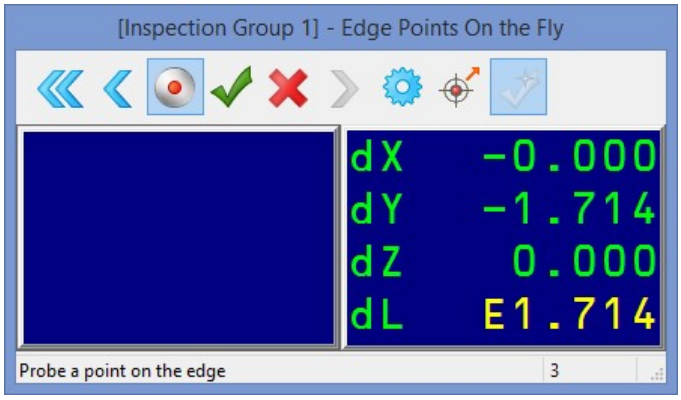


4 Click **OK**.

5 Probe a point on the surface near the edge you want to inspect. A triangle is displayed to show that the point has been recorded.



The deviation of each point is displayed on the right-hand side of the **Edge Points On the Fly** dialog as it is probed.



6 Continue probing the edge until you have taken the required number of points.

7 When probing is complete, click  .

PowerInspect displays the edge points on the model as triangles. The display can be customized by displaying labels, for example. The triangles represent the deviation using the same colour scheme as ***Confetti****.*

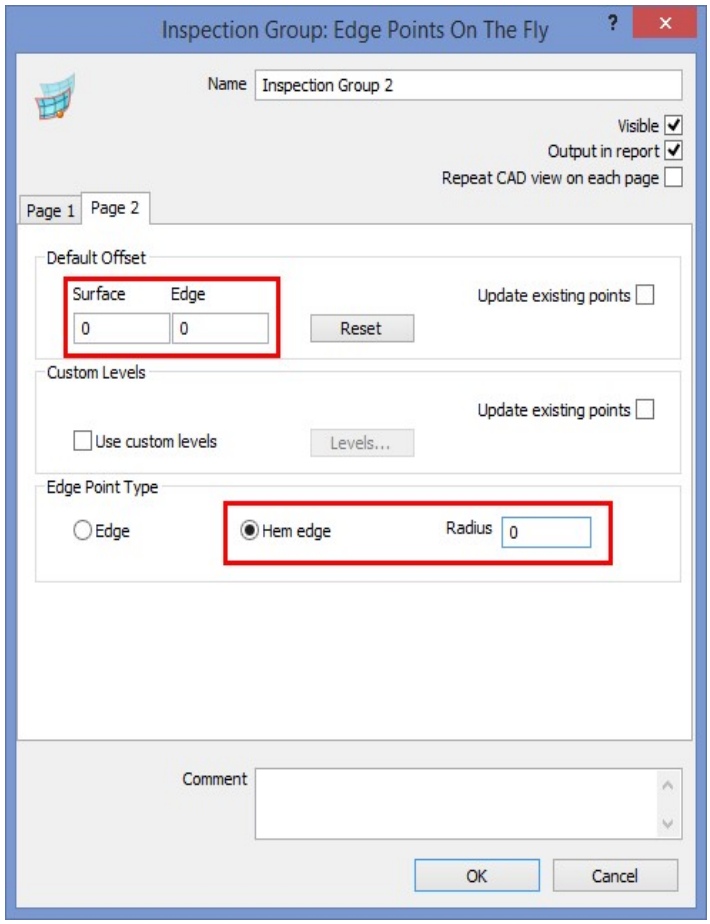
Surface Inspection Group – Hem Edge Points

The **Surface Inspection Group** dialog contains a **Hem edge points** option that enables you to measure a rounded edge on a part.

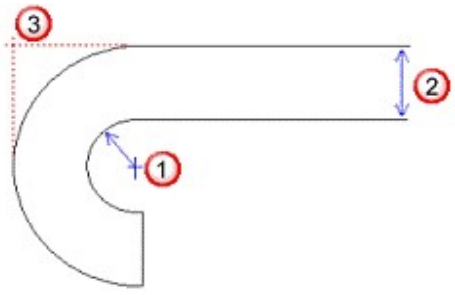
To create a surface inspection group to measure a hem edge.

1 Click the **Surface Inspection Group**  and then select **On the fly Edge Points**  .

2 Click **Page 2** and the folllowing dialog box is displayed below:



3 Select **Hem edge points**, and enter the radius of the edge ① and the thickness of the surface ② in the **Surface Offset** box.



If the surface is to be probed from the top, enter the surface offset as a positive value. If the surface is to be probed from the bottom, enter the surface offset as a negative value.

4 Complete the dialog and click **OK** to save your changes.

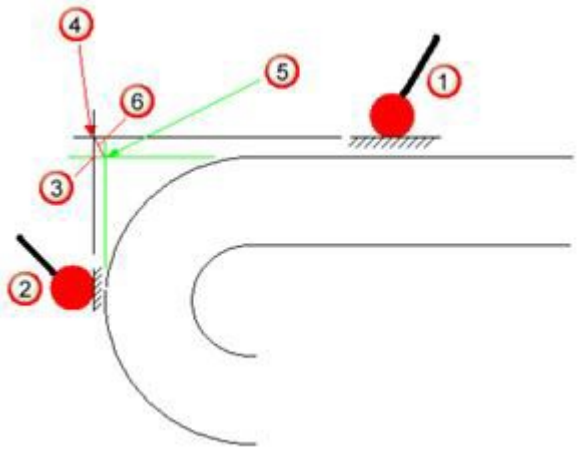
5 The nominal position of the hem edge is calculated as the sum of the **Radius** and the **Surface offset** (3).

Probing the Hem Edge

For each hem edge point measurement you want to make, you must probe two points:

- The first point (1) must be located on the nearest surface of the CAD model. This sets the nominal plane and direction used to calculate the position of the hem edge point.
- The second point (2) must be located on the edge itself. It is projected on the nominal plane (3) to determine the actual position of the hem edge (4).

PowerInspect then uses the projection and the hem-edge nominal position (5) to calculate the deviation parallel to the nominal plane, and perpendicular to the nominal edge direction (6).



Surface Inspection Group – Guided Points

There are a number of ways to create a surface inspection from **guided** or **targeted** points. Guided points can be used to check the accuracy of a feature for which the CAD coordinates are known.

Using the Dynamic Points Editor

1 Select  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part**, and then click **Next**.

2 Browse for **Demoblock2008(CMM+Arm).dgk** and click **Finish** to open the file.

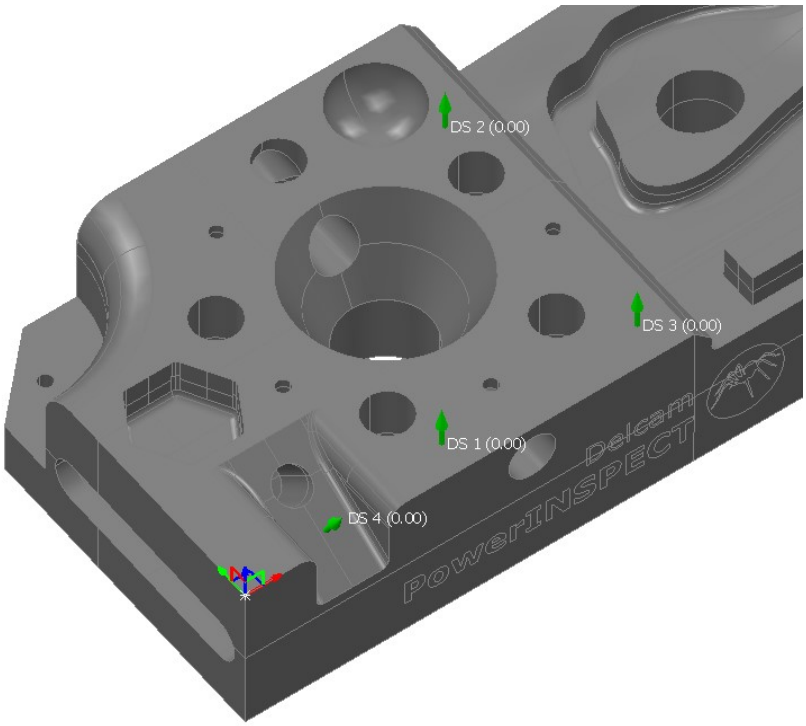
3 Create an alignment using one of the previously learnt methods.

4 Select the **Dynamic Points Editor**  from the **Mouse Context** toolbar.

The cursor changes to a target sight  *in the CAD view. This allows you to select the points on the CAD surface that are to be inspected as guided points.*

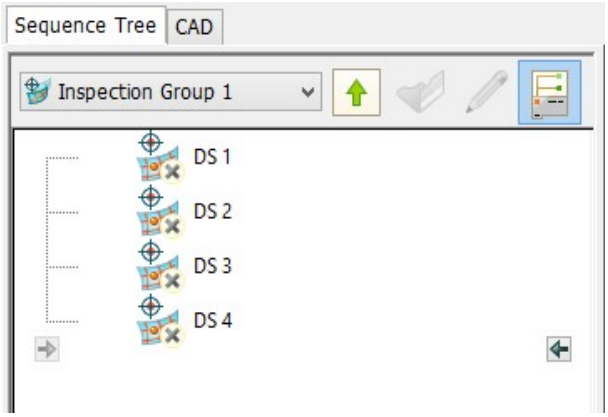
Points are created by double-clicking on the surface of the CAD model.

5 Create a number of target points on the model as shown below.

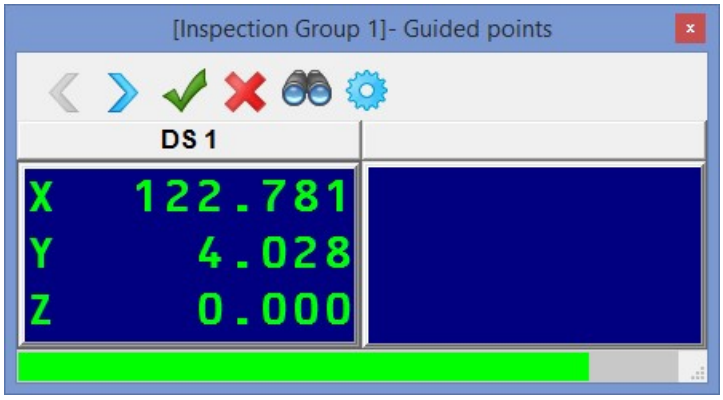


6 From the **Dynamic Points Editor** toolbar, click **Create a Guided Surface Inspection Group**  .

The points are added to a surface inspection group in the Sequence Tree.



7 Select **Play Item**  from the **Main** toolbar. PowerInspect switches to full-screen measure mode, and the first dynamic point to be probed is identified by the target icon  . As you probe each point, the target moves to display the next point in the inspection sequence.



*As you move the probe, a green bar at the bottom of the **Guided Points** dialog indicates its proximity to the target.*

*Only points that are within proximity criteria specified in the **Measure Parameters** dialog are accepted. To view or modify these parameters, click*  .

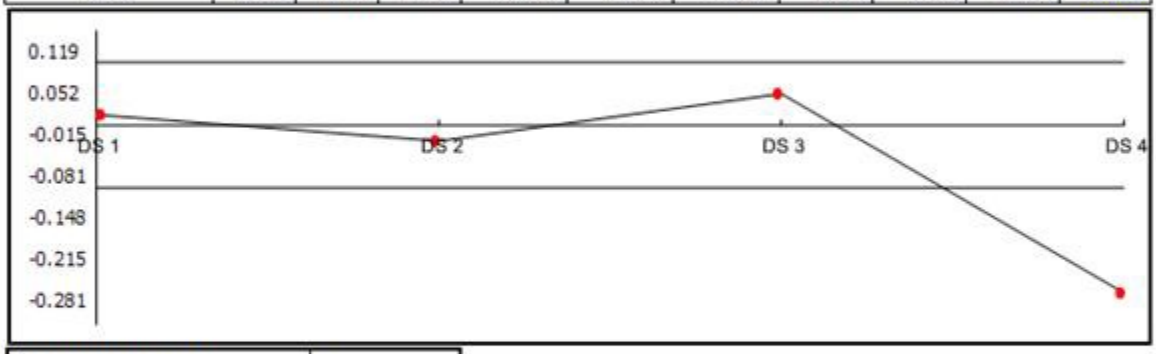
8 Probe all points in the order indicated.

9 Check the results in the **Report** tab.

Inspection Group 1

Datum - *Geometric PLP Alignment 1*

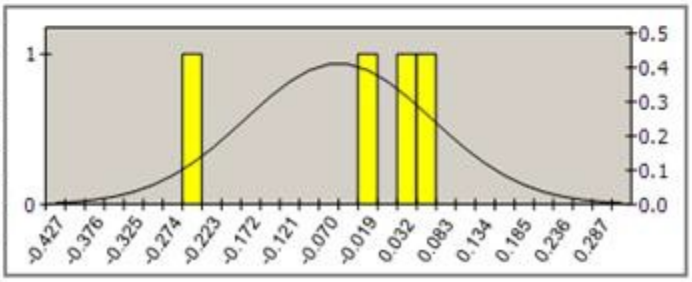
Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
DS 1	0.000	-0.100	0.100	61.073	9.801	0.000	0.000	0.000	0.015	0.015
DS 2	0.000	-0.100	0.100	115.964	81.009	0.000	0.000	0.000	-0.028	-0.028
DS 3	0.000	-0.100	0.100	118.603	11.753	0.000	0.000	0.000	0.048	0.048
DS 4	0.000	-0.100	0.100	31.768	11.928	-12.361	0.000	0.131	-0.228	-0.263



Number of points	4
Out of tolerance	1
Performance	75%

Mean	-0.057
Std. Deviation	0.122

Max. Value	0.048
Min. Value	-0.263



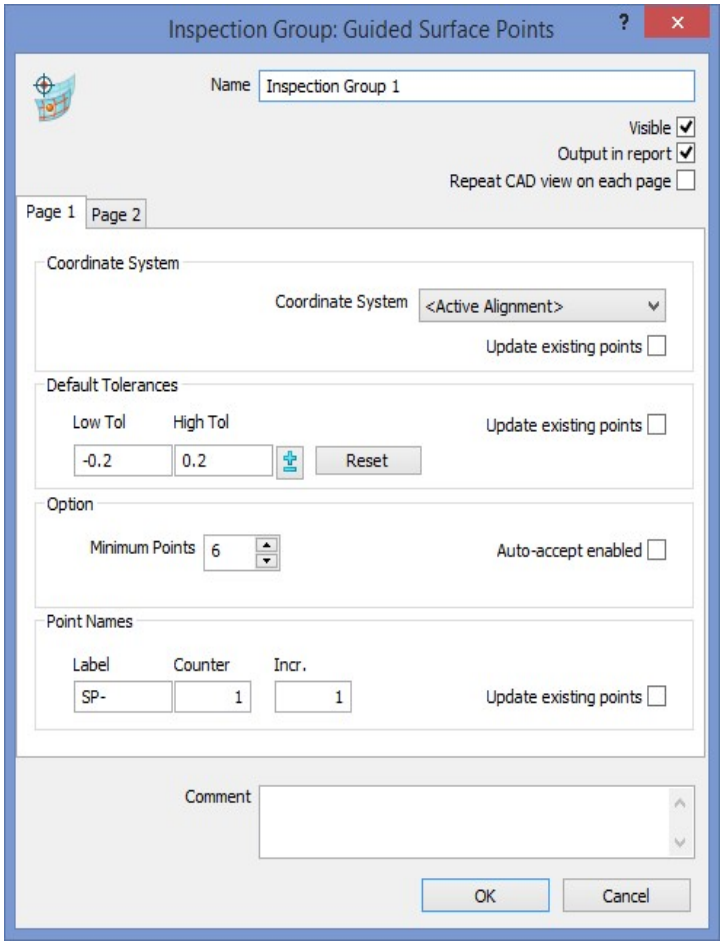
Manually Entering Points

You may need to control exactly where points are positioned on a part. This can be accomplished using the **Guided Point** dialog to manually input coordinates or edit existing coordinates. An advantage of this method is that you can ensure repeatability.

1 Ensure **Delay Measure**  is selected.

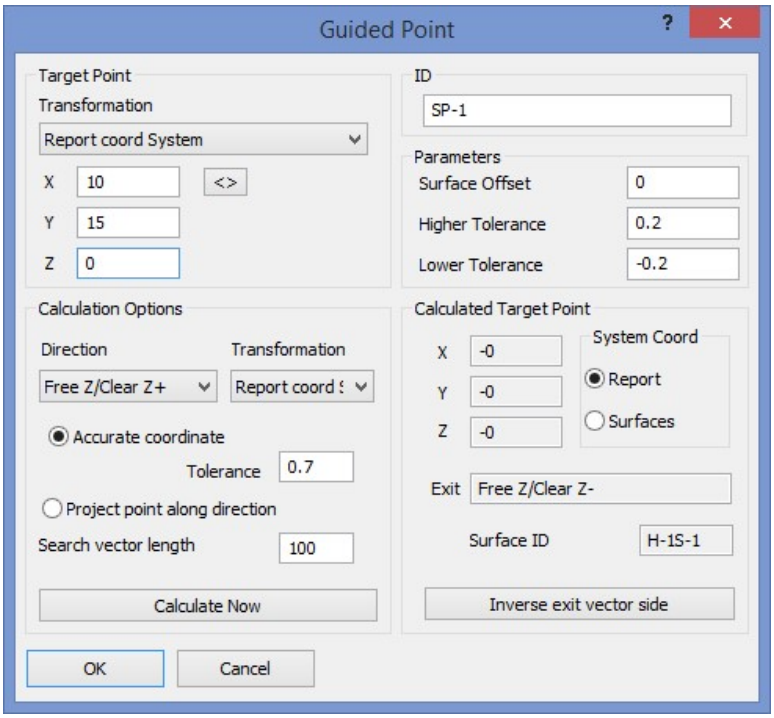
2 Click  to open the **Surface Inspection Group** dialog.

3 Select  **Guided Surface Points** Click **OK**.



4 Click  to open the empty **Surface Inspection** group.

5 Click  . The **Guided Point** dialog is displayed.

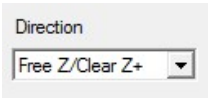


This dialog allows you to modify existing points or enter new coordinates to create points.

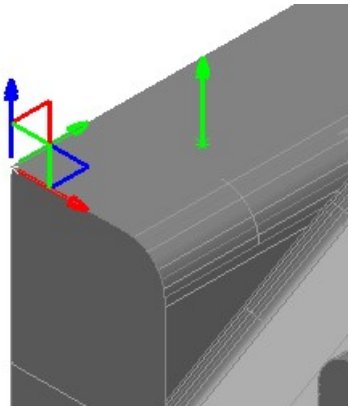
6 Enter the following co-ordinates in the **Target Point** area: **X 10, Y 15, Z 0.**

7 Select the **Accurate coordinate** option.

*The option **Project point along direction** can be used to project points (approximate values known) onto the surface in the direction defined.*

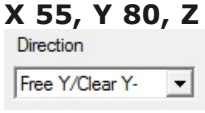


8 Click **Calculate Now** then click **OK**. The point is positioned on the surface at the specified coordinates.



9 Repeat the process two more times and enter the following coordinates :

- **X 55, Y 80, Z 0 Click Calculate Now & X 60, Y 0, Z -10** and select the direction



Click Calculate Now.

10 Select **Play All**  from the **Main** toolbar.

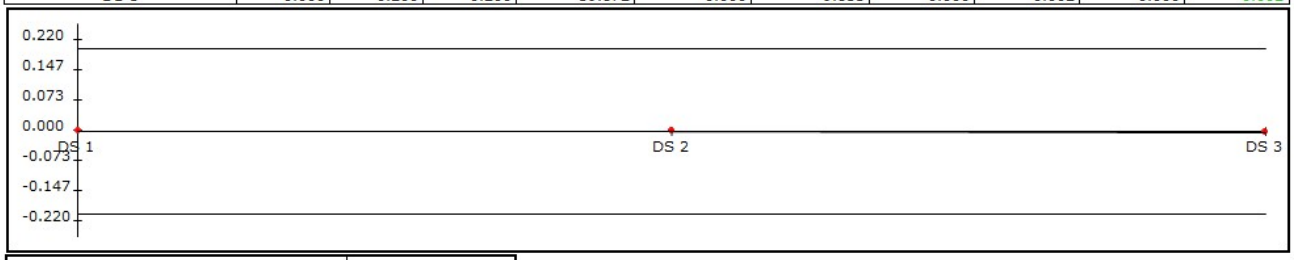
11 Probe the points in sequence order.

12 Check the measurement results in the **Report** tab.

Inspection Group 1

Datum - *Geometric PLP Alignment 1*

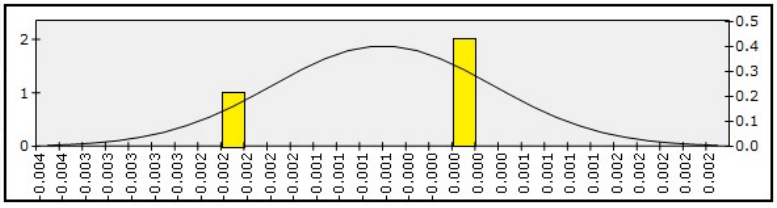
Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
DS 1	0.000	-0.200	0.200	9.708	15.366	0.000	-0.000	-0.000	0.000	0.000
DS 2	0.000	-0.200	0.200	54.868	80.350	0.000	0.000	0.000	0.000	0.000
DS 3	0.000	-0.200	0.200	59.872	0.000	-9.833	0.000	0.002	0.000	-0.002



Number of points	3
Out of tolerance	0
Performance	100%

Mean	-0.001
Std. Deviation	0.001

Max. Value	0.000
Min. Value	-0.002



Introduction to Section Inspections

You may want to perform an inspection of a cross-section across a part. PowerInspect allows you to create section lines, in any direction, along a surface. These sections can then be probed to inspect the deviation of that surface, in relation to the CAD data.

Preparing a session for section inspection

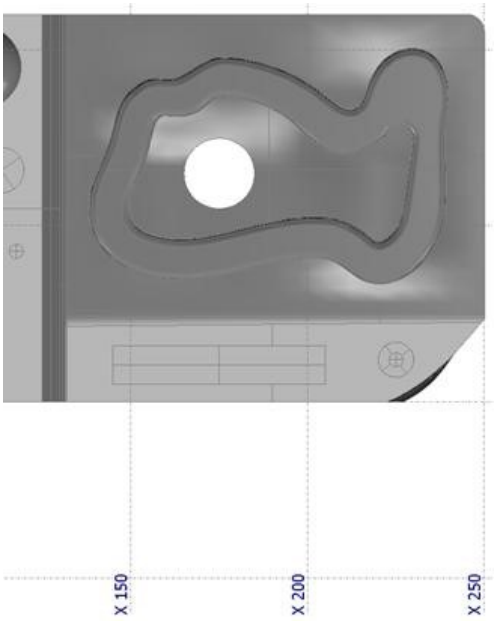
1 Select  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part**, and click **Next**.

2 Browse for **Demoblock2008(CMM+Arm).dgk** and click **Finish** to open the file.

3 Create an alignment using one of the previously learnt methods.

If you already have a session open, this can be used for the section inspection without the need to start a new session.

At this stage, with the CMM aligned to the CAD data, you would manipulate the view to choose where to take the section. However, for the purposes of this exercise, a section is going to be taken in the X plane (in the face of XZ) at a distance of **X=200**.



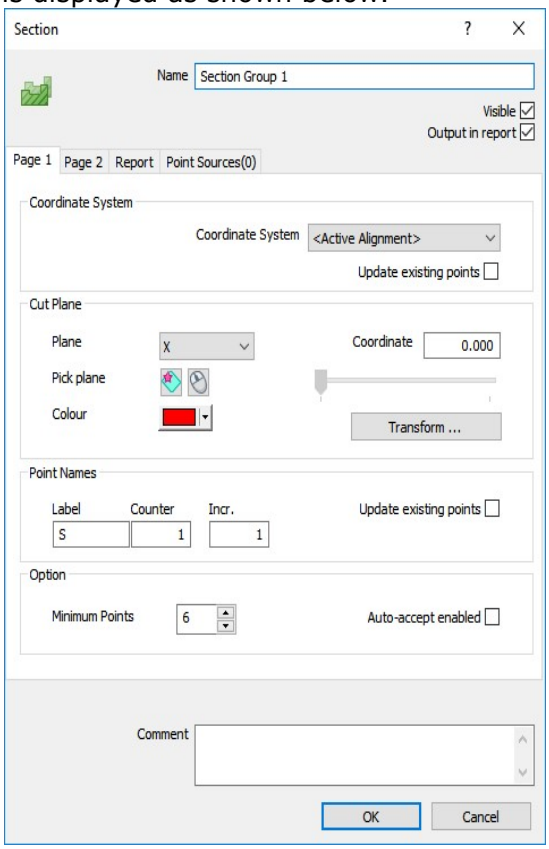
Generating a Section Inspection

To inspect a sectional area you must define the section to be probed.

1 Select **Delay Measure**  .

2 From the **Item** toolbar click **Section Group**  .

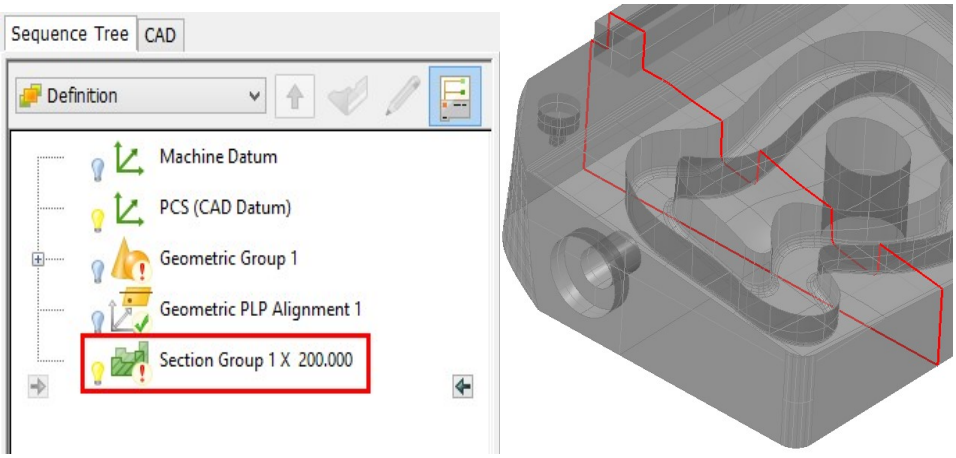
The **Section** dialog box is displayed as shown below.

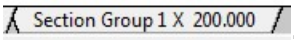


*If you click the **Transform** button, you can input a **Rotation** or **Translation** of the **Section Line**.*

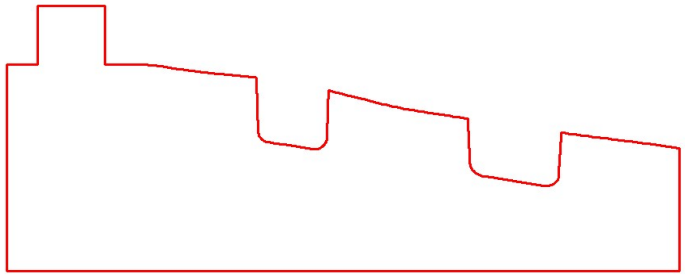
3 In the **Cut plane** area, set the **Plane** to **X**, and enter a **Coordinate** distance of **200**.

4 Leave the other settings unchanged, and click **OK**. The section line is displayed in the CAD View, and a section group is displayed in the Sequence Tree.



When a section group is created, a new **Section Group** tab appears below the graphics window 

. Select this tab to open the Section view.



*You can reorient the view by pressing the arrow keys on the keyboard (for mirror moves) and by pressing **Page Up** (anti-clockwise) and **Page Down** (clockwise) keys for 5-degree rotations.*

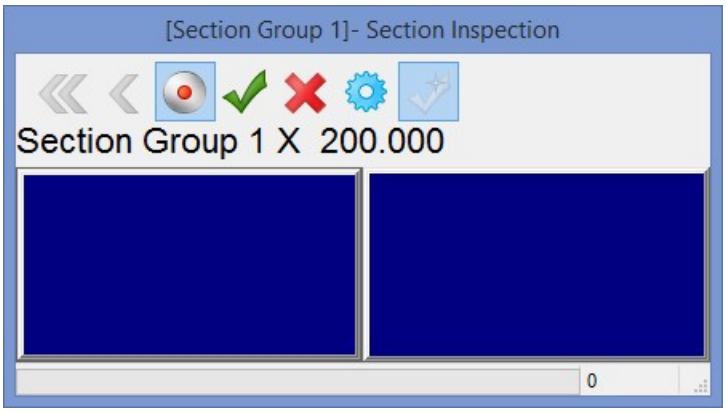
*Using the **Shift** key in conjunction with the **Page Up** and **Page Down** rotates the view by 90 degrees.*

With the part securely fixed to the CMM bed and aligned, the **Section Group** is now ready to be probed.

5 Orientate the view so that the section may be easily taken (for example, down the **Z Axis**).

6 Click the **Section Group** to highlight it in the Sequence Tree, and click **Play Item**  . PowerInspect enters full screen measure mode, and the **Section Inspection** dialog is displayed.
The **Section Inspection** dialog displays information about each point as you probe along the section line.

It is up to you to aim the probe along the section line and probe any number of points to measure that section. This can be done easily if your CMM has lockable axes, however if this is not available, then PowerInspect has some useful tools to help.

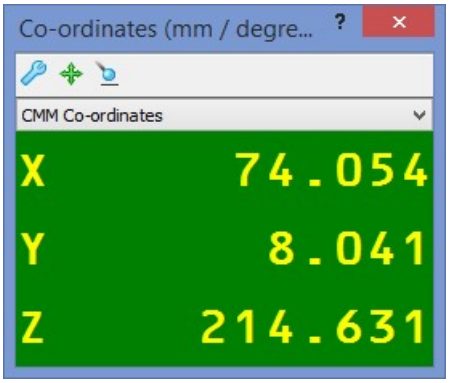


7 In the **Full Screen Options** toolbar on the right of the screen, click the **Tracking Box** 

button.

The **Tracking Box** is displayed, allowing you to see a numerical representation of the probe position. Because the part is aligned, the tracking can be set to the CAD Coordinates to monitor the **X=200** value.

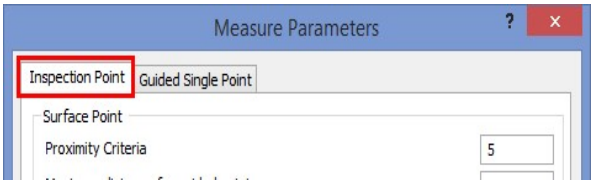
8 In the drop-down list of the **Tracking Box**, select **CAD Co-ordinates**.
You can now use the **Tracking Box** to check the probe position is as close to **X=200** as possible.



However, to keep the probe at **X=200** can prove very difficult if the axes cannot be locked, so you must specify a wider range (around **X=200**) in which to obtain probed positions.

9 In the **Section Inspection** dialog, click the **Measure Parameters**  button.

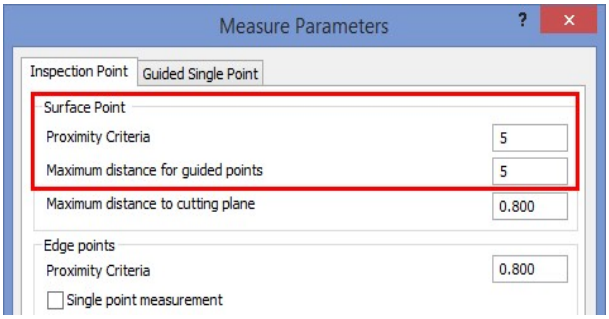
10 Select the **Inspection Point** tab at the top of the dialog.



*This dialog allows you to increase or decrease the search parameters for inspection points for both **Surfaces** and **Edges**.*

*Since this is a surface inspection, changes will only be made to the **Surface Points**.*

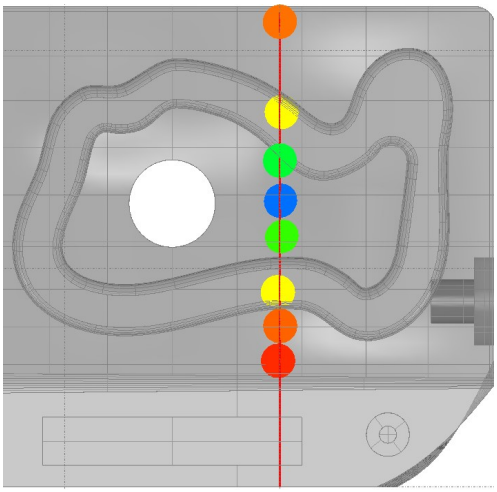
11 Change the **Proximity Criteria**, and the **Maximum distance for guided points** to the same value (in this example, the value is **5**).



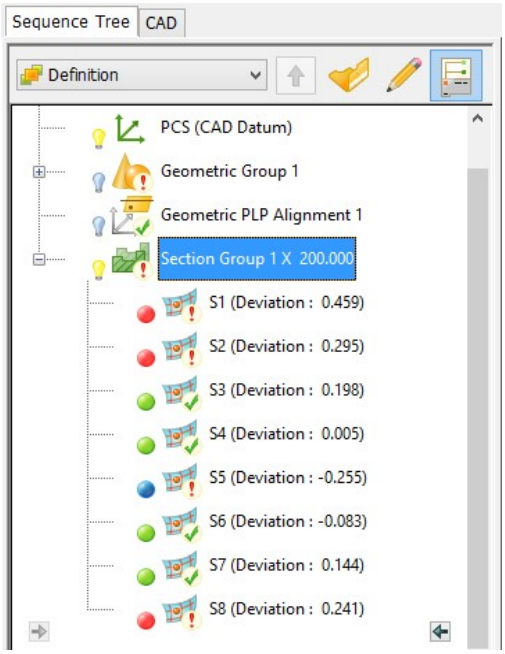
*PowerInspect will now allow points to be probed for the **X=200** line at a minimum of **X=195** and a maximum of **X=205.***

12 Probe along the **X=200** line as accurately as possible.

13 When you have probed all the required points, click .

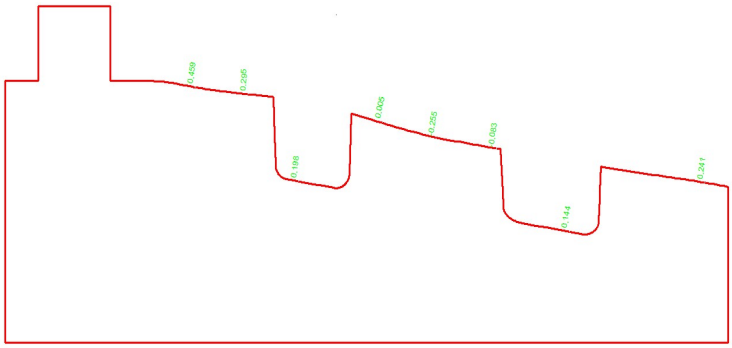


When the section group is opened, the points are displayed in the Sequence Tree each with their corresponding deviation as shown below.

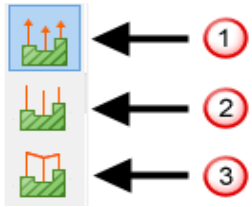


Section View

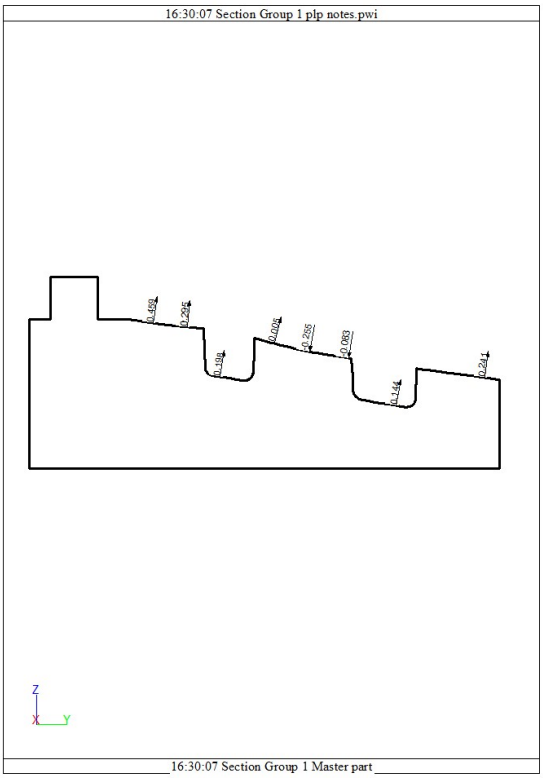
1 Select the **Section Group** tab to open the **Section View**.



The above **Section View** is shown with the deviations as vectors. You can choose whether to display the results as **Vectors** ①, **Pins** ② or **Linked Pins** ③ using the **Section View** toolbar.



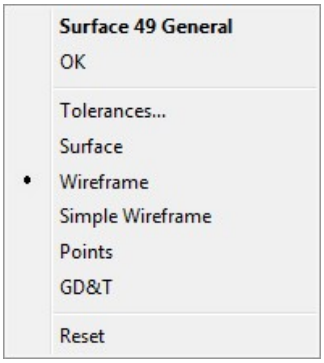
2 Click **Print Preview**  to view how the printed output would appear.



Geometric Feature Inspection (using Wireframe Checker)

The **Wireframe Checker**  button allows you to pick geometric features directly from a model in the CAD view. The details of the feature you select are automatically shown in the **Inspect** dialog for that feature. The **Wireframe Checker** works in conjunction with the **Geometry Explorer** tab.

The **Wireframe Checker** has the following picking modes.



To change the picking mode, right-click an empty area of the graphics window and select a picking mode from the popup menu. Select.

- **Surface** to pick a whole CAD surface. If the surface is part of a complex 3D-wireframe entity, PowerInspect extracts the entity's details.
- **Wireframe** to pick a complex 2D-wireframe-entity, such as a slot. PowerInspect extracts the arc at each end and the line on each side and displays their details in the **Geometry Explorer** tab. You can also use this mode to select arcs (circles).
- **Simple Wireframe** to pick a single component of a complex 2D-wireframe entity, such as the arc at the end of a slot. You can also use this mode to select arcs (circles), lines and points.
- **Points** to pick single points on the CAD wireframe. If you pick two points, PowerInspect constructs a line. If you pick three points, PowerInspect constructs an arc. The point is very useful when the CAD has been drawn using trimmed entities which can be difficult to select. For example a slot can be made up of two circles and two tangent lines that are trimmed.
- **GD&T** (Geometric Dimensioning and Tolerancing) to pick GD&T items from the CAD model.

GD&T mode applies only to models containing GD&T information, such as those saved in CATIA and Unigraphics files.

*When an item is selected from the CAD model, the **Nominal data** is displayed on the **Geometry Explorer** tab. When you click*  *to accept an item, PowerInspect creates a new item in the inspection sequence.*

Feature Inspection Example

1 Select  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part**, and click **Next**.

2 Browse for **Demoblock2008(CMM+Arm).dgk** and click **Finish** to open the file.

3 Create an alignment using one of the previously learnt methods.

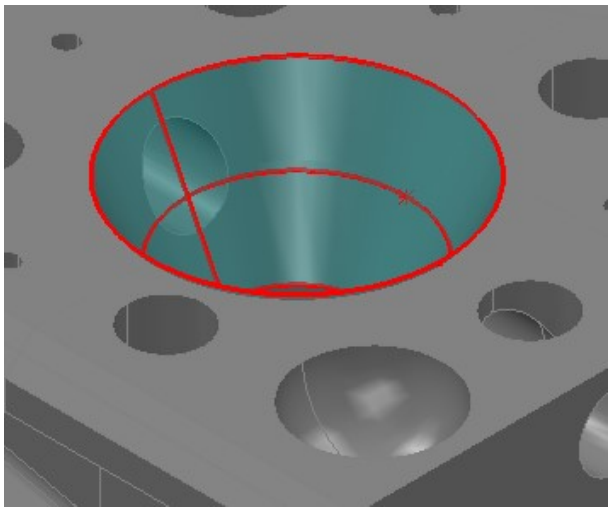
4 Click **Delay Measure**  .

5 Click the **Activate sequence items**  button from the **Geometry Explorer** tab.

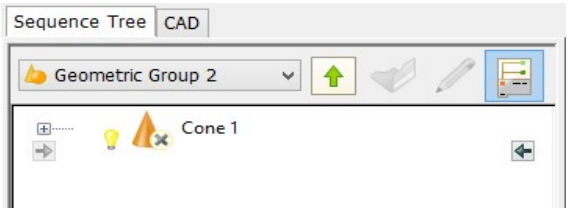
6 Click **Wireframe Checker**  .

7 Right-click an empty space in the CAD view and select **Surface** from the local menu.

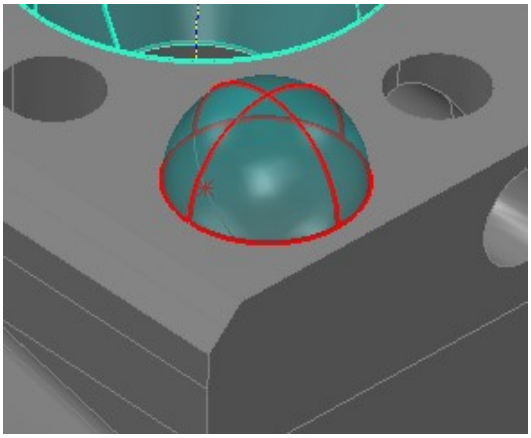
8 Select the central cone feature on the CAD model, and click  in the **Geometry Explorer** tab.



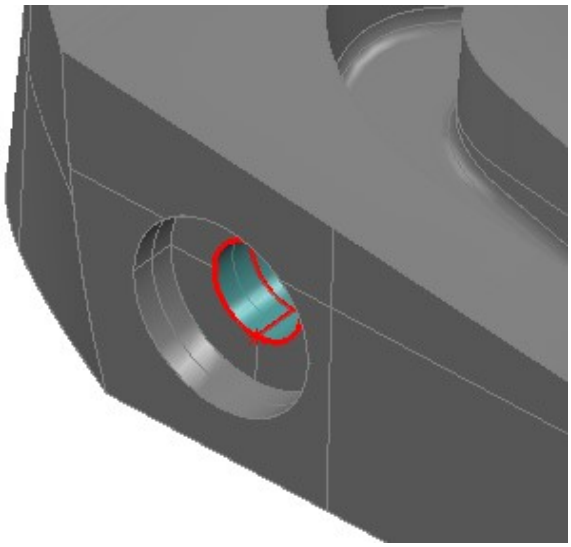
The feature has been added to the Sequence Tree within a new **Geometric** group.



9 Select the **Sphere** feature on the CAD model, and then select  on the **Geometry Explorer** tab.

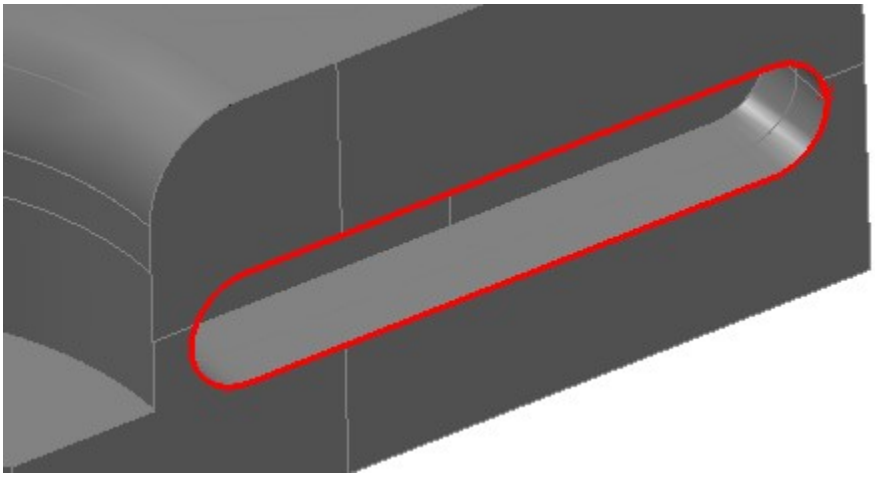


10 Select the **Cylinder** feature on the CAD model and then select  on the **Geometry Explorer** tab.

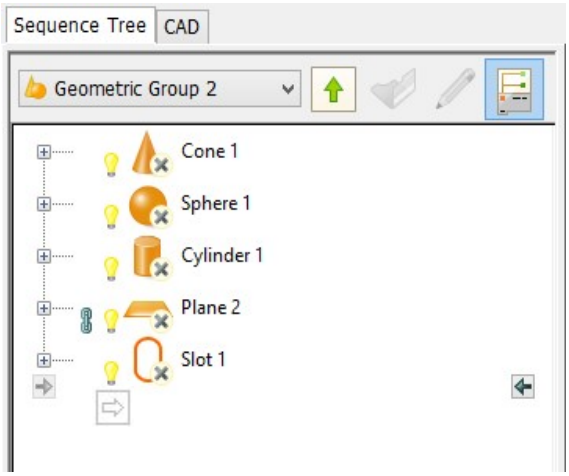


11 Right-click an empty space in the CAD view and change the picking mode to **Wireframe**.

12 Select the **Slot feature**. In the **Linked to** list of the **Geometry Explorer** tab, select **New Probed Plane** and then select  .



The Sequence Tree lists all items you selected.



13 In the **Main** toolbar, click **Play All**  .

14 Select the **Report** tab to check the results on the **Report** page.

Geometric Group 2

Cone 1 (Datum - *Geometric PLP Alignment 2*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Half Angle	0.100	-0.100	26.565	26.263	-0.302	-0.202

Sphere 1 (Datum - *Geometric PLP Alignment 2*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Diameter		0.100	-0.100	24.000	24.127	0.127	0.027
Centre	X	0.100	-0.100	107.000	106.962	-0.038	-
	Y	0.100	-0.100	95.000	94.901	-0.099	-
	Z	0.100	-0.100	0.000	-0.005	-0.005	-

Cylinder 1 (Datum - *Geometric PLP Alignment 2*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Diameter	0.100	-0.100	10.000	10.051	0.051	-

Slot 1 (Datum - *Geometric PLP Alignment 2*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Centre	X	0.100	-0.100	0.000	0.101	0.101	0.001
	Y	0.100	-0.100	37.500	37.603	0.103	0.003
	Z	0.100	-0.100	-25.000	-24.979	0.021	-
Length		0.100	-0.100	67.000	67.201	0.201	0.101
Width		0.100	-0.100	12.000	12.100	0.100	-

Geometric feature display

In *5.9 Displaying data in the CAD view,* various ways in which measurements can be displayed in the CAD view are shown. The measurement data of geometric features can also be displayed in a number of ways.

The **Geometry Feature Display** toolbar  is available from the **View Options** toolbar. This provides a tolerance-based visual representation of the points taken to create the geometry.

The following options are available.

 — **Deviation Lines**

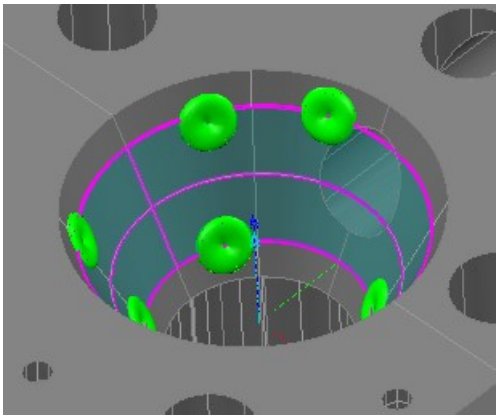
 **— Connected deviation lines**

 **— Vector lines**

 **— Discs**

1 Select the **CAD View** tab and zoom into the cone feature.

2 Click **Discs**  on the **Geometry Feature Display** flyout.

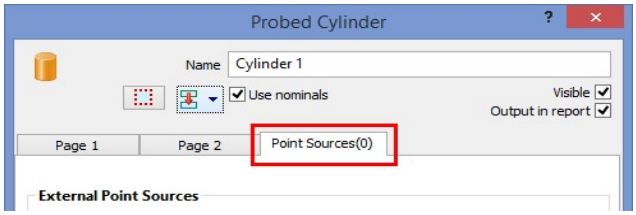


The probe points used to calculate the cone feature are displayed as colour-coded discs (green in this case, because all points are within tolerance).

3 Select the remaining three options to view the results.

Combining measurements (Point sources)

The **Point Sources** tab is available in the definition dialog of geometric items, inspection groups and section groups. It enables you to create a measured item using probed points from other items in the inspection sequence. For example, you can use this feature to create a cone by combining the probed points of circle items that measure different cross-sections of the feature.

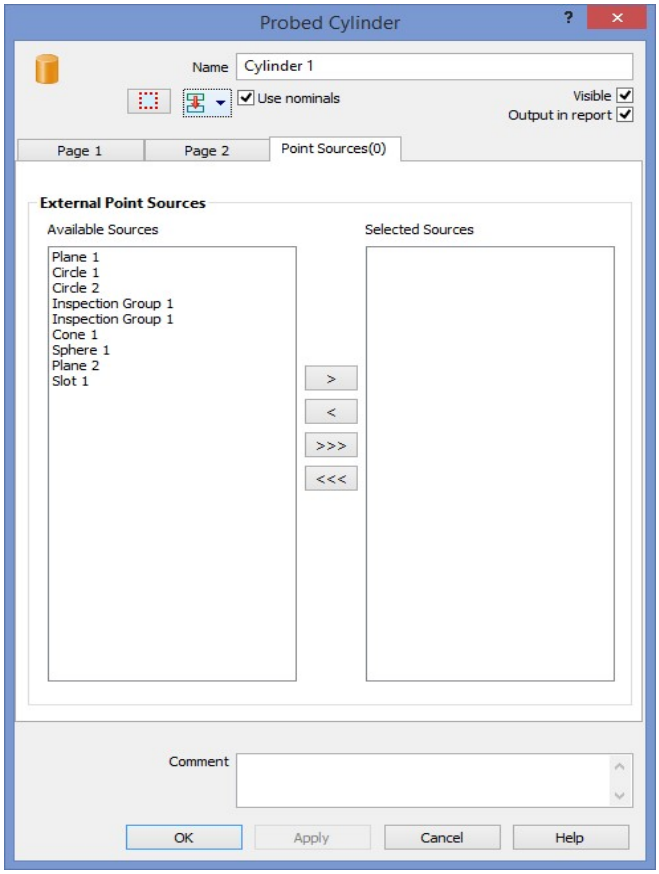


To create a compound item.

1 Open the definition dialog of an *unmeasured* item.

2 Select the **Point Sources** tab.

3 In the **Available sources** list, select the items to be used to supply the probed points for this item, then click > to add them to the **Selected sources** list.



4 Select **OK** to save your changes and close the dialog.
The below icon is displayed on the item's entry in the inspection sequence to indicate that it uses measurements from other items.

Simple Measures

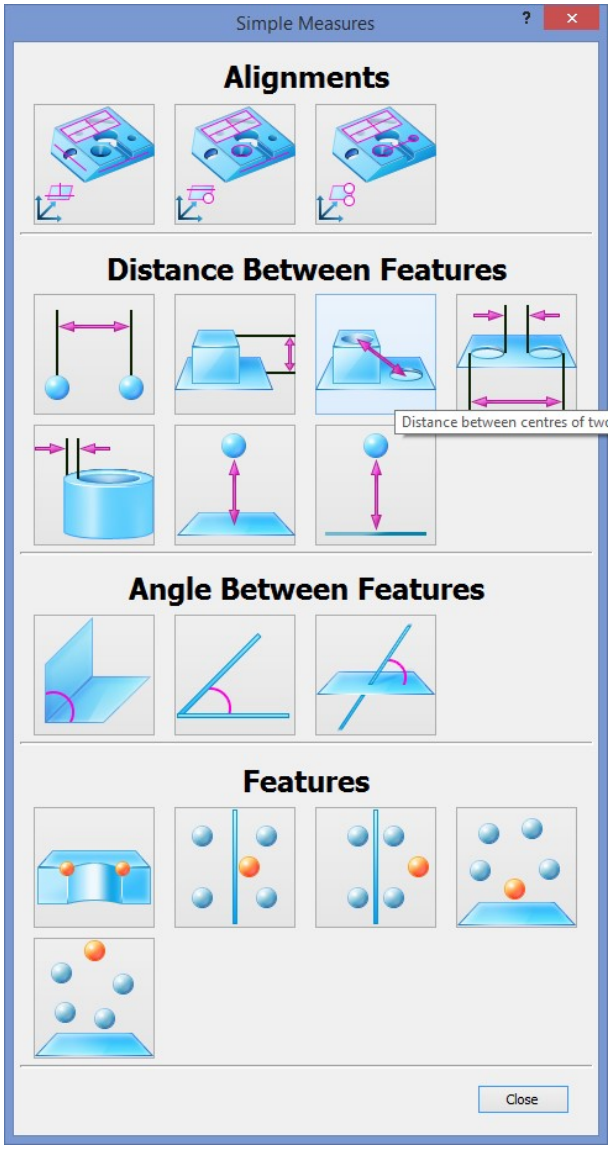
Simple Measures provides a quick and easy method of creating Geometric PLP alignments, and of measuring dimensions between features. With simple measures, PowerInspect automatically creates and plays all the items you need to make the measurement, so you can access the measurement immediately.

To create a simple measure.

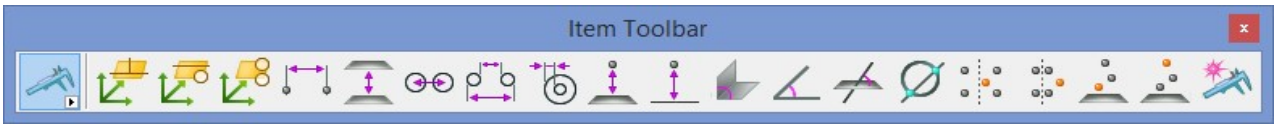
1 Click the **Simple Measures** button on the **Item** toolbar.

An alignment or CAD model is not necessary for Simple Measures.

When **Delay Measure** is deselected, clicking **Simple Measures** connects to the measuring device and displays the **Simple Measures Wizard**. This allows you to quickly select the type of measurement required, and probe it immediately.



When **Delay Measure**  is selected, clicking the **Simple Measures**  button displays the **Simple Measures** toolbar. The icons represent the same functions as available from the wizard.
You can then create the same simple measure items by clicking the corresponding buttons in the toolbar.



2 Select a measurement type. The following measurements are available:

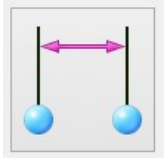
- **Alignments**

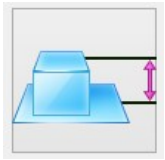
 — Creates a Geometric PLP Alignment defined by Plane, Line, and Perpendicular Line.

 — Creates a Geometric PLP Alignment defined by Plane, Line, and Circle.

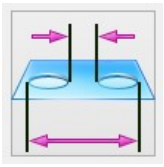
 — Creates a Geometric PLP Alignment defined by Plane, and two Circles.

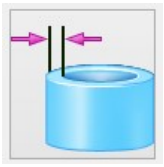
- **Distance Between Features**

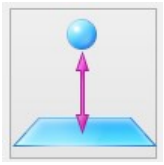
 — Measures the distance between two points.

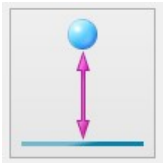
 — Measures the distance between two planes.

 — Measures the distance between the centres of two circles.

 — Measures the shortest and longest distances between two separate circles.

 — Measures the minimum and maximum wall thickness between two nested circles.

 — Measures the distance between a point and a plane.

 — Measures the distance between a point and a line.

- **Angle Between Features**

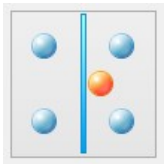
 — Measures the angle between two planes.

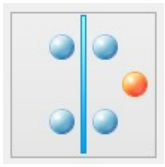
 — Measures the angle between two lines.

 — Measures the angle between a plane and a line.

- **Features**

 — Measures the points where a line intersects a circle.

 — Compares group of probed points to the axis of a circle, and reports location and distance of the point nearest the axis.

 — Compares group of probed points to the axis of a circle, and reports location and distance of the point furthest from the axis.

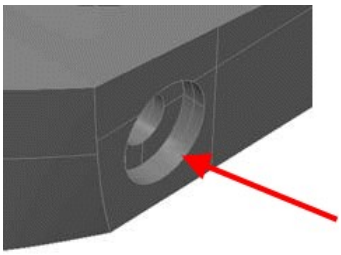
 — Compares group of probed points to a plane, and reports location and distance of the point nearest the plane.

 — Compares group of probed points to a plane, and reports location and distance of the point furthest from the plane.

Min/Max wall example

1 Deselect **Delay Measure**  .

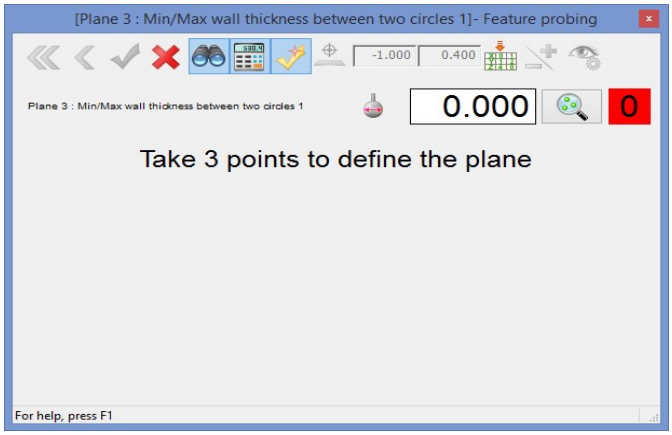
2 Zoom into the hole feature.



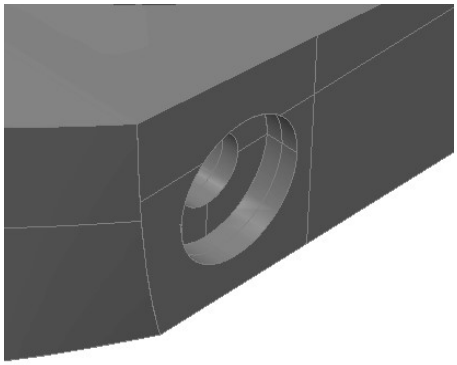
These two cylinders will be used to illustrate a wall thickness measurement.

3 From the **Item** toolbar, click **Simple Measures**  . The Simple Measures wizard is displayed.

4 From the **Distance Between Features** section, select **Min/Max wall thickness between two circles**  . Full screen measure mode is activated and the **Feature Probing** dialog is displayed.



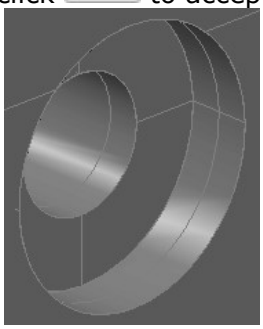
5 Probe the plane and then click  to accept the points.



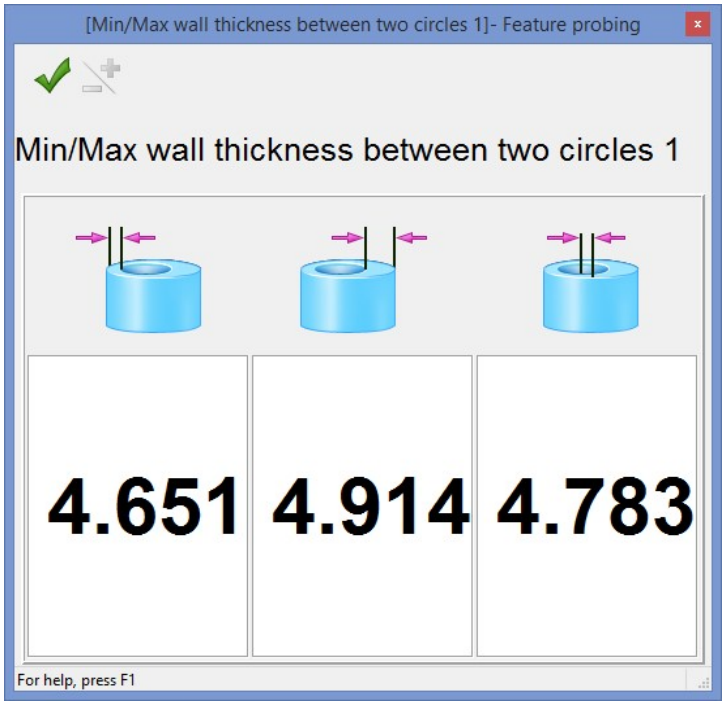
*If the **Auto accept** button is selected, PowerInspect automatically moves to the next item in the inspection sequence.*

6 Probe the outer cylinder and then click to accept the points.

7 Probe the inner cylinder and then click to accept the points.



The results are displayed on the screen.



8 Click the **Accept** button to reopen the Wizard, allowing you to take further measurements.

9 Click **Close** to close the Wizard.

10 Select the **Report** tab to view the results on the **Report** page.

Simple Measures Group 1

Min/Max wall thickness between two circles 1 (References: ***Circle 5 : Min/Max wall thickness between two circles 1, Circle 6 : Min/Max wall thickness between two circles 1***)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Minimum Thickness	-----	-----	-----	-12.252	-----	-----
Maximum Thickness	-----	-----	-----	-12.066	-----	-----
Average Thickness	-----	-----	-----	-12.159	-----	-----

Auto-extract, Auto-detect and Auto-accept

PowerInspect includes options designed to help streamline the probing process.

- **Auto-extract nominals from CAD —** PowerInspect automatically loads the nominal data of features as they are probed.
- **Auto-detect type for extraction** — PowerInspect automatically detects the type of feature being probed when using Geometric Multi-Measure. This can be used in conjunction with **Auto-extract nominals from CAD.**
- **Auto-accept** — PowerInspect automatically accepts items after a predefined number of points have been probed.

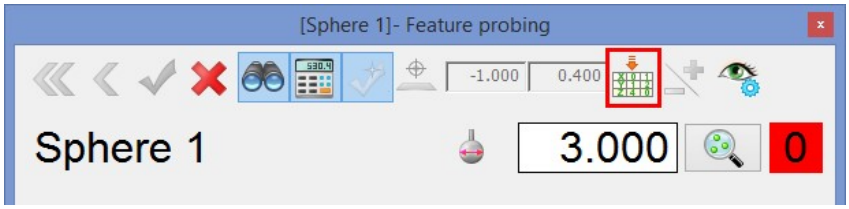
All of these options require a CAD model and a valid alignment.

Auto-extracting nominals

An **Auto-extract nominals from CAD** button is available in the **Feature Probing** and **Geometric Multi-measure** dialogs. If activated, the nominal data is extracted and loaded from the feature. This provides an instant comparison (deviation and error) in the reported results.

1 Select an unmeasured item in the inspection sequence, and click the **Play Item** button in the **Main** toolbar. The **Feature Probing** dialog is displayed.

2 Select **Auto-extract nominals from CAD** .



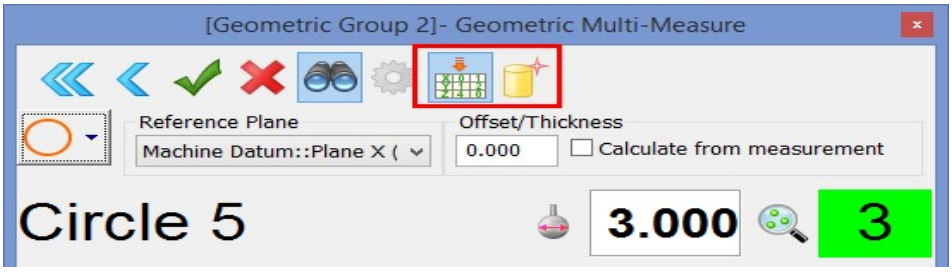
3 Probe the item, and click to save.

Auto-detecting features with Geometric Multi-Measure

The **Auto-detect type for extraction** button in the **Geometric Multi-Measure** dialog automatically identifies the features you are probing by comparing probed points to the CAD. Use it as a quick alternative to switching the item type each time you want to probe a different type of feature.

1 Open a geometric group, and click the **Geometric Multi-Measure** button in the **Item** toolbar. The CAD view switches to full screen measure mode and the **Geometric Multi-Measure** dialog is displayed.
If you want to display the coordinates and deviation of the points as you probe them, click the **Point Details** button.

2 Select the **Auto-extract nominals from CAD** button and then select the **Auto-detect type for extraction** button.

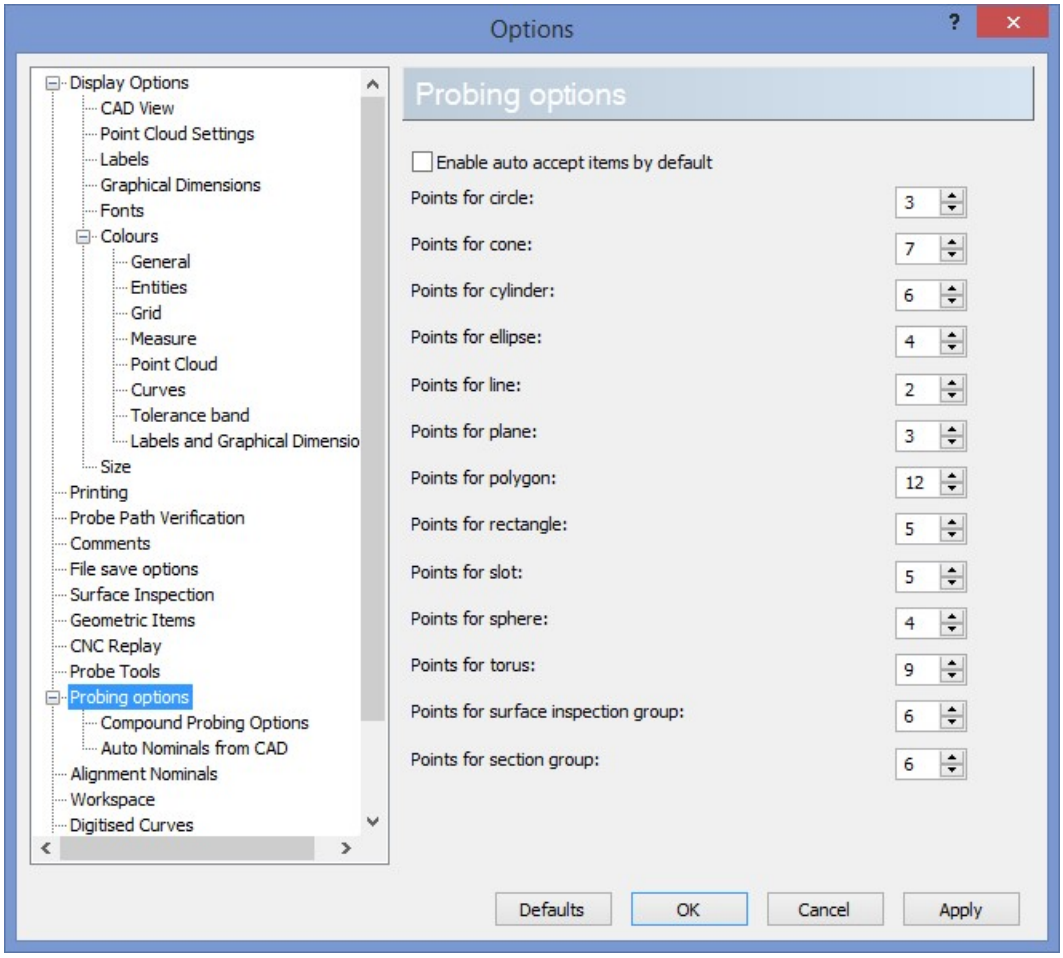


3 When you have probed all the points for the feature, click to save the points, and add the item to the inspection sequence or move to the next feature.

Auto-accepting items

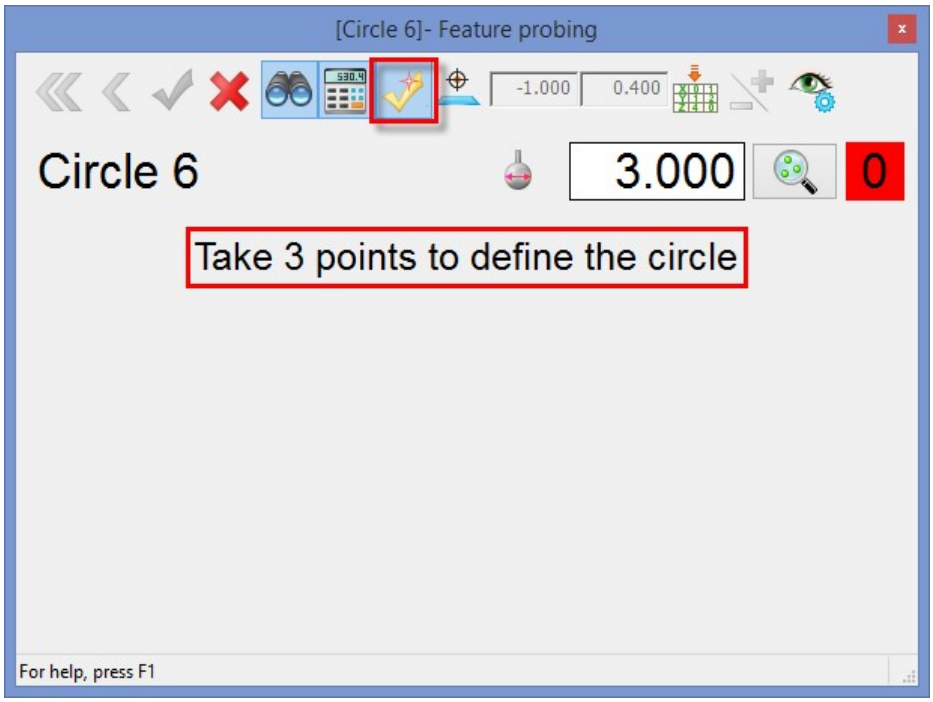
When the **auto-accept**  button on the **Feature probing** dialog is selected, PowerInspect saves your measurements as soon as you have probed the number of points specified on the probing dialog.

You can modify the number of points required for different items on the **Tools > Options > Probing options** page.



4 Select an unmeasured item in an inspection sequence and click **Play Item**  to display the **Feature probing** dialog.

5 Click **Auto-Accept**  . The number of points that you need to measure is displayed in the dialog.



6 Play the item. When you have probed the required number of points, PowerInspect saves the measurement and closes the dialog.

If you are playing more than one item, PowerInspect automatically plays the next item in the sequence.

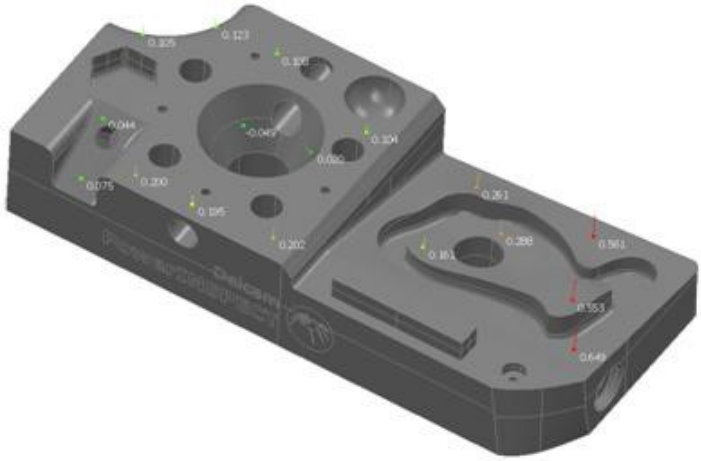
Displaying data in the CAD view

Geometric features and probed points can be displayed in the CAD view in several ways using the View Options toolbar.

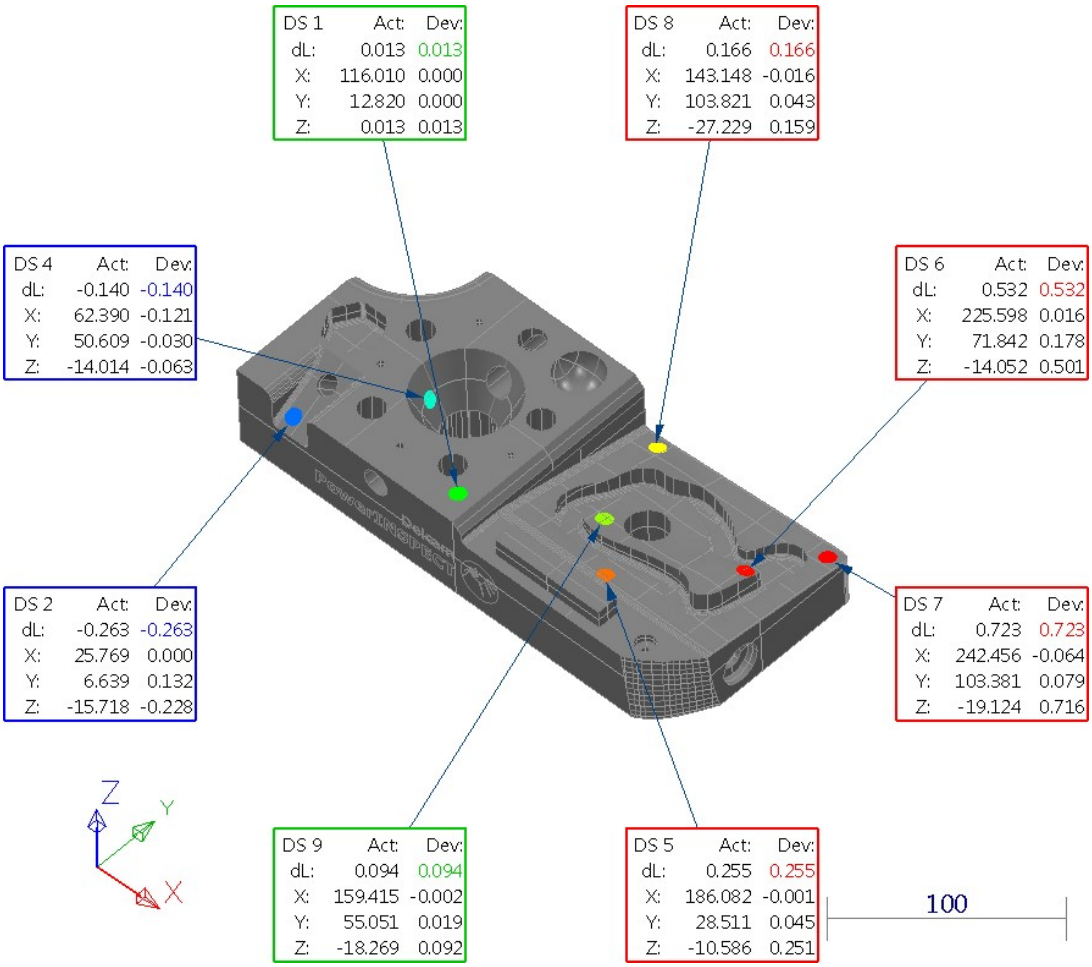
The **Confetti**  button displays the probed points as coloured spots. Points shown in green are within tolerance; points shown in red are above tolerance; and points shown in blue are below tolerance.

The **In Place** 

button displays the points as pins, with the length of each pin representing the amount of deviation. The pin colour also indicates whether the measurement is within, above, or below tolerance.



The **Labels**  buttons display measurement details of the geometric features and probed points. Labels can be displayed in conjunction with **Confetti** or **In Place** options as shown on the example below.

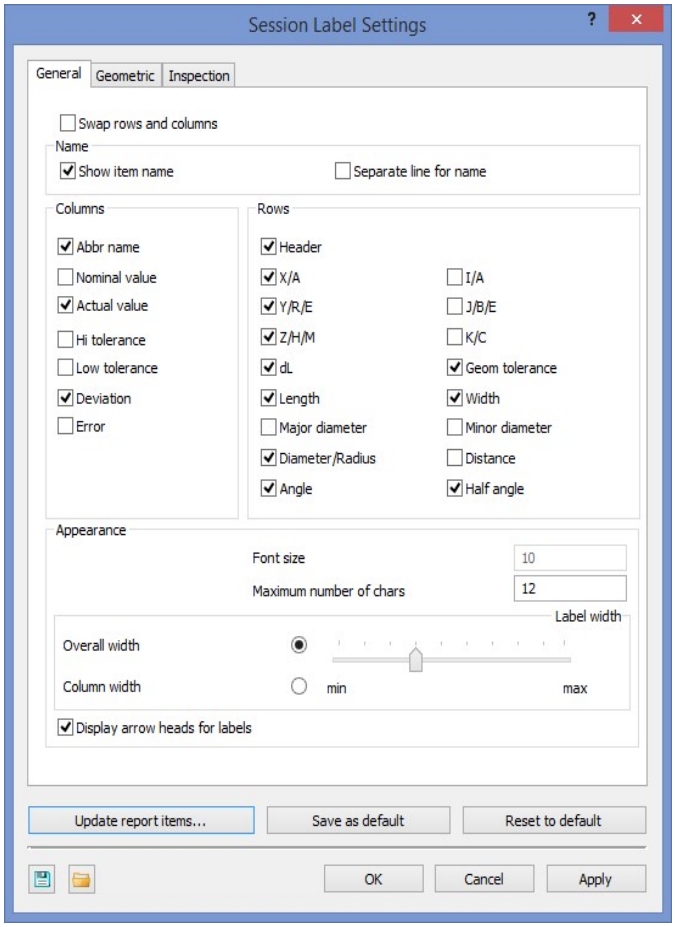


- Click  to display all the labels in a horizontal format.

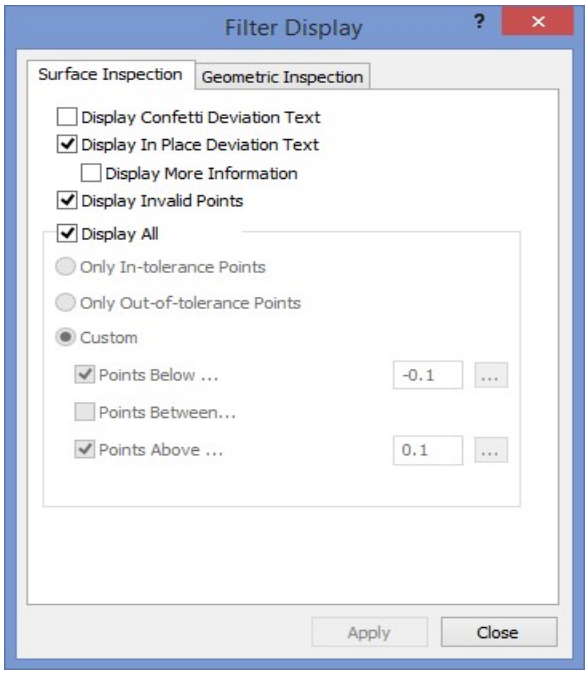
- Click  to display the labels in standard format (horizontal and vertical).

- Click  to enable you to customise the position of labels. Individual labels can be moved by holding the left mouse button down on a label while dragging it within the CAD view.

Click the **Global Label Settings**  button to open the **Global Label Settings** dialog and choose which information is displayed on the labels. You can also use this dialog to decide the width of the labels.



Further filter display options are available by selecting the **Filter Display**  button. It allows you to filter out the displayed points for surface and geometric inspections. For example, to display only out-of-tolerance points in the CAD view, deselect **Display all**, and select **Only Out-of-tolerance Points**.



The **Custom** option allows you to set your own filter criteria.

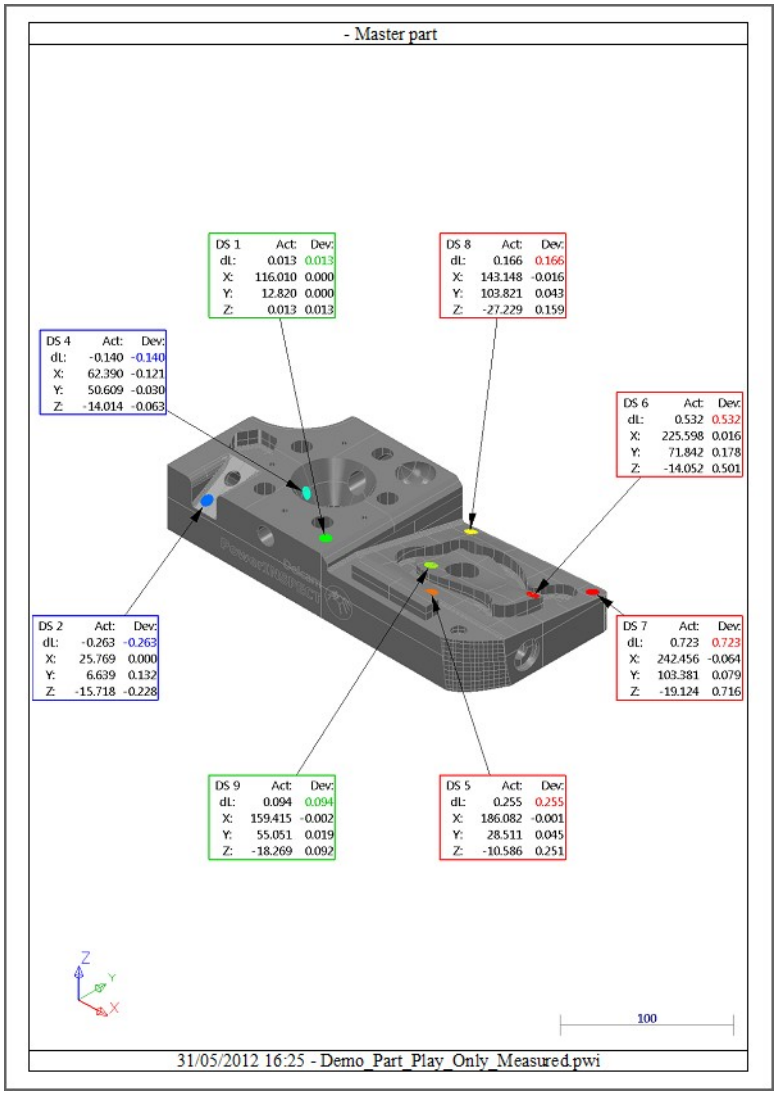
Print Previews

After you have performed a surface inspection, you can print the data.

1 Click the **ISO 2**  button to position the CAD model.

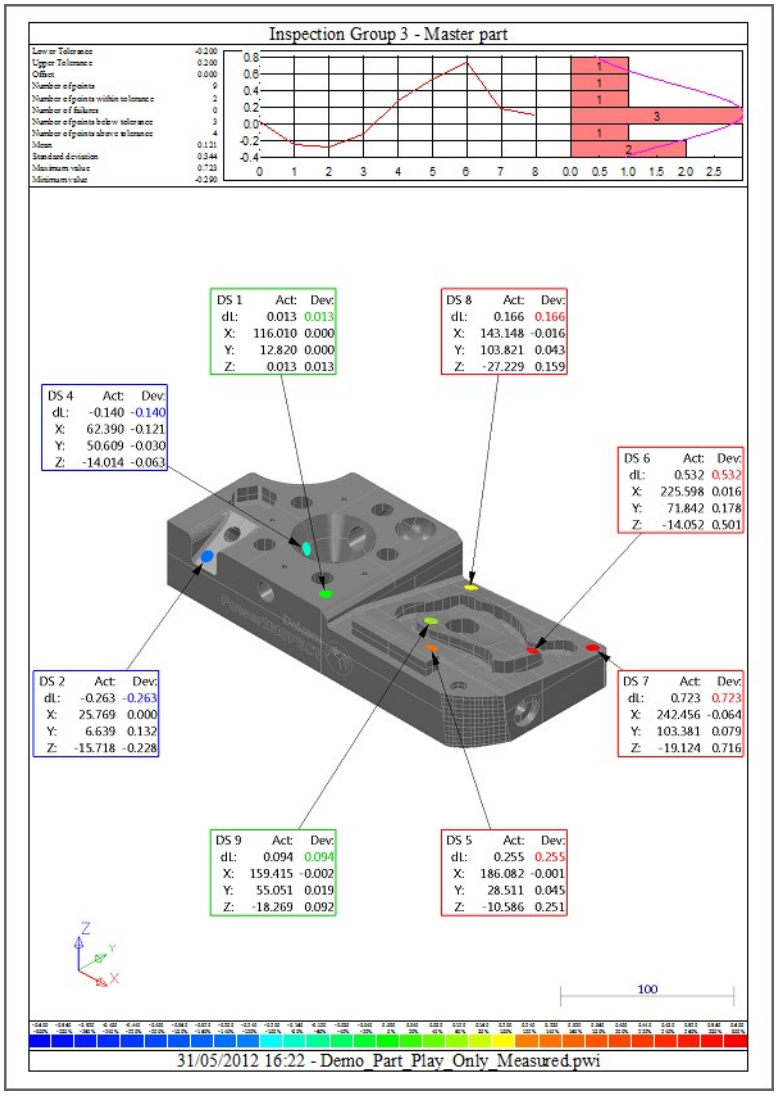
2 Click **Print Preview**  . A preview of the CAD File, along with the probed points, is displayed on the screen.

If you were to print the page, this would be the output.



If the **Print Preview**  button is selected while a geometric group is open, then a graph of deviation and a coloured bar is included in the preview.

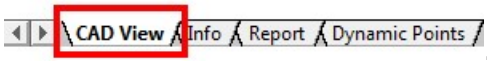
*Ensure **Draw the coloured scale with confetti** is also selected from **Tools > Options > Display Options**.*



CAD view report

PowerInspect lets you include CAD images in the final report. This feature is called CAD View Report.

To create a CAD view report.

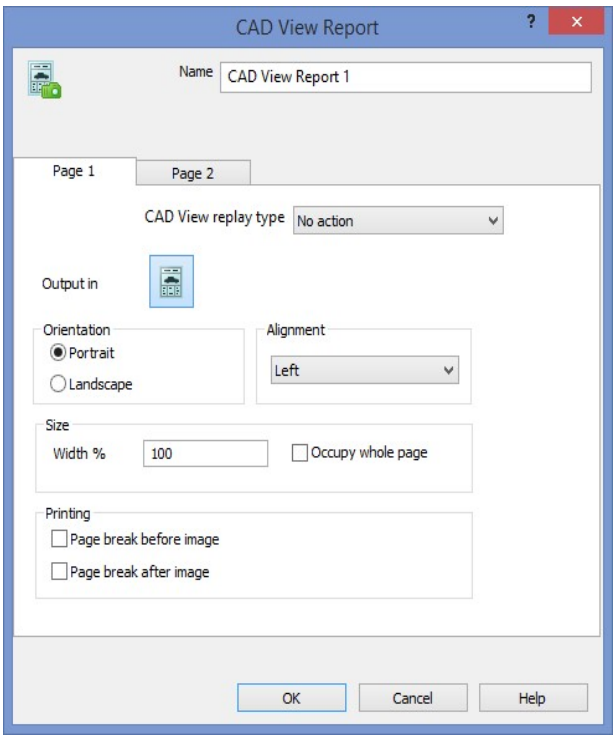
1 Select the **CAD View** tab 

.

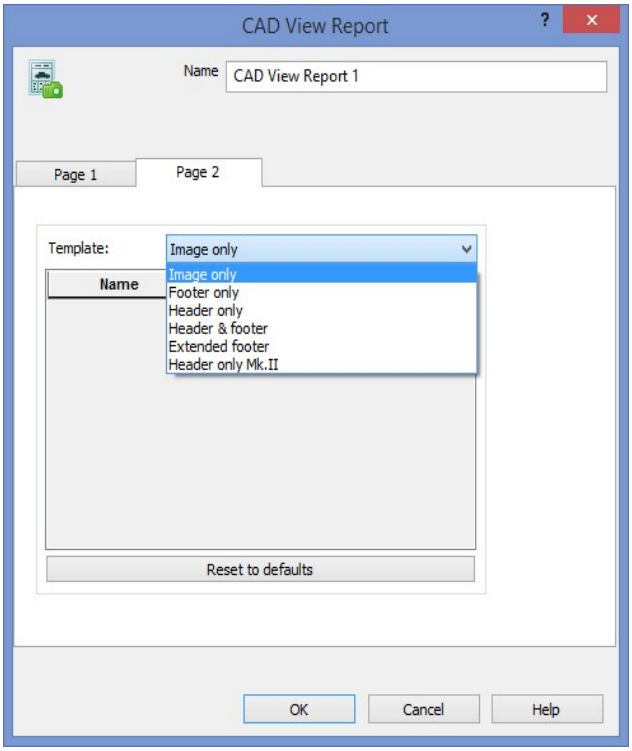
2 Click  to select your preferred label style, and orientate the model to a suitable view.

3 Click on the **Miscellaneous**  toolbar and select **CAD View Report**  .

Page 1 of the dialog allows you to change the **Orientation**, **Alignment**, **Size** and **CAD View replay** type. This enables you to choose how the CAD view behaves when a **CAD view report** item is played.



Page 2 of the dialog allows you to select a template which displays information in the report image itself.



4 Leave the settings unchanged, and click OK to insert the CAD View Report item into the Sequence Tree.

5 Select the Report tab to view the final results and image in the report.

6 If you delete the **CAD View Report** item from the Sequence Tree, the image is also deleted from the report.

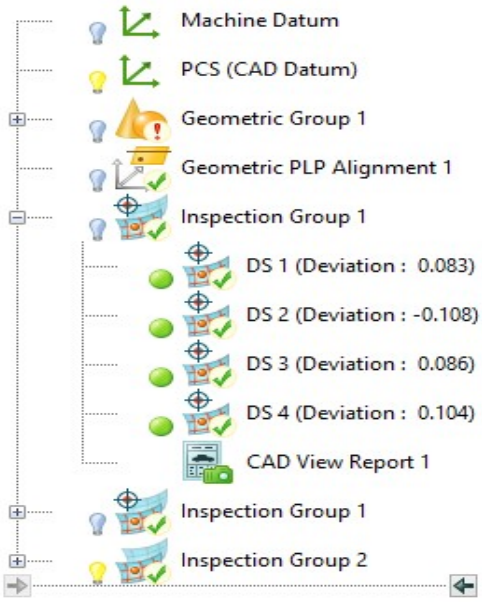
Inserting a CAD View Report within an inspection sequence

CAD View Reports can also be inserted within inspection and geometric groups.

1 Select **Inspection group 1** and open  it. The **Item** toolbar is replaced with the **Inspection** toolbar.

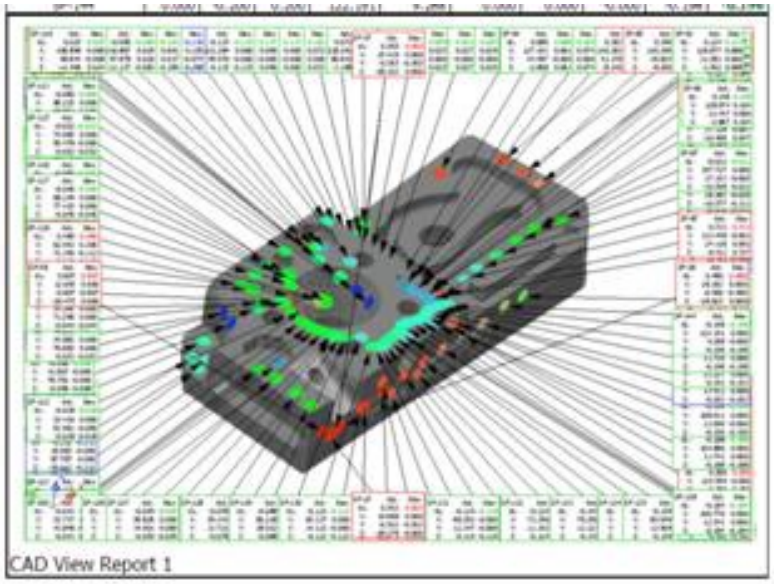
2 Click  to display labels.

3 Click  to insert a **CAD View Report** item into the inspection sequence.



4 View the results and image in the **Report** tab. The image is displayed just below the Inspection Group 1 table.

Reports with attached images can be difficult to interpret if a high volume of points are taken, as shown.

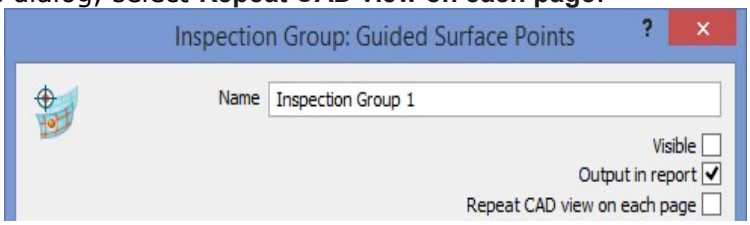


PowerInspect has the ability to split the report automatically and display the corresponding CAD view image only for those points on each page.

5 Click  to close **Inspection Group 1**.

6 Click  to modify **Inspection Group 1**.

7 At the top of the dialog, select **Repeat CAD view on each page**.



8 Click **OK**.

9 Check the results in the report. A CAD view is created at the top of each page with associated labels and table displayed. This produces a report that is easier to read and present.

6. Measurements without CAD

PowerInspect does not have to work with CAD Data. Parts can be inspected for dimensional measurement and comparison without reference to a CAD model. Measurements are created using geometric elements, which are either probed, or referenced to probed elements. An alignment is required in order to have the data in the components coordinates rather than machine coordinates.

Preparing a Session for Measurement Inspection

For the purposes of this exercise, no CAD data will be used.

Select **New Document**  on the **Main** toolbar. A new empty session is loaded.

The new session is now ready for inspection.

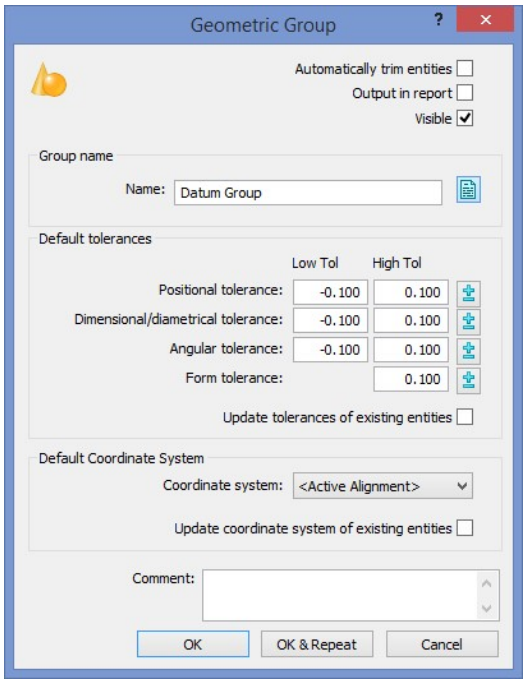
Generating a Measurement Inspection

The **DemoBlock2008(CMM+Arm)** will be used for measurement. Align the block so that the side with the logo is facing towards the front of the CMM/Table.

Initially a datum will be created at the lower rear left corner of the block. This will allow measurements to be taken relative to this point and aligned to the axes of the datum. First a geometric group will be created and a Geometric PLP alignment generated from the elements probed.

1 Select the **Delay Measure**  button.

2 From the **Item Toolbar**, click the **Geometric Group**  button.
The **Geometric Group** dialog is displayed and prompts you to name the group and add a comment if required.



3 Name the group **Datum Group.**

4 Deselect **Output in report**.

5 Leave the other settings unchanged and click **OK**.

Next, a series of geometric items need to be created for measurement from the **Item** toolbar. In this example, six items are created. These include three **Probed Planes** defining the **X**, **Y** and **Z** planes, two **Plane Intersection Lines** for the **X** and **Y** axes and a **Line Intersection Point** for the origin.

6 Click the **Planes**  button to open the flyout.

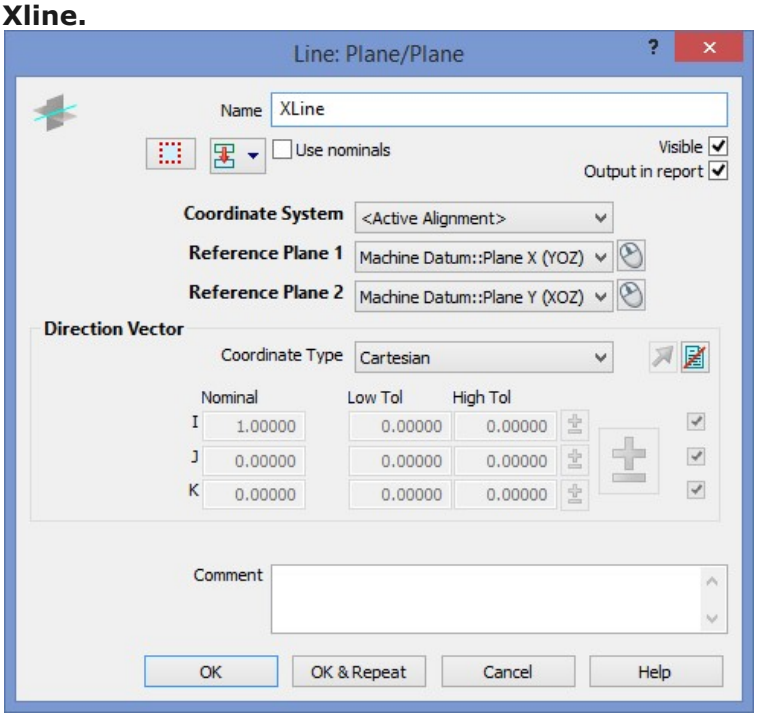
7 Click the **Probed Plane**  button**.**

8 Click **OK & Repeat** twice, and then click **OK** to create three planes.

- **Plane 1** is probed from the CMM bed and defines the Z 0 plane.
- **Plane 2** is probed from the left face of the block and defines the Y 0 plane.
- **Plane 3** is probed from the front face of the block defining the X 0 plane.

9 From the **Lines**  flyout, click **Line: Two Planes**  .

10 Enter the name **Xline.**

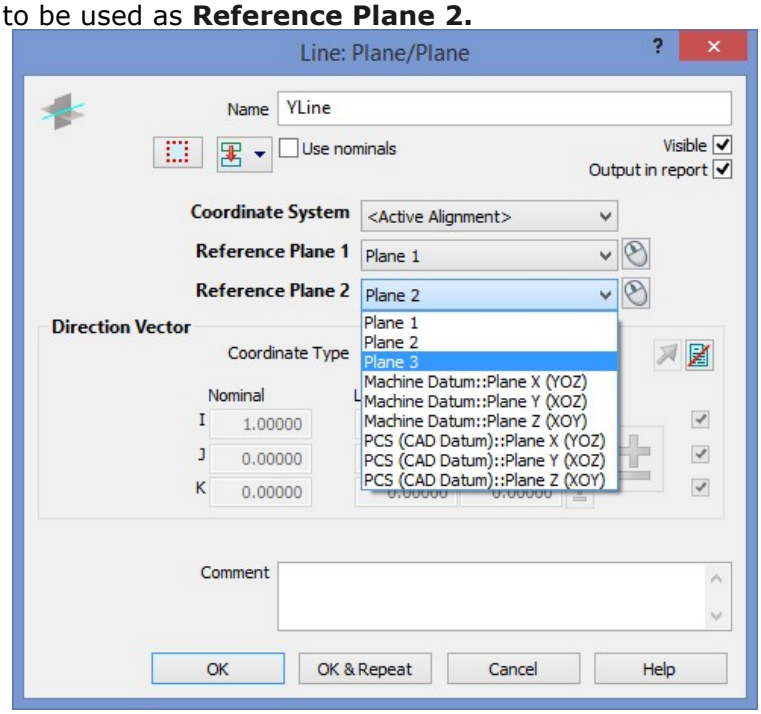


The new line is to be generated from the intersection of the CMM table, Plane 1, and the front face of the block, Plane 2. This line is used to define the X axis in the PLP later.

11 Click **OK & Repeat**.

12 Enter the name **Yline.**

13 Select **Plane 3** to be used as **Reference Plane 2.**

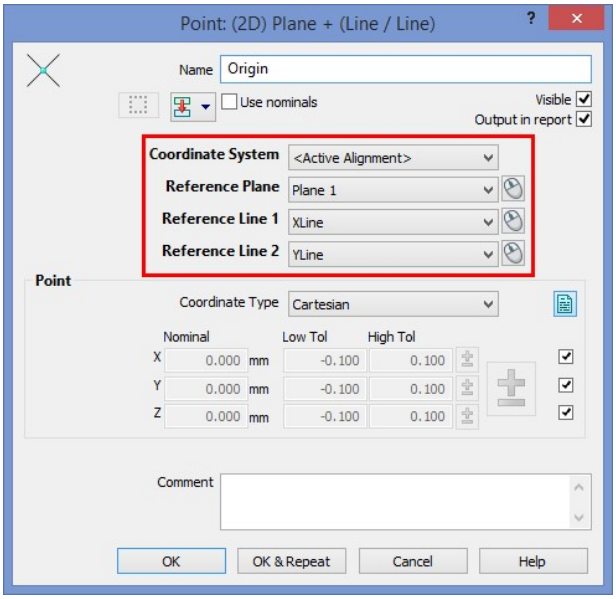


14 Click **OK**.

Now, a point is created at the intersection of **Xline** & **Yline**.

15 From the **Points**  flyout, select **Point from Lines Intersection**  .

16 Enter the name **Origin** and ensure the dialog is completed as below.



 *This point is used as the **XYZ** origin in the PLP alignment.*

17 Click **OK**.

18 Select **Play All**  to probe the three planes.

- Plane 1 — Z0
- Plane 2 — Y0
- Plane 3 — X0

The lines and point are generated automatically from the probed planes.

Creating the PLP alignment

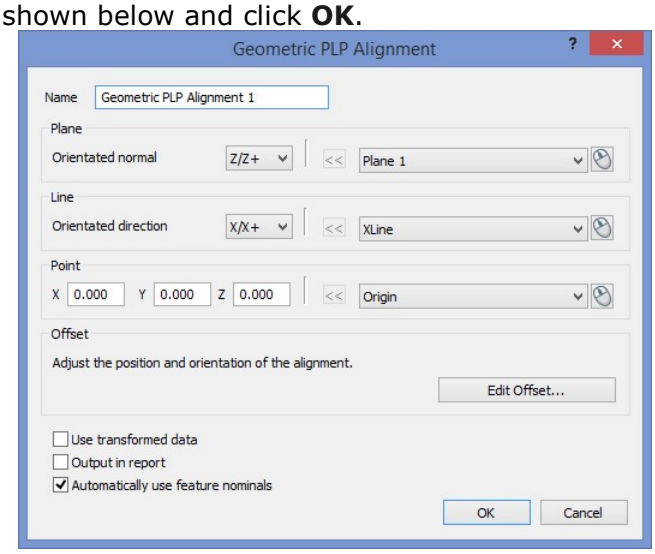
With the geometric items defined and measured, we can create the PLP alignment.

1 Click  to exit **Datum Group**.

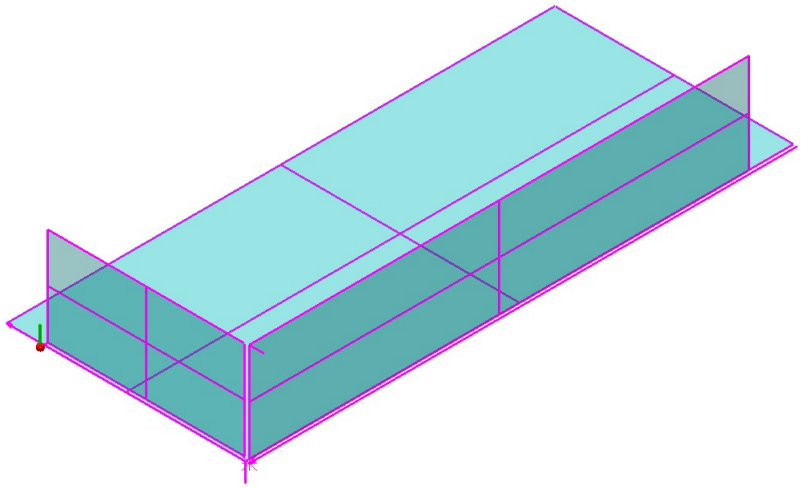
2 Click **Alignments**  to open the **Alignments** flyout.

3 Click **Geometric PLP Alignment**  .

4 Fill in the dialog as shown below and click **OK**.

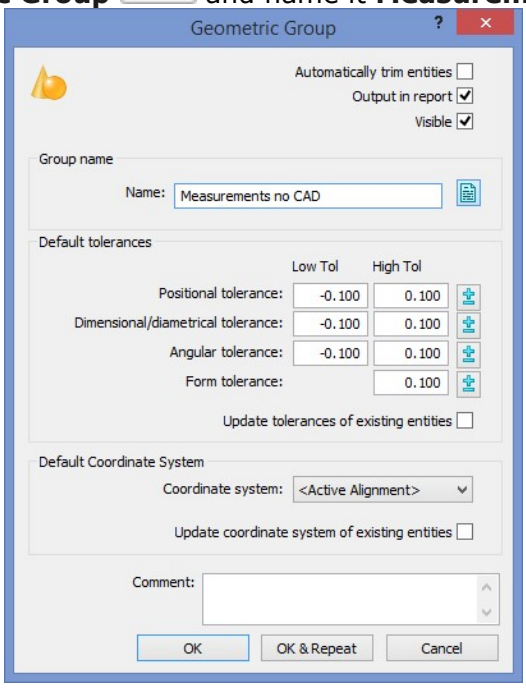


The CAD co-ordinate system is now aligned to the demoblock2008 part. The generated lines and point can be seen in the CAD view. This allows measurements to be made relative to the X, Y and Z axes of the block.



The first dimension to be measured is the height of the block. In this case a **Probed Parallel Plane** item is used.

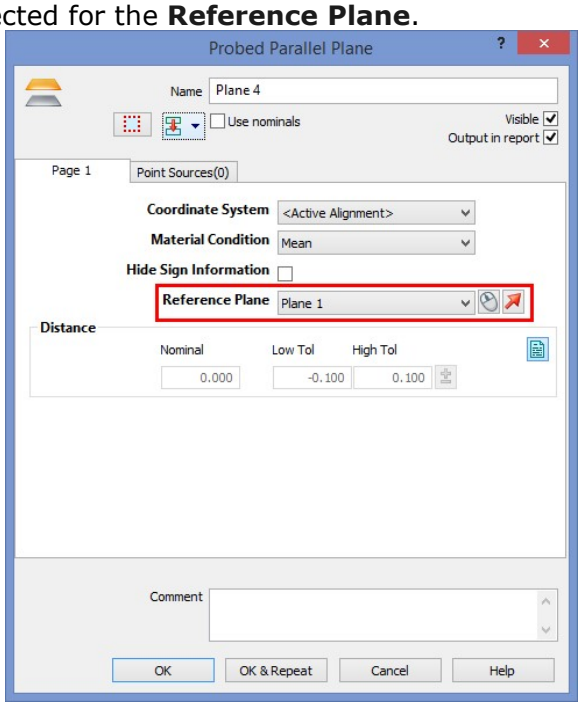
5 Create a new **Geometric Group**  and name it **Measurements no CAD.**



Although the measurements are output in the report, there are no CAD nominals to report against. If a drawing is available, the nominals can be set by modifying each geometrical item to be probed.

6 From the **Planes**  flyout, click **Probed Parallel Plane**  .

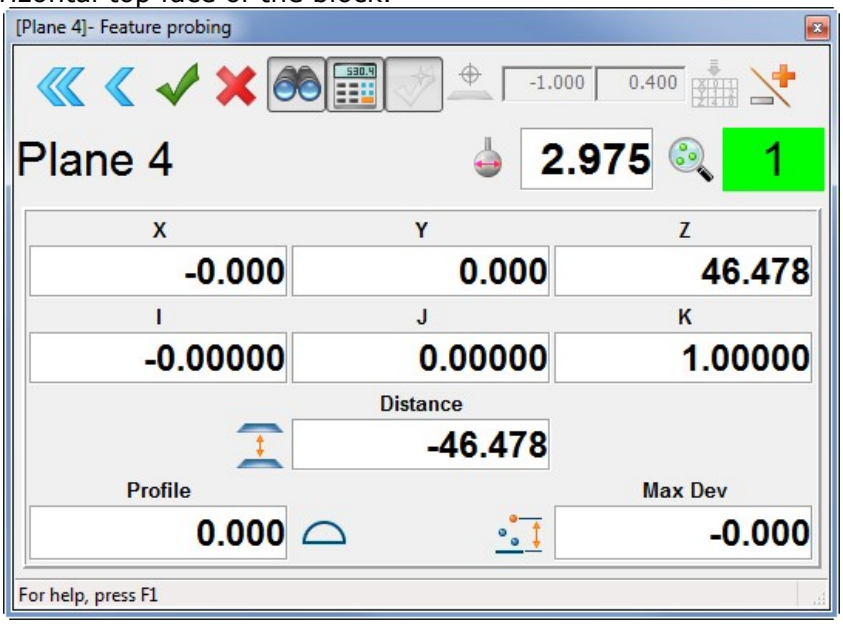
7 Ensure **Plane 1** is selected for the **Reference Plane**.



8 Click **OK.**

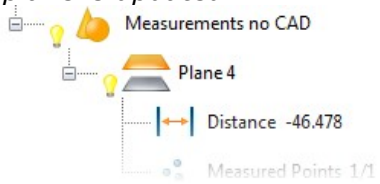
9 Click **Play All**  to play the item.

10 Probe the horizontal top face of the block.

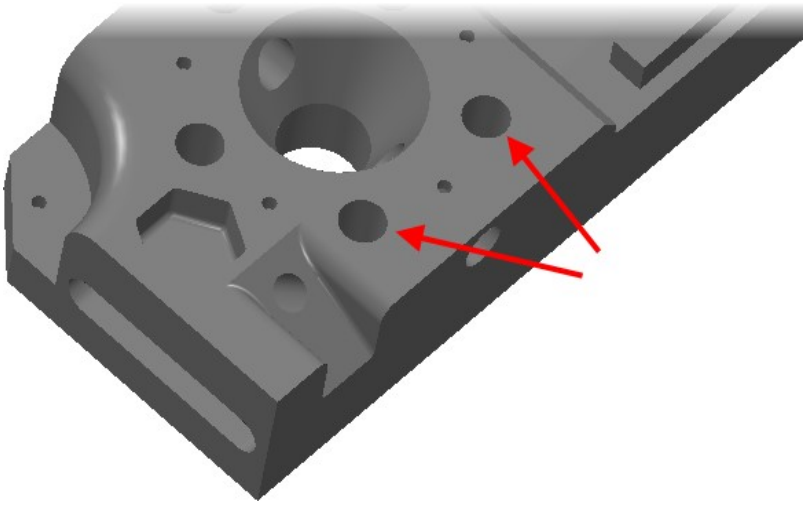


*The **Feature Probing** dialog shows the distance between the CMM surface and the top of the block, the flatness of the top surface, and the maximum deviation of points measured from the best fit plane.*

The flatness and maximum deviation are not shown in the report.In the Sequence Tree, the height of the plane is updated.



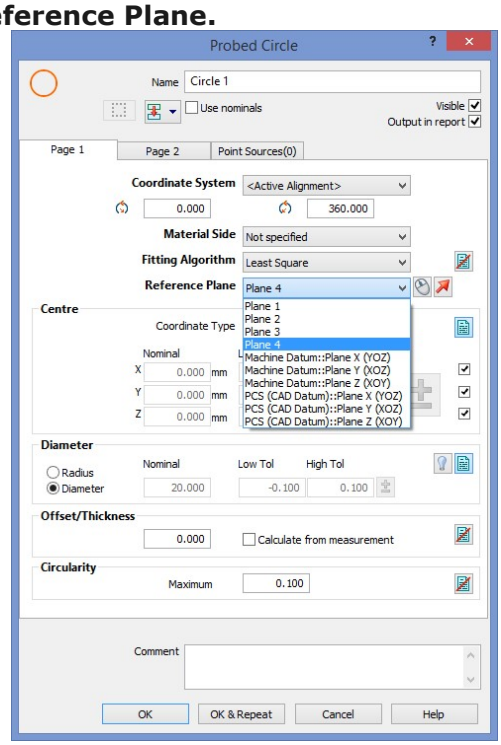
We will now measure the sizes and relative positions of two holes.



11 Click  to open the **Measurements No CAD** group.

12 In the **Features**  toolbar, select **Probed Circle**  .

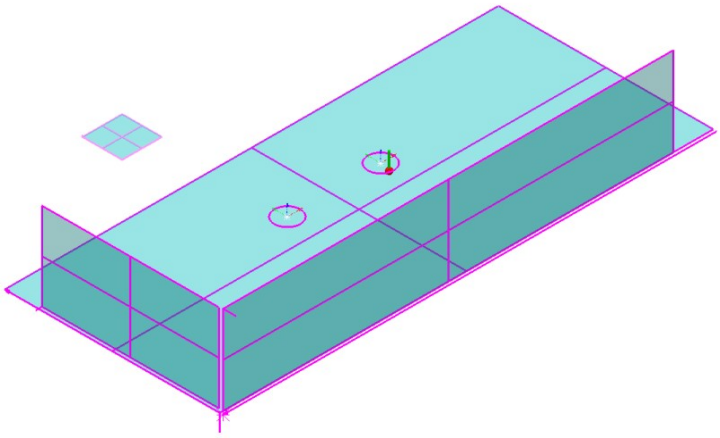
13 Select **Plane 4** as the **Reference Plane.**



14 Leave the **Fitting algorithm** as **Least square**.

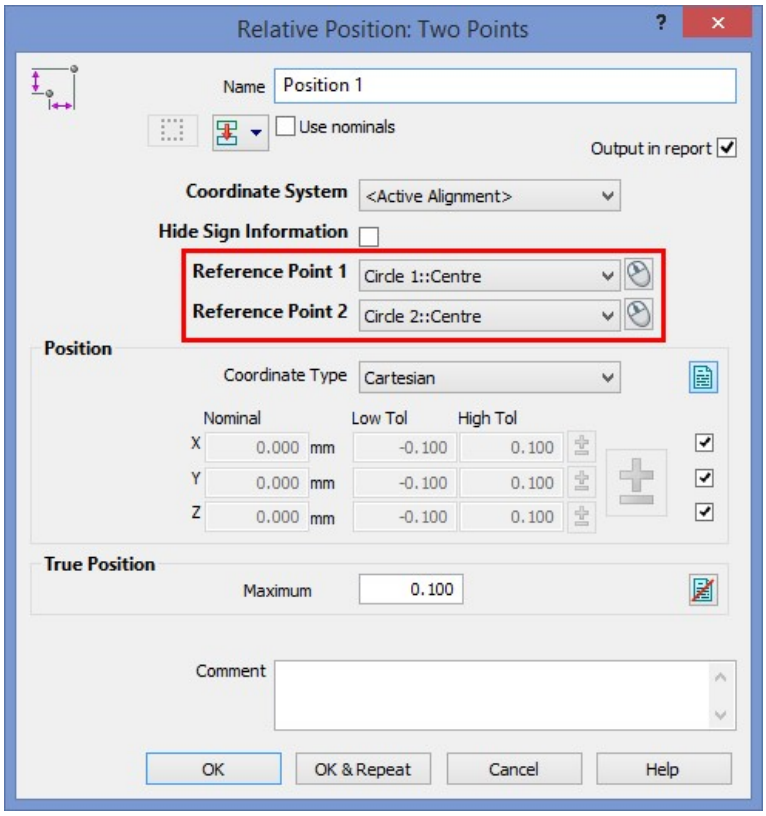
15 Select **OK & Repeat** to create a second Probed Circle.

16 Play both items and measure the circular holes in the block using the techniques previously learned.

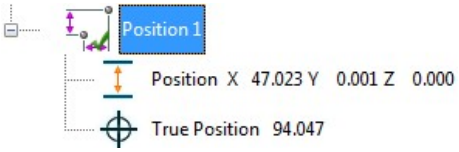


17 In the **Dimensions**  toolbar, click the **Relative Position**  button.

18 Complete the dialog as below and select **OK**.

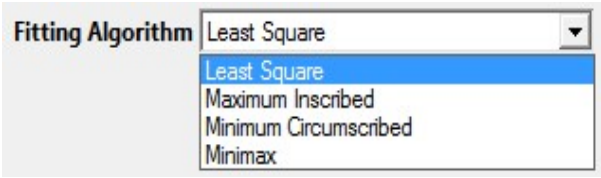


From the Sequence Tree, the relative positions between the two circle centres can be seen referenced to the X and Y axes of the PLP.



Fitting Algorithms

If required, a different fitting algorithm can be applied to the feature. Where available, fitting algorithms can be selected from the drop-down list in the feature form.



The following algorithms are available:

- **Least Square** — Creates a feature that best fits the probed points. The feature is calculated by minimizing the sum of squares of the deviations between the feature and each probed point.
- **Maximum Inscribed** — Creates the largest feature that fits within the probed points.
- **Minimum Circumscribed** — Creates the smallest feature that contains all the probed points.
- **Minimax** — Creates the feature by averaging the maximum inscribed and minimum circumscribed features that have the same centre.

  By default, items are created using the **Least square** algorithm.

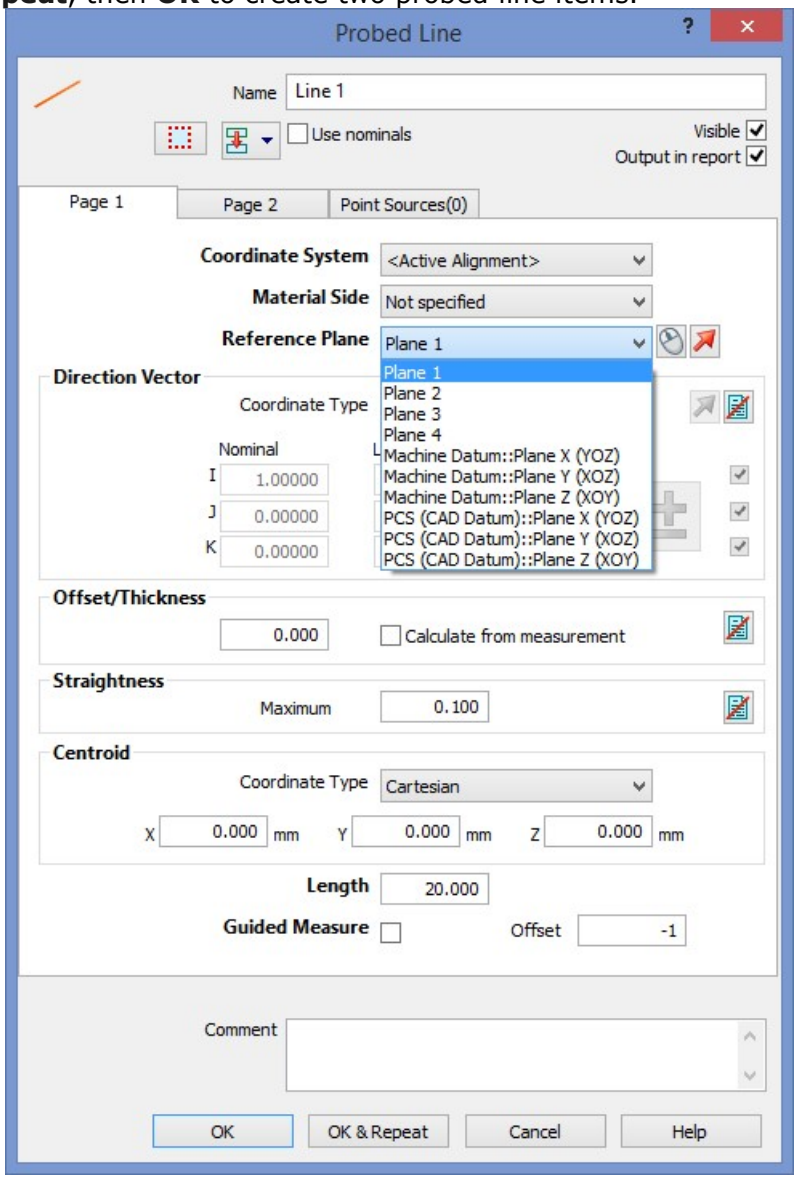
Measuring the length of the block in X and Y

Finally, we will measure the overall length of the block in the X and Y directions. The distances are measured using the origin point created at the beginning of this chapter, plus two new Probed Lines.

1 In the **Lines** flyout, select **Probed Line** .

2 Select **Plane 1** as the **Reference Plane**.

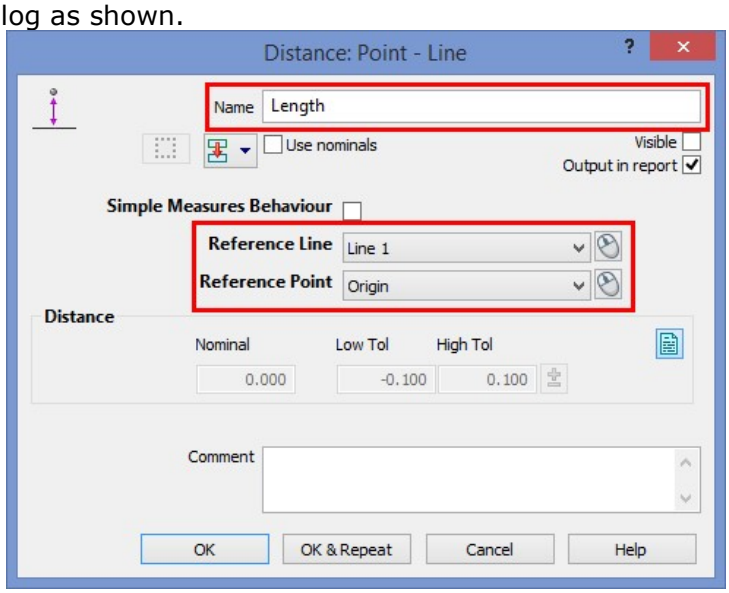
3 Click **OK & Repeat**, then **OK** to create two probed line items.



4 Click to play the lines. Measure the first line on the *right* of the block (parallel to plane 3) and the second on the *back* (parallel to plane 2).

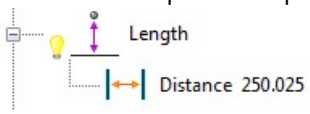
5 In the **Dimensions** flyout, click **Distance: Point - Line** .

6 Complete the dialog as shown.



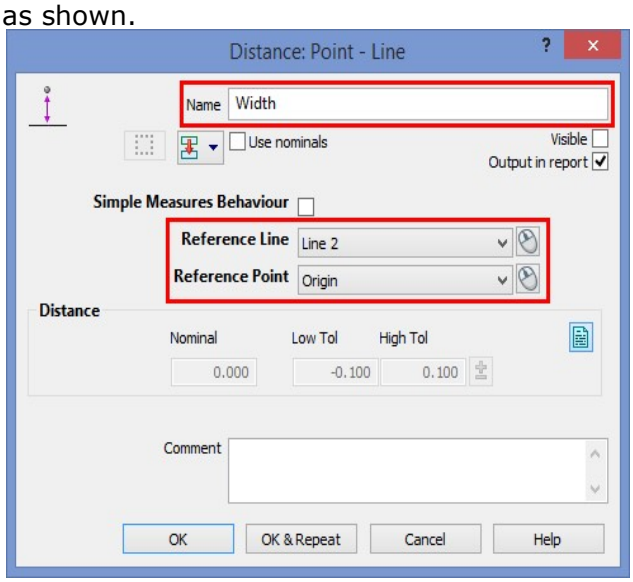
7 Click **OK**.

Line 1 is on the right face of the block, the origin is at the PLP datum origin. The distance between them gives the overall length of the block in X. This is shown in the updated Sequence Tree below:



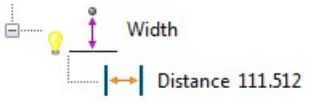
8 In the **Dimensions**  toolbar, select **Distance: Point - Line**  .

9 Complete the dialog as shown.



10 Click **OK**.

***Line 2** is on the back face of the block, the origin is at the PLP datum origin. The distance between them gives the overall width of the block in Y. This is shown in the updated Sequence Tree below:*



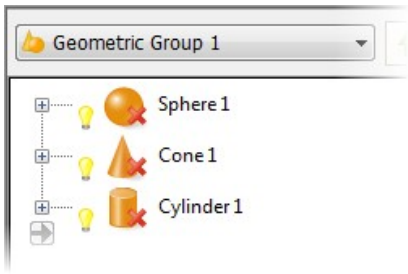
Probing 3D Features

3D features will now be added to the inspection.

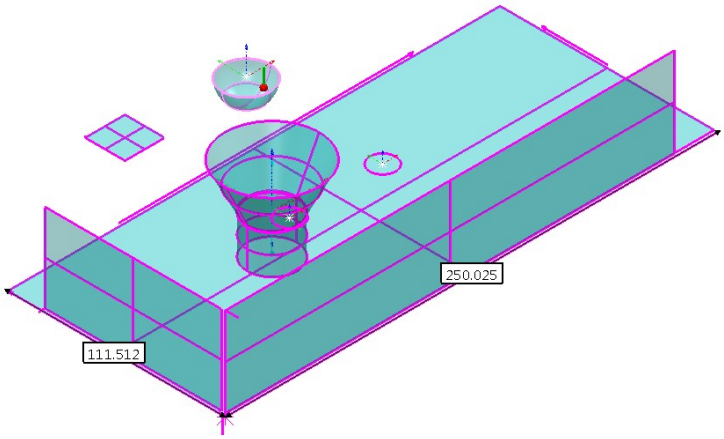
1 Click  to exit the **Measurement no CAD** group.

2 Click  , then **OK** to create a new Geometric group.

3 From the **Features** toolbar  , select **Probed Sphere**  , **Probed Cone**  and **Probed Cylinder**  . Keep the default settings for all three items.
The Sequence Tree lists the features.

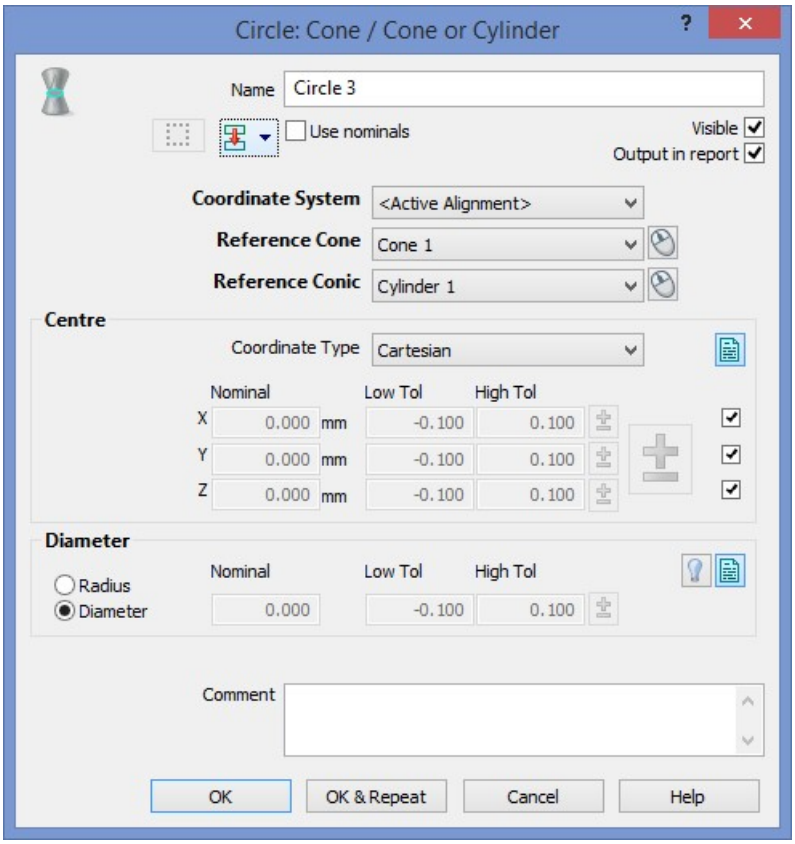


4 Select **Play All**  and probe the three features in the order defined.



 *The CAD view is updated to show the new probed features.*

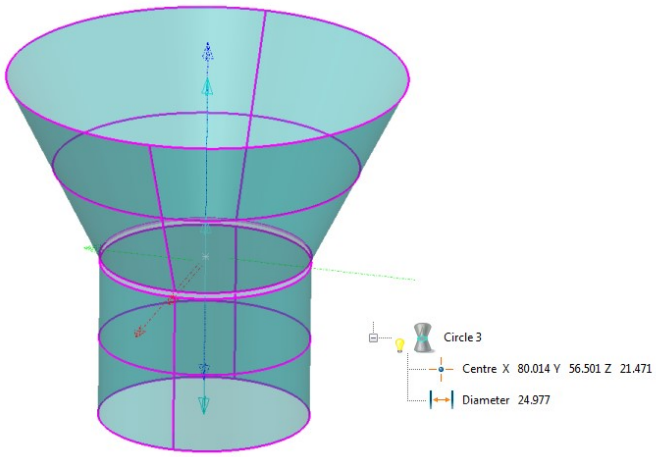
5 In Geometric Group 1, click **Circle: Cone Cylinder intersection**  in the **Features** toolbar  .



6 Select the **Cone** and **Cylinder** as reference features.

7 Click **OK**.

Circle 3 *is generated at the intersection of the cone and cylinder.*



The report shows the details of this inspection. Nominal dimensions need to be added to provide error/deviation results. These can be added at this stage by using the **Modify Item** button in the Sequence Tree to open each item in turn and change the nominal values.

Measurements no CAD

Plane 4 (Datum - *Geometric PLP Alignment 1*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Distance	-----	-----	-----	44.928	-----	-----

Circle 1 (Datum - *Geometric PLP Alignment 1*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	X	-----	-----	-----	59.620	-----	-----
Centre	Y	-----	-----	-----	25.391	-----	-----
	Z	-----	-----	-----	44.928	-----	-----
Diameter		-----	-----	-----	7.134	-----	-----

Circle 2 (Datum - *Geometric PLP Alignment 1*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	X	-----	-----	-----	106.625	-----	-----
Centre	Y	-----	-----	-----	25.397	-----	-----
	Z	-----	-----	-----	44.928	-----	-----
Diameter		-----	-----	-----	7.014	-----	-----

Position 1 (Datum - *Geometric PLP Alignment 1*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	X	-----	-----	-----	47.005	-----	-----
Position	Y	-----	-----	-----	0.005	-----	-----
	Z	-----	-----	-----	0.000	-----	-----

Measurements no CAD

length (References: *Origin, Line 1*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Distance	-----	-----	-----	255.213	-----	-----

Width (References: *Origin, Line 2*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Distance	-----	-----	-----	116.308	-----	-----

Geometric Group 1

Sphere 1 (Datum - *Geometric PLP Alignment 1*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Diameter		-----	-----	-----	18.086	-----	-----
	X	-----	-----	-----	111.905	-----	-----
Centre	Y	-----	-----	-----	97.267	-----	-----
	Z	-----	-----	-----	41.943	-----	-----

Cylinder 1 (Datum - *Geometric PLP Alignment 1*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Diameter	-----	-----	-----	19.133	-----	-----

Cone 1 (Datum - *Geometric PLP Alignment 1*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Half Angle	-----	-----	-----	26.391	-----	-----

Circle 3 (Datum - *Geometric PLP Alignment 1*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	X	-----	-----	-----	85.526	-----	-----
Centre	Y	-----	-----	-----	56.911	-----	-----
	Z	-----	-----	-----	17.572	-----	-----
Diameter		-----	-----	-----	19.133	-----	-----

There are other ways in which it is possible to measure dimensions from a part. The aim of this example is to give an introduction to the basics of measuring without CAD data. For details of alternative methods, please ask your tutor.

Points can also be taken on the component without a CAD model. This can be useful when, for example, reverse engineering parts. Point clouds, Points on the Fly and digitised curves are methods outlined in the following chapter. The points taken will be included into the current inspection sequence.

7. Dimensioning

The Dimensioning function in PowerInspect enables you to create and display measurements in the CAD view, which can then be included in the report and viewed using the **Info** tab. You can use it to display distances and angles between features, and the sizes of features that are included in the inspection sequence.

In the examples used in this chapter, the dimensions are created from measured features, but you can also create dimensions from nominals.

Distance dimensions

Create distance dimensions to display distances between features on a part. You can create several types of distance dimension, for example.

- Linear distance between two points.
- Linear distance between a datum and a point.
- 3D distance between two points.
- Distance between two planes.
- Distance between two circles.
- Perpendicular distance between a point and a line.

Each type creates a different item in the inspection sequence.

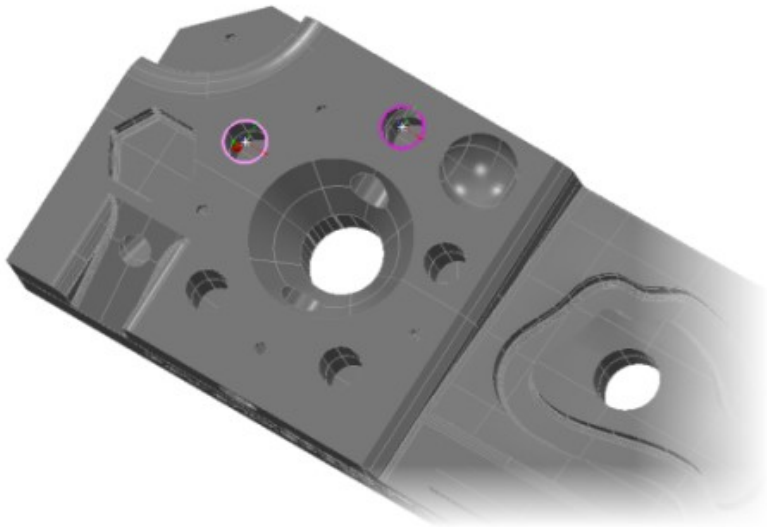
The following examples demonstrate how to display the linear distance between two points, and the distance between two circles.

Displaying the linear distance between two points in a datum plane

To create a linear dimension between two points.

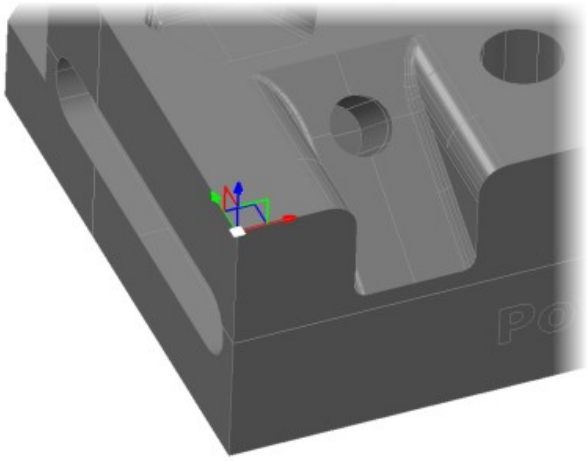
1. Click  to create a **New Document using the Wizard**. In the wizard, select **Measurement with a single CAD part** and click **Next**.
2. Browse for **Demoblock2008(CMM+Arm).dgk** and click **Finish** to open the file.
3. Create an alignment using one of the previously learned methods.

4 Create a new geometric group, and then create and play the two Probed Circle items shown below.



5 Click **View from Top (Z)**  to ensure that the points are projected onto the **XY** plane.

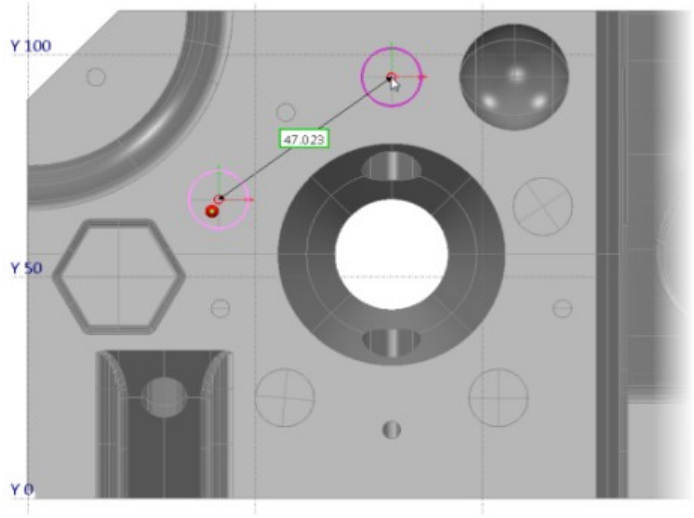
6 In the **Mouse Context** toolbar, select **Dimensioning**  . A white dimensioning plane is displayed on the datum, showing you the plane in which the labels will be drawn.



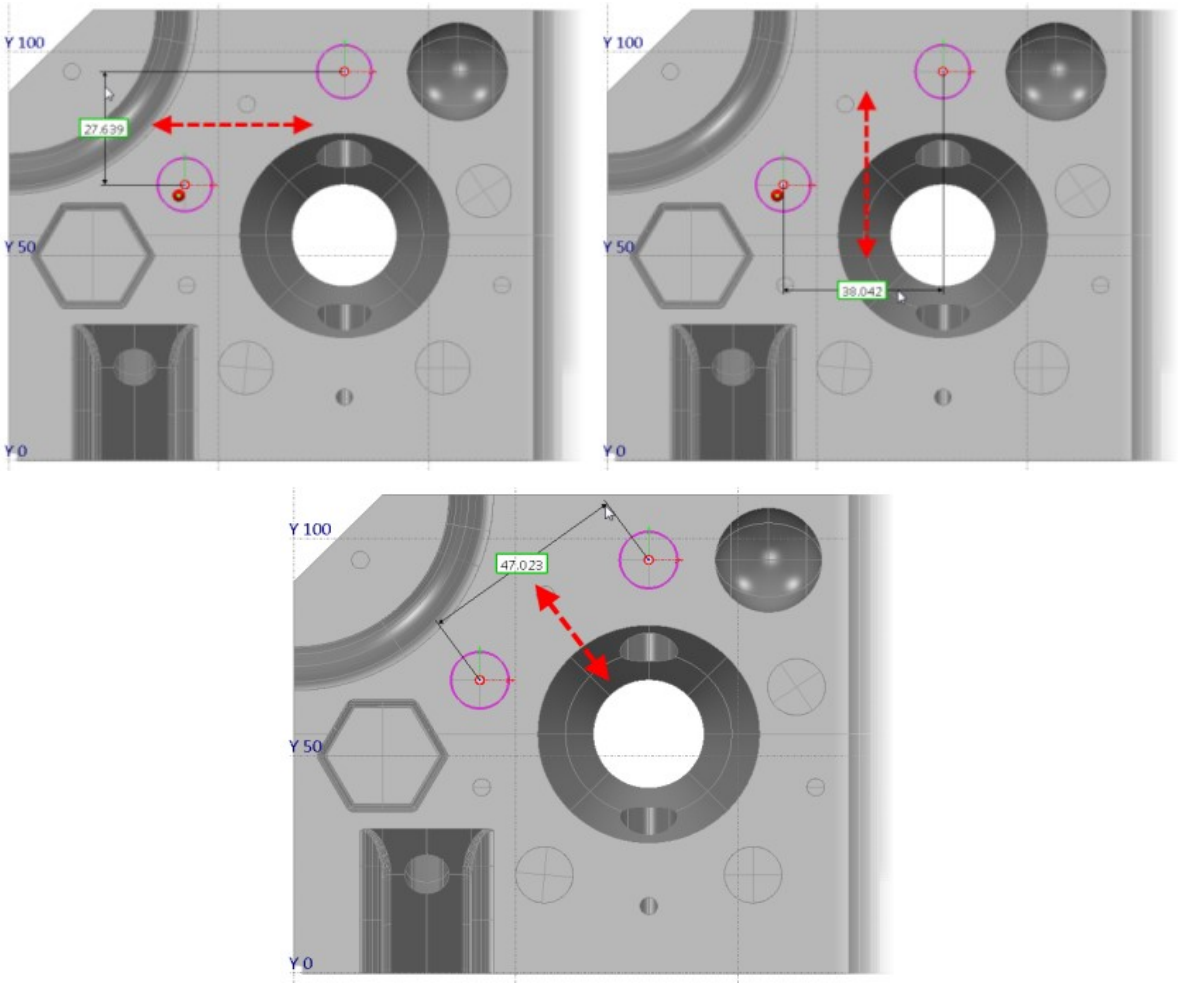
If you want to change the plane that is used, rotate the model in the CAD View until you are looking down the plane you want to use, and then double-click the dimensioning plane.

7 Move the cursor over the centre of the first circle. When it is displayed in yellow, click to select it. The colour changes to red.

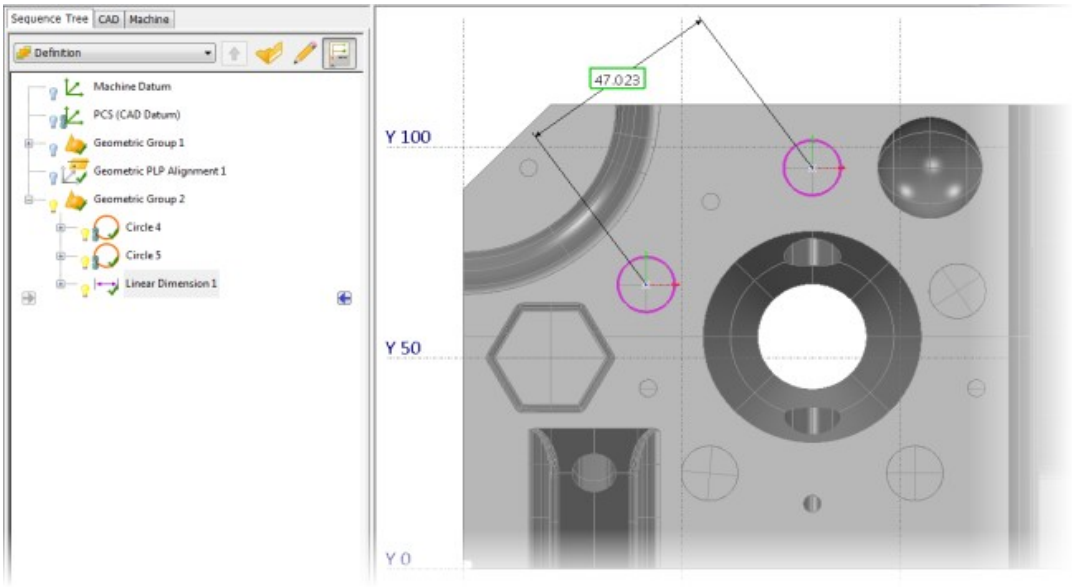
8 Move the cursor over the centre of the second circle. When it is displayed in yellow, click to select it. The colour changes to red, and a label showing the linear dimension is displayed.



9 Use the cursor to control which measurement is displayed.



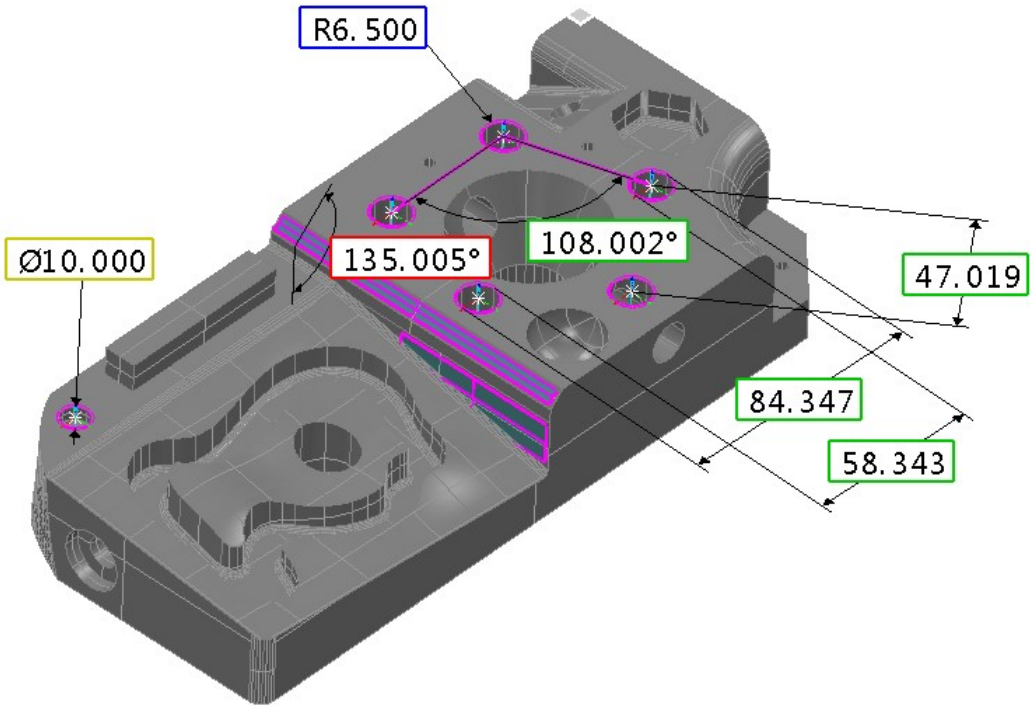
10 Left-click to drop the label and create a Linear Dimension item in the inspection sequence.



Understanding the label colours

When you create a Dimension, the borders are coloured to indicate its status.

- Gold indicates one or both of the items are unmeasured.
- Green indicates the dimension is within tolerance.
- Red indicates the dimension is above tolerance.
- Blue indicates the dimension is below tolerance.
- Black indicates the dimension has not been compared with the CAD nominals.



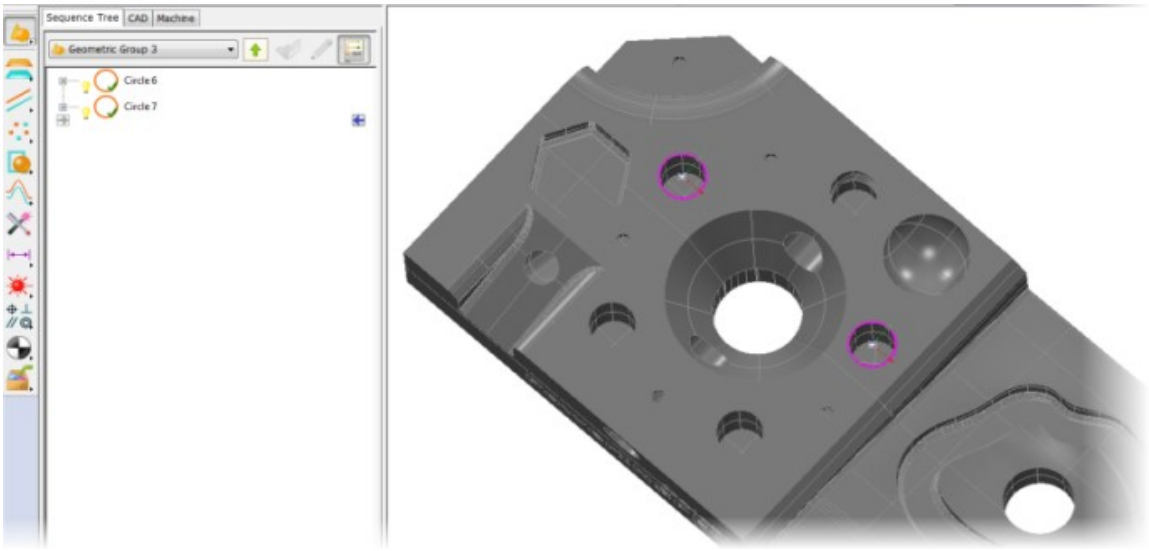
You can change the colours that are used for the labels by selecting **Tools > Options > Display Options > Colours** and editing the **Labels and Graphical Dimensions** page.

Displaying the distance between two circles

When you measure the distance between two circles, the measurement displayed depends on the relative positions at which the circumferences are clicked. For example, if you click the circumferences where they are closest, the inside measurement is displayed.

To display the distance between two circles.

1 Create a new geometric group, and then create and play the two Probed Circle items shown below.

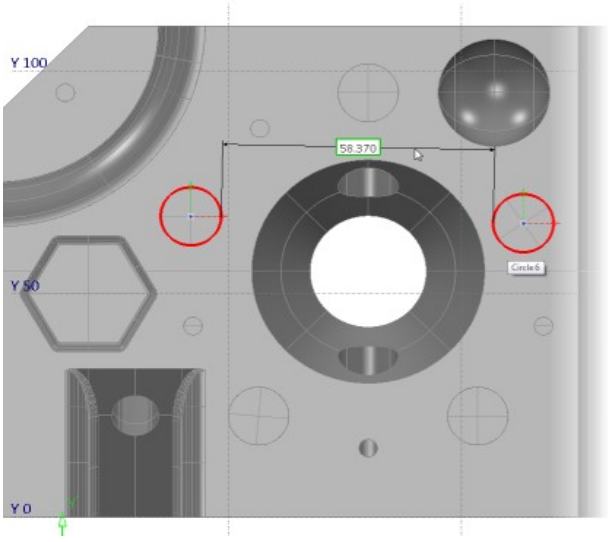


2 Click **View from Top (Z)**  .

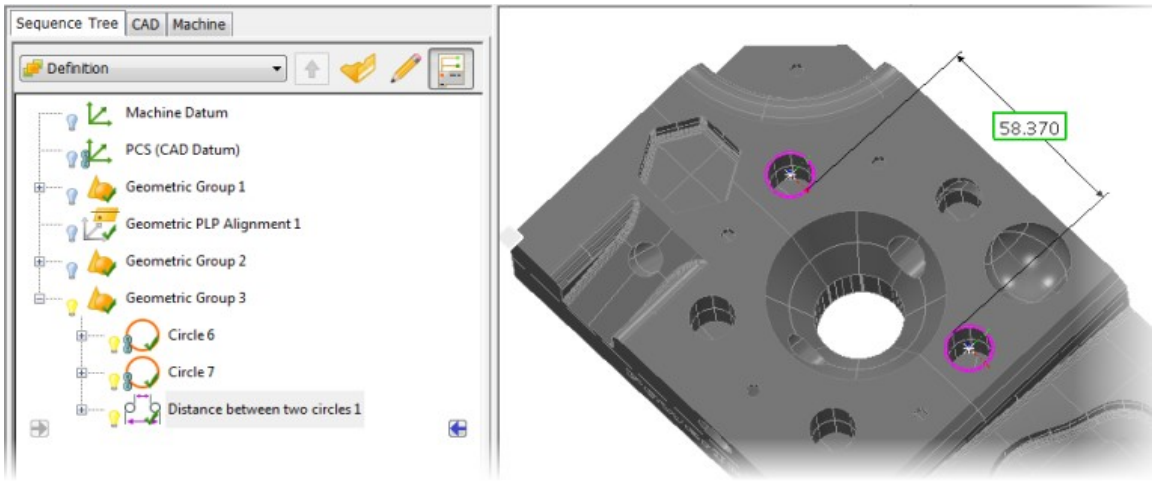
3 In the **Mouse Context** toolbar, select **Dimensioning**  .

4 Press the **Ctrl** key, move the cursor over the first circle, and when it is displayed in yellow, click to select it. The colour changes to red.

5 Keeping **Ctrl** pressed down, move the cursor over the second circle. When it is displayed in yellow, click to select it. The distance between the two circles is displayed. Release **Ctrl**.



6 Move the cursor to position the label, then left-click to drop it and create a Distance between two circles item in the inspection sequence.

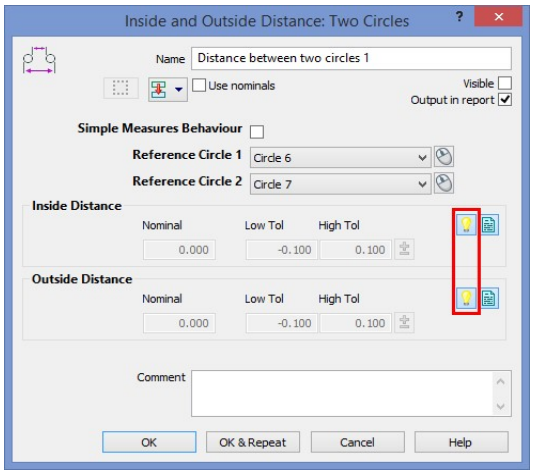


Editing the label

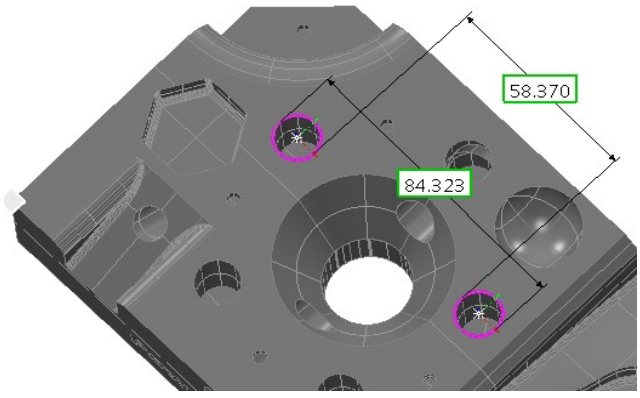
After the label has been dropped, you can choose whether to display the outside distance, inside distance, or both.

1 Double-click the label to open the **Inside and Outside Distance between Two Circles** dialog.

2 Select both light bulbs to display the inside and outside distance in the CAD View.



3 Click **Apply**.



4 Right-click the new label, then select **Move** on the context menu to reposition it.

Angle dimensions

You can create angular Dimensions between features, for example:

- Two lines in a datum plane.
- Two lines projected on a specified plane.
- Two planes.
- A vector and a plane.

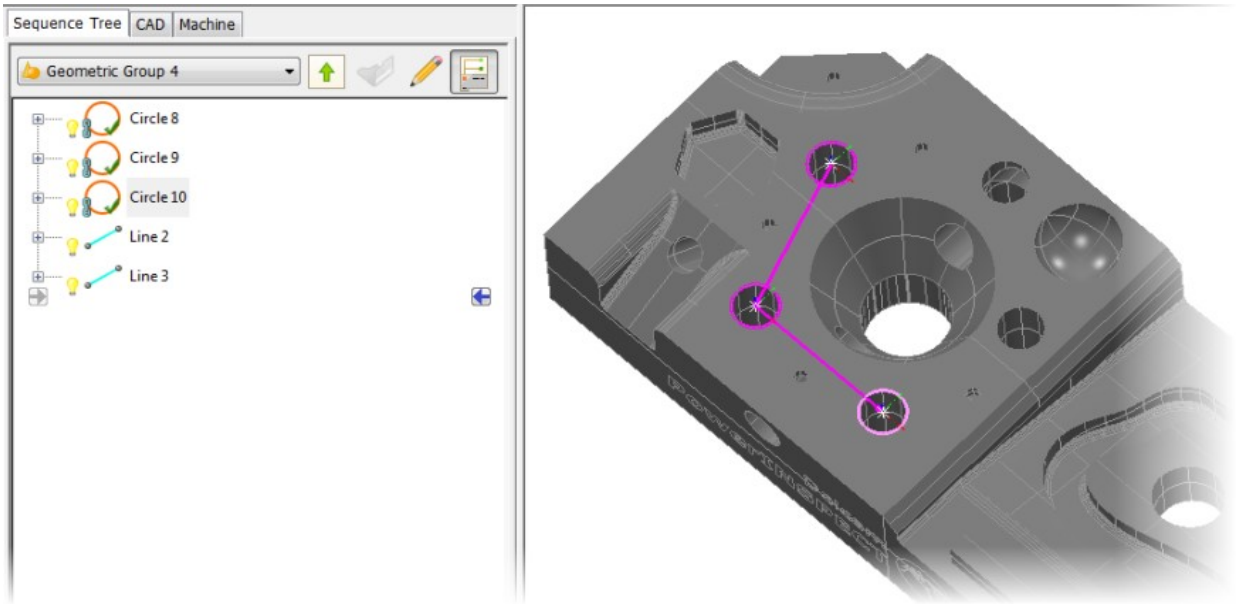
When measuring 2D angle dimensions, it is important to select the correct dimensioning plane because the values displayed depend on the plane in which the measurements are taken.

The following examples demonstrate how to display a 2D angle between two lines, and how to display a 3D angle between two planes.

Displaying an angle between two lines

To display the angle between two lines.

1 Create three probed circles, then create two lines between the centres of the circles, as shown below.

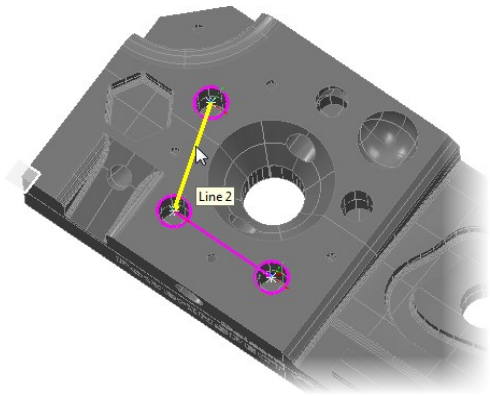


2 Play all the items.

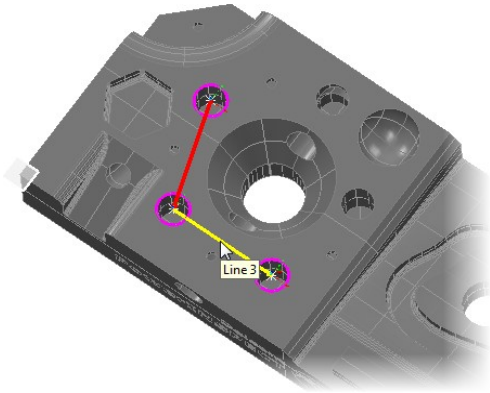
3 Click **View from Top (Z)**  to ensure that the **XY** plane is used for creating the dimension.

4 In the **Mouse Context** toolbar, select **Dimensioning**  .

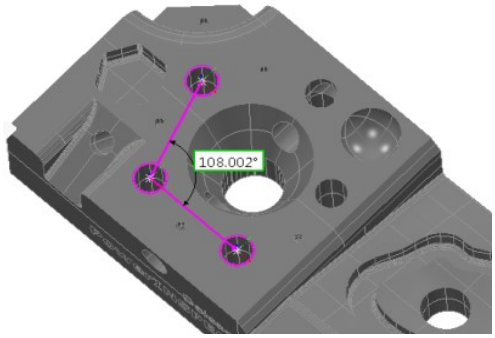
5 Move the cursor over the first line. When it is displayed in yellow, click to select it.



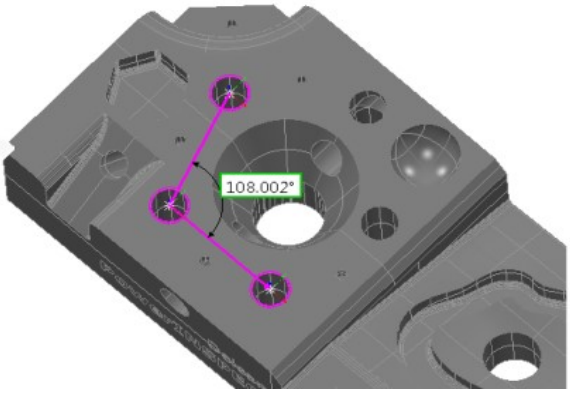
6 Move the cursor over the second line. When it is displayed in yellow, click to select it.

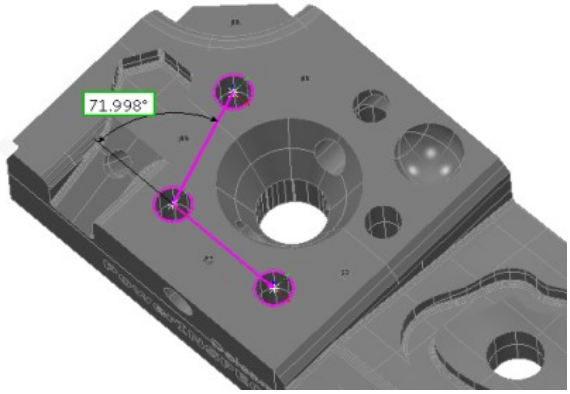


The angle between the two lines is displayed.

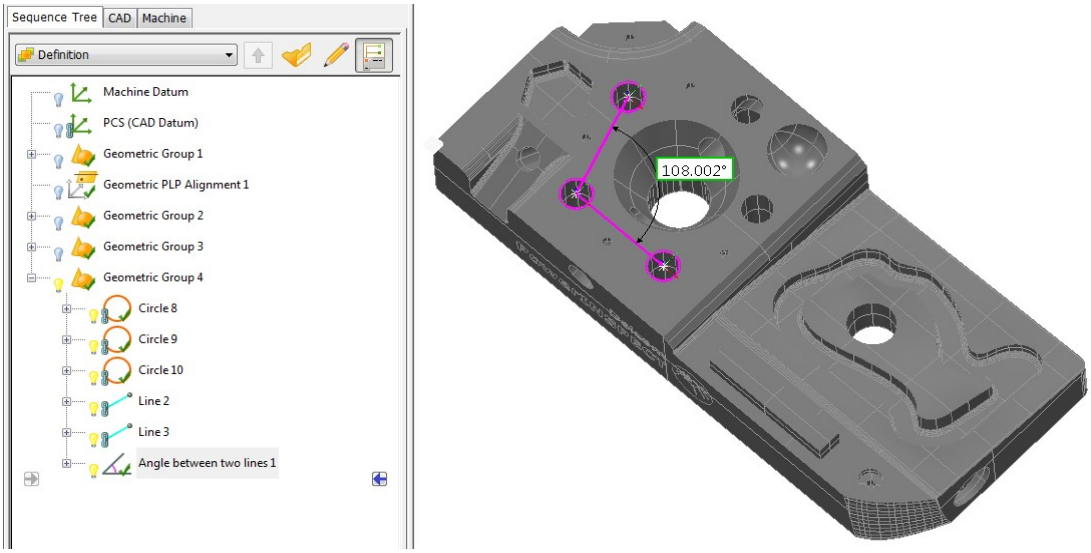


7 Move the cursor to position the label and select which angle is shown.

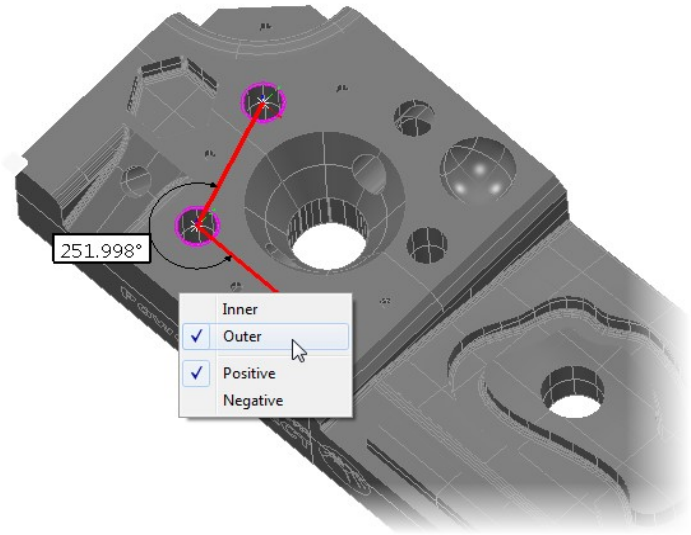




8 When the label is positioned, left-click to drop it and create an Angle between two lines item in the inspection sequence.



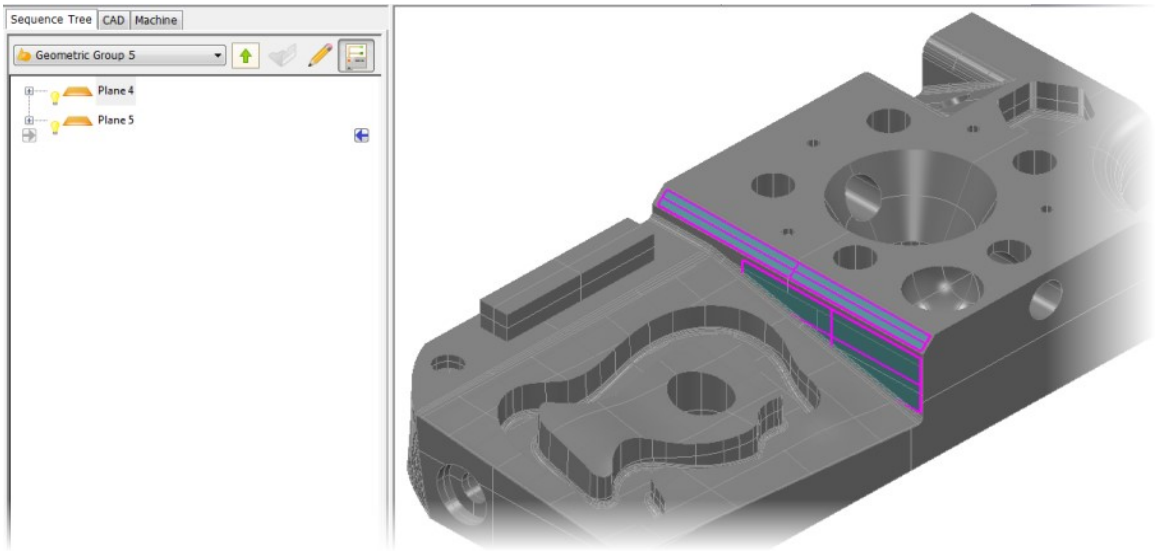
9 To display the outer angle, right-click before positioning the label to open the context menu. You can choose whether to display the inner or outer angle, and whether the angle is displayed as a positive or negative.



Displaying a 3D angle between two planes

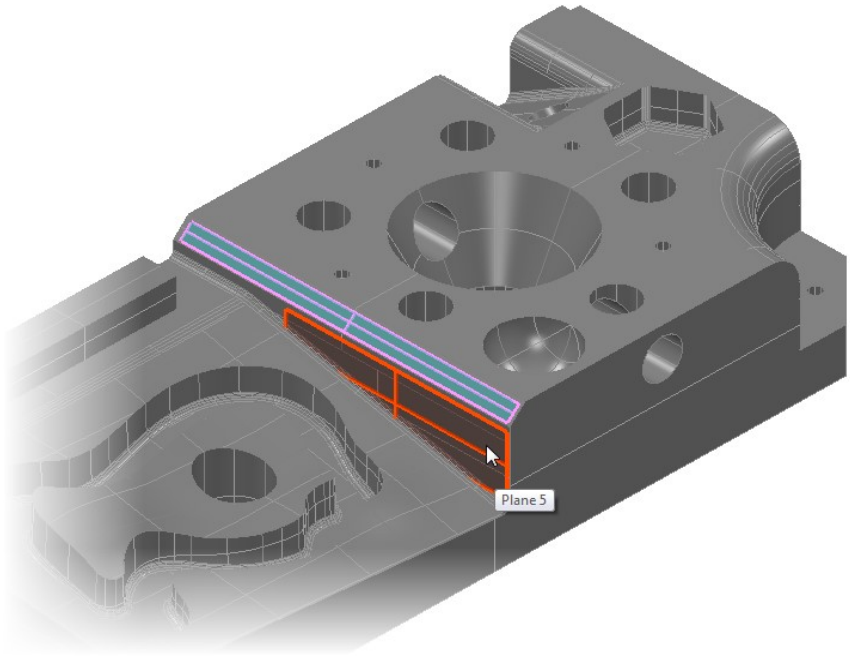
To display the 3D angle between two planes.

1 Create a new geometric group and then create and play the two Probed Plane items shown below.

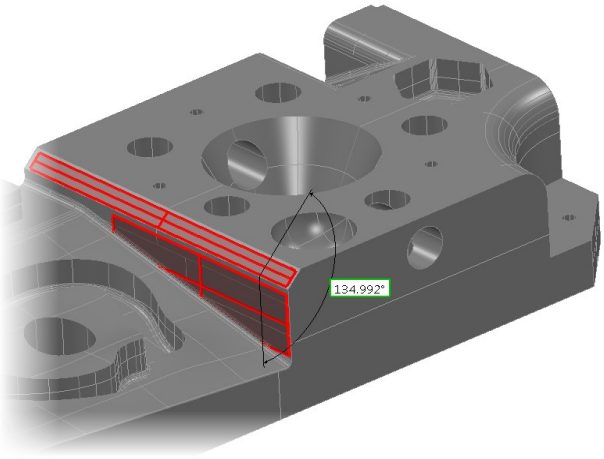


2 In the **Mouse Context** toolbar, select **Dimensioning**  .

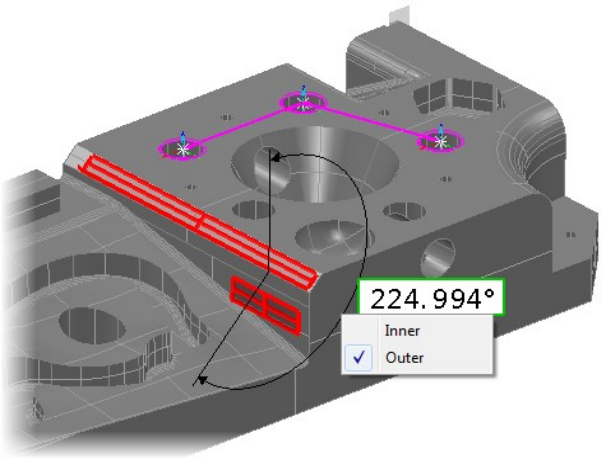
3 Move the cursor over the first plane. When it is displayed in yellow, click to select it. The plane changes to red.



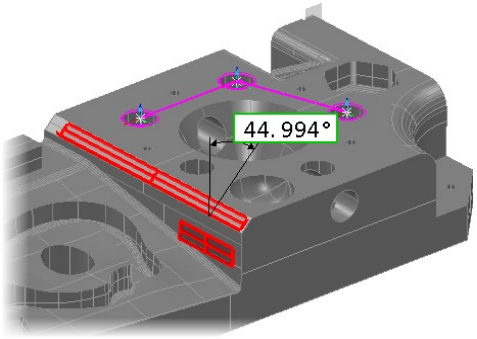
4 Move the cursor over the second plane. When it is displayed in yellow, click to select it. The plane changes to red and the angle is displayed.

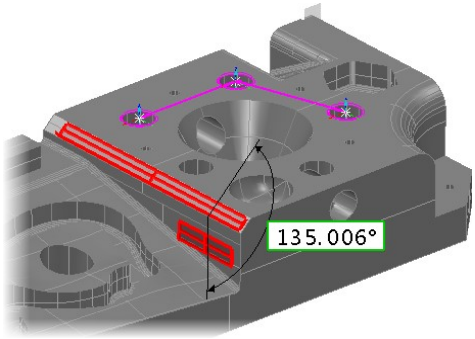


5 Right-click the label to display the context menu and switch between displaying the inner or outer angle.

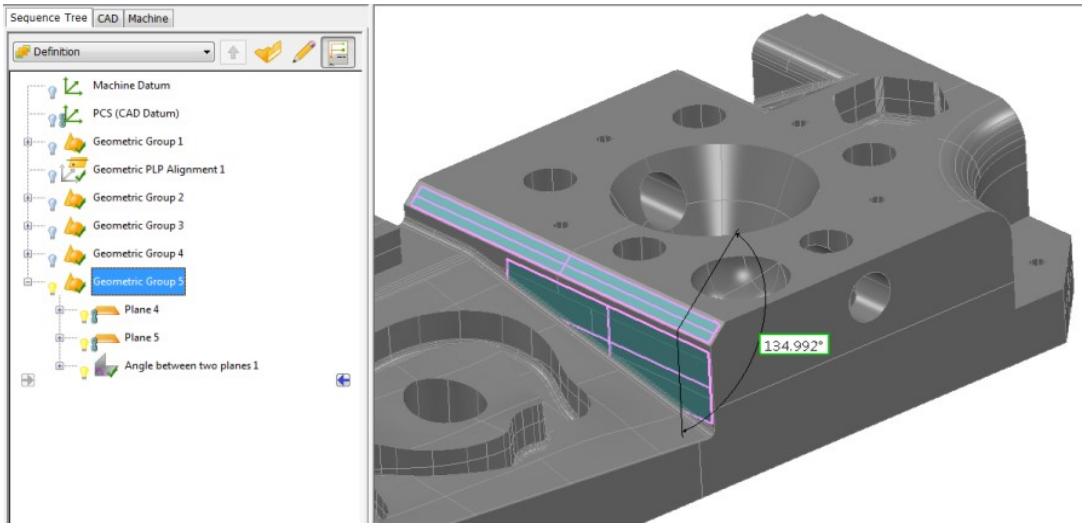


6 Move the cursor to position the label and select which angle is shown.





7 Left-click to drop the label and create an Angle between two planes item in the inspection sequence.



8 To reposition the Dimension after it has been placed, right-click the Dimension and select **Move** from the context menu.

Feature dimensions

You can create dimensions for features using a similar method to creating other dimensions. Unlike other dimensions, however, you do not need to select a dimensioning plane, and these dimensions can only be created for features that are already in the inspection sequence. Feature dimensions that can be displayed include:

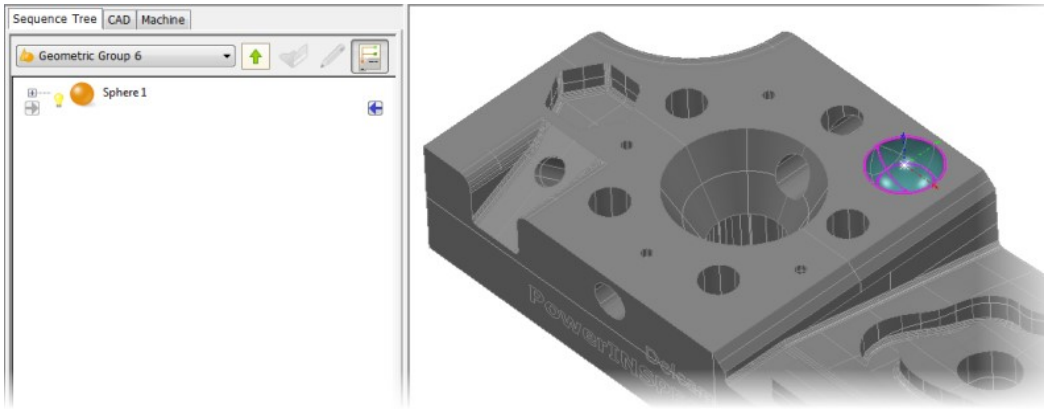
- The diameter of a sphere or circle.
- The radius of a sphere or circle.
- The length of a slot or rectangle.
- The width of a slot or rectangle.

The following examples demonstrate how to display the diameter of a sphere, the radius of a circle, and the length and width of a slot.

Displaying the diameter of a sphere

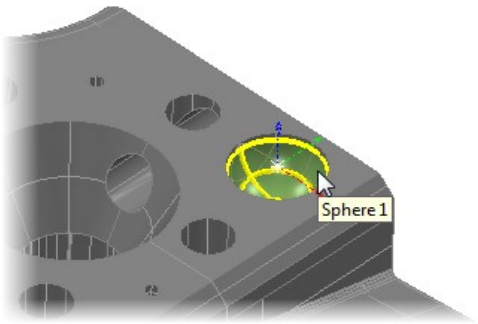
To display the diameter a sphere.

1 Create a new geometric group, and then create and play a Probed Sphere item, as shown below.



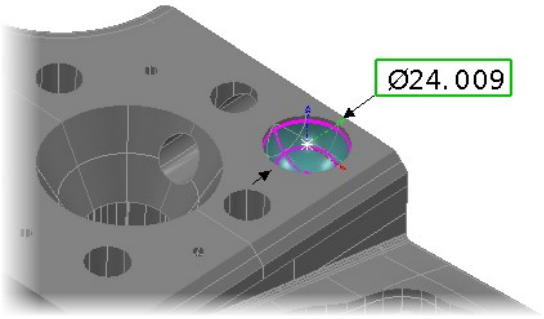
2 In the **Mouse Context** toolbar, select **Dimensioning**  .

3 Move the cursor over the sphere. When the circumference is displayed in yellow, click to select it.



The circumference changes to red and a label is displayed showing the diameter.

4 Move the cursor to position the label, then left-click to drop it.



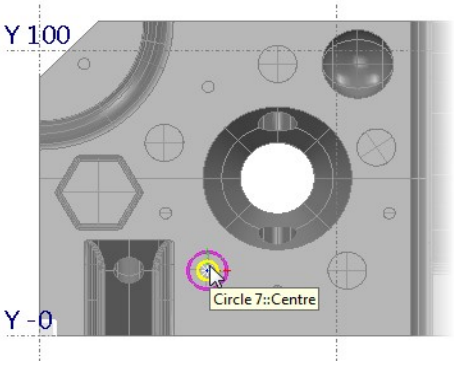
Displaying the radius of a circle

To display the radius of a circle.

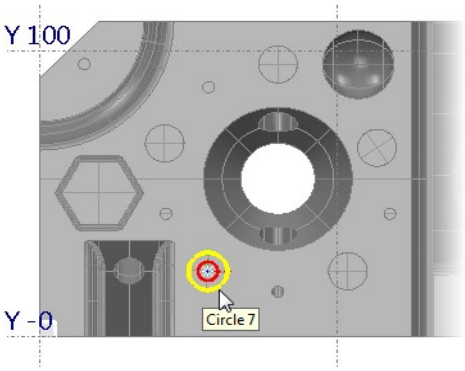
1 Create and play a Probed Circle item.

2 In the **Mouse Context** toolbar, select **Dimensioning**  .

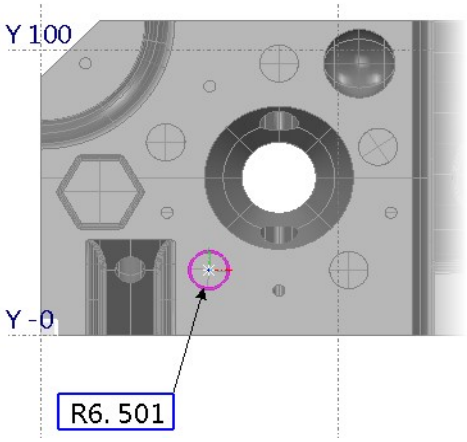
3 Move the cursor over the centre of a circle. When it is displayed in yellow, click to select it.



4 Move the cursor over the circumference of the circle. When it is displayed in yellow, click to select it.



5 Move the cursor to position the label, then left-click to drop it.

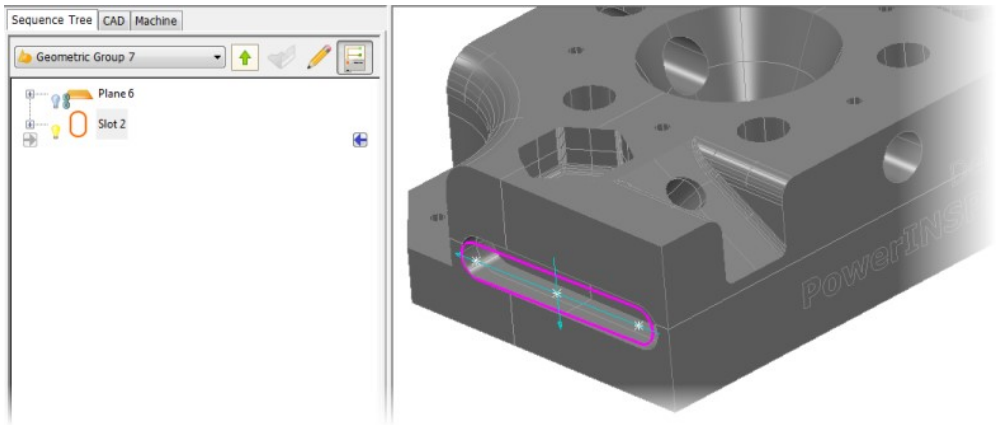


6 To hide the label, right-click it and select **Hide** from the context menu.

Displaying the length and width of a slot

To display the length and width of a slot.

1 Create a probed slot item, as shown below.

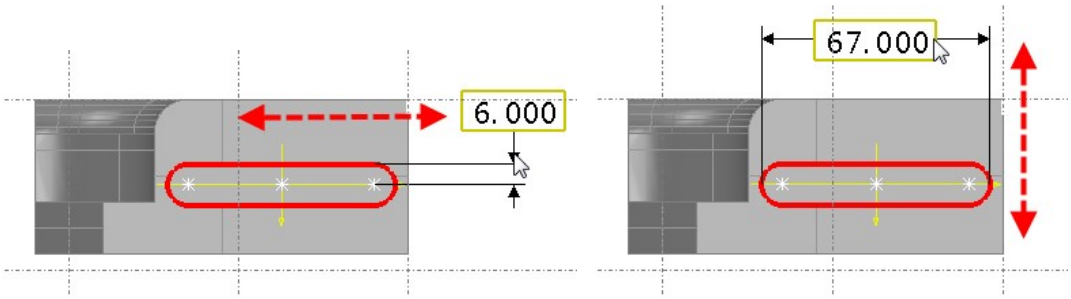


2 In the **Mouse Context** toolbar, select **Dimensioning**  .

3 Move the cursor over the slot. When it is displayed in yellow, click to select it. The colour changes to red and a label is displayed.

If you created a new plane with the slot item, you need to hide the plane before you can select the slot.

4 Use the cursor to change which measurement is displayed in the label.

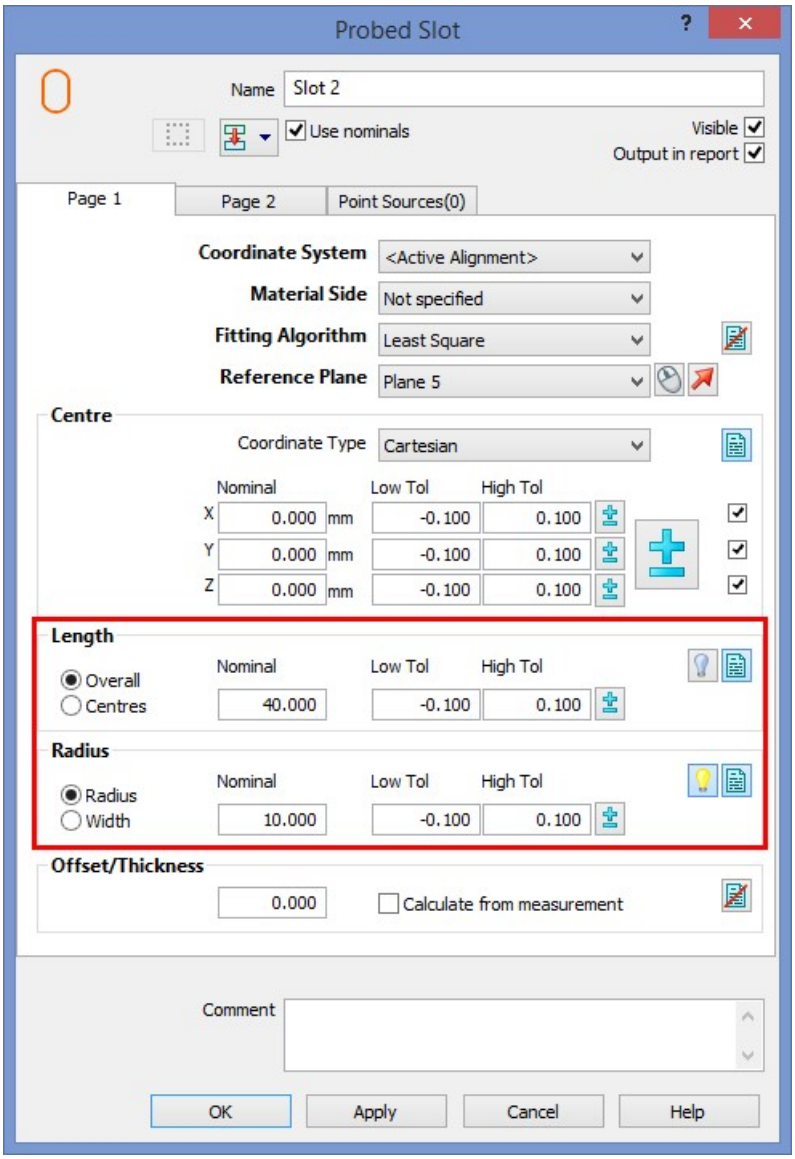


5 Left-click to drop the label.

After you have positioned a label, you can choose which length and radius measurements are displayed.

6 Right-click the label and select **Edit** from the context menu. The **Probed Slot** dialog is displayed.

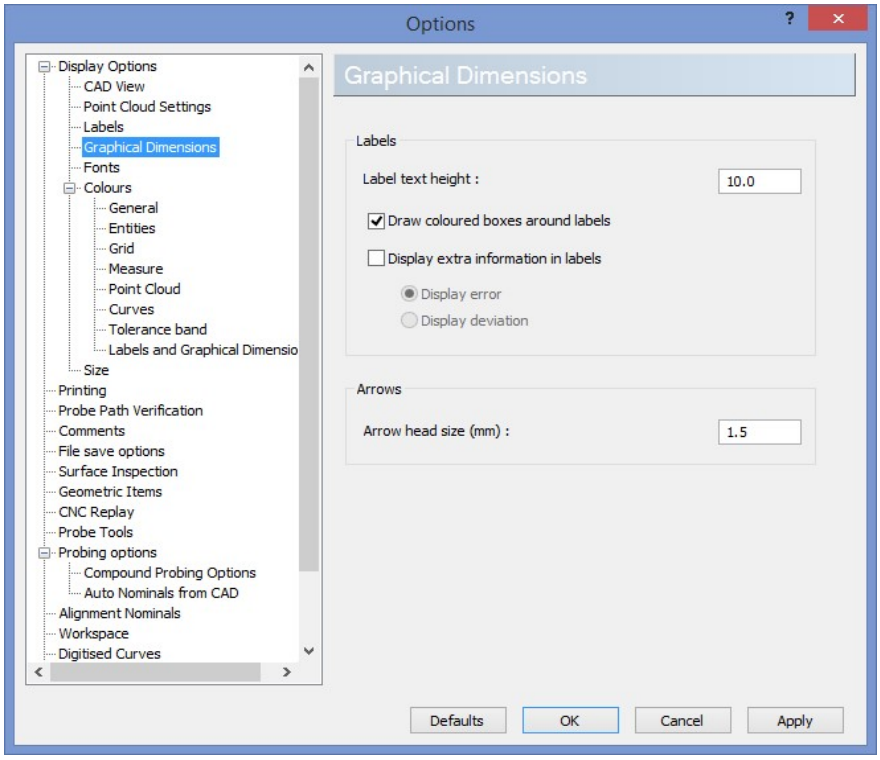
7 Choose the measurements you want to display and click **Apply**.



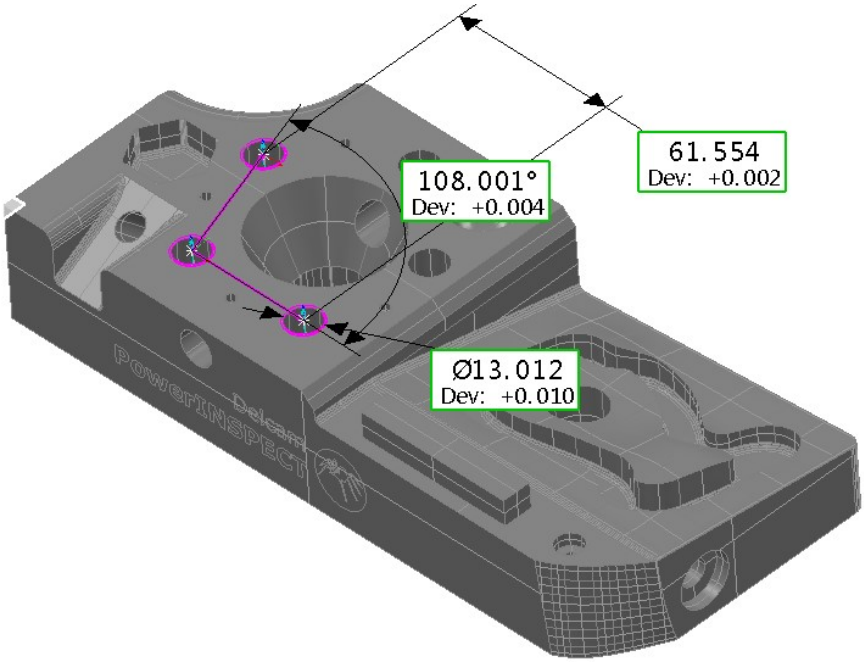
Customising labels and leaders

After you have positioned the dimension labels, they can be hidden from the CAD view or removed completely. Right-click a label and select **Hide** to remove it from the CAD view, or select **Delete** to remove it from the CAD view and the inspection sequence.

You can further customise the labels by selecting the **Graphical Dimensions** page in the **Options** dialog.



In the following example, the **Arrow head size** was increased and **Display deviation** was selected.



8. Inspection Exercise

The following example creates a simple inspection sequence. The sequence includes various features shown in previous chapters, including a **PLP** alignment, geometric feature measurement, surface points, dynamic points, and section inspection.

Using the methods, previously described in the course, for starting a new session.

1 Click **New Document Wizard**  to open the **New Inspection Session** dialog.

2 Select **Measurement with a single CAD Part** and click **Next**.

3 Click **New** and browse to **DemoBlock2008(CMM+Arm).dgk**.

4 Leave the **Offset** and **Tolerance** settings unchanged, and click **Next.**

5 In the **Variables** dialog. browse for any **HTML Report Template** (**Excel** could be used, but for this example HTML is to be used). The variables in the report are displayed in the table.

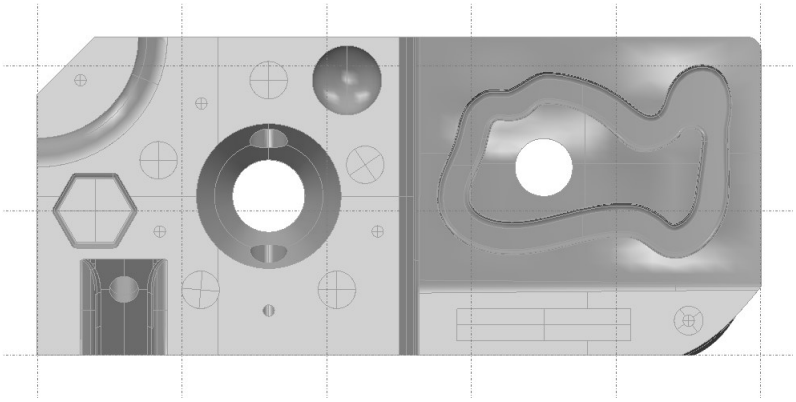
6 Click **Finish**.

7 From the **CAD View Toolbar**, click **Show shaded and wireframe**  to shade the model.

8 Click **View from Top (Z)**  to re-orientate the model.

9 Click **Delay Measure**  .

The new session is now ready for programming.



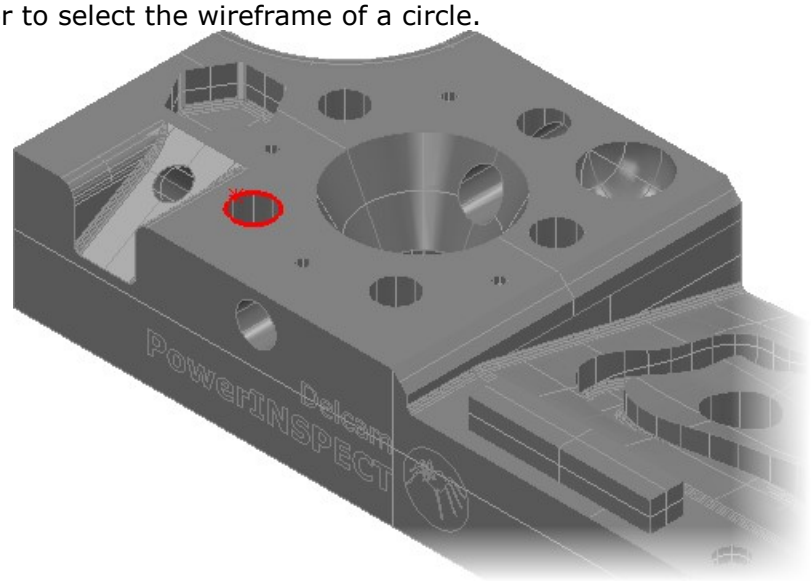
8.1 Geometric PLP Alignment

The part will be aligned using a Geometric PLP alignment.

1 From the **Mouse Context** toolbar, select **Wireframe Checker**  .

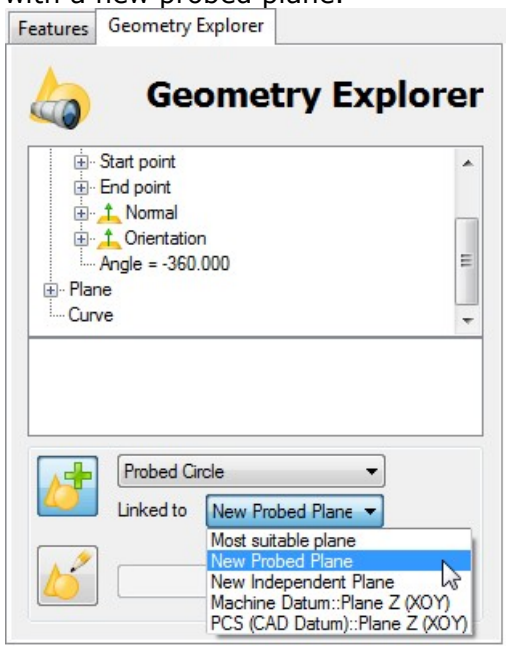
2 Right-click an empty area of the CAD view and select **Wireframe** from the popup menu.

3 Use the cursor to select the wireframe of a circle.



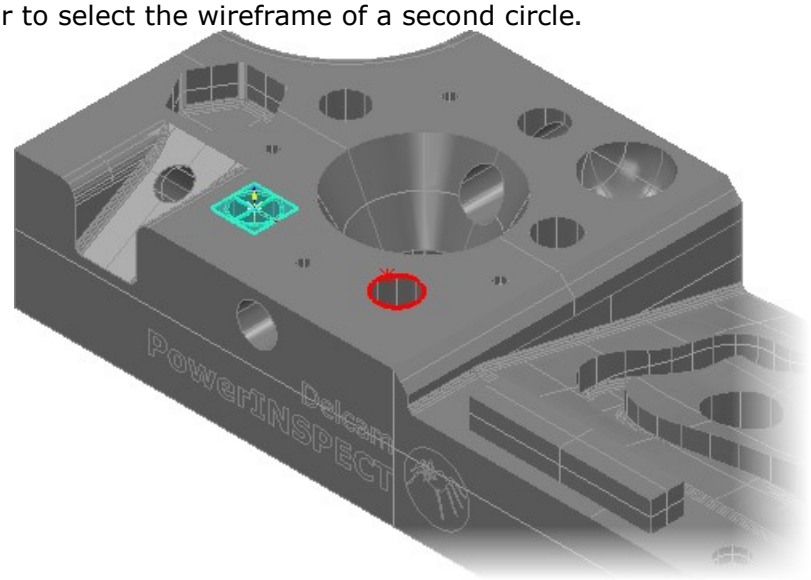
The details of the circle are displayed in the **Geometry Explorer** tab below the Sequence Tree.

4 In the **Geometry Explorer** tab, select **New Probed Plane** from the **Linked to** drop-down list. This associates the circle with a new probed plane.



5 Click  to add the circle and plane items to the inspection sequence.

6 Use the cursor to select the wireframe of a second circle.

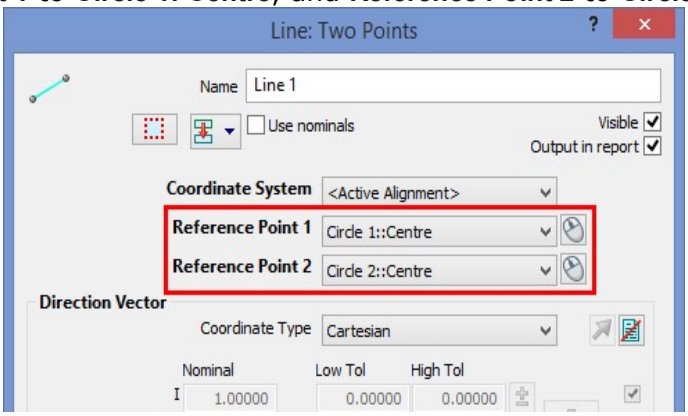


7 Select **Plane 1** from the **Linked to** drop-down list.

8 Click  to add the second circle item to the inspection sequence.

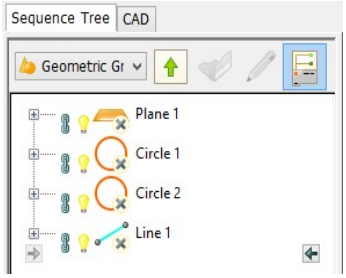
9 Click **Line: Two Points**  on the **Lines** flyout  .

10 Set **Reference Point 1** to **Circle 1: Centre**, and **Reference Point 2** to **Circle 2: Centre**.



11 Click **OK**.

The **Line** has now been created and the Sequence Tree has been updated to include all the geometric items.

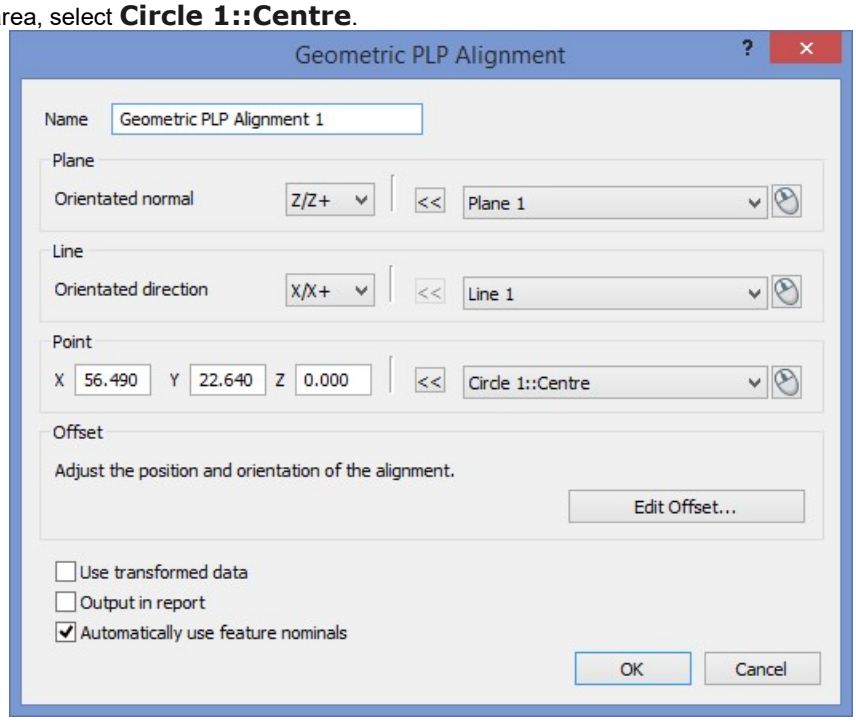


12 Click **Up One Level**  to close the geometric group.

13 Click **Alignments**  , and select **Geometric PLP Alignment**  . The **Geometric PLP Alignment** dialog is displayed.

14 Use the dialog to set the reference items for the Geometric PLP alignment:

- In the **Plane** area, select **Plane 1**.
- In the **Line** area, select **Line 1**.
- In the **Point** area, select **Circle 1::Centre**.



15 Click **OK**.

8.2 Geometric Feature Inspection

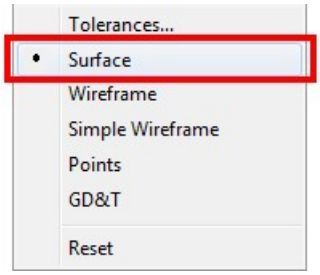
Some 2D/3D Geometric Features will now be selected to begin the inspection sequence.

1 Click **Geometric Group**  on the **Item** toolbar. Leave the default settings unchanged and click **OK**.

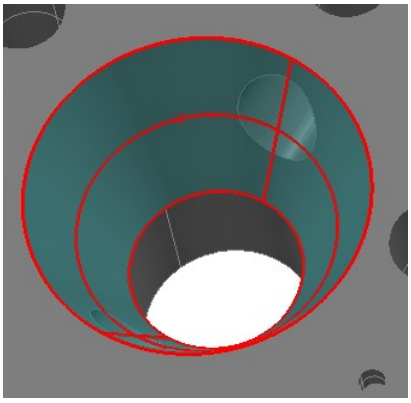
In this group, the Slot and Cone will be measured.

2 Select **Wireframe Checker**  .

3 The cone is a surface, so the Wireframe Checker needs to be in surface selection mode. Right-click in an empty space in the CAD view and choose **Surface** from the popup menu.



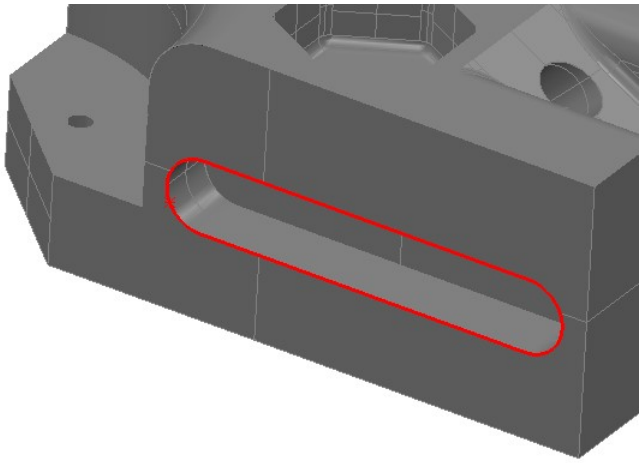
4 Select the cone in the centre of the model, and click  to add the cone item to the inspection sequence.



The slot is a Wireframe item so the **Wireframe Checker** needs to be returned to the Wireframe selection mode.

5 Right-click in an empty space in the CAD view and choose **Wireframe** from the popup menu.

6 Select the rounded slot on the side of the block.



7 In the **Geometry Explorer** tab, select **New probed plane** in the **Linked** to drop-down list.

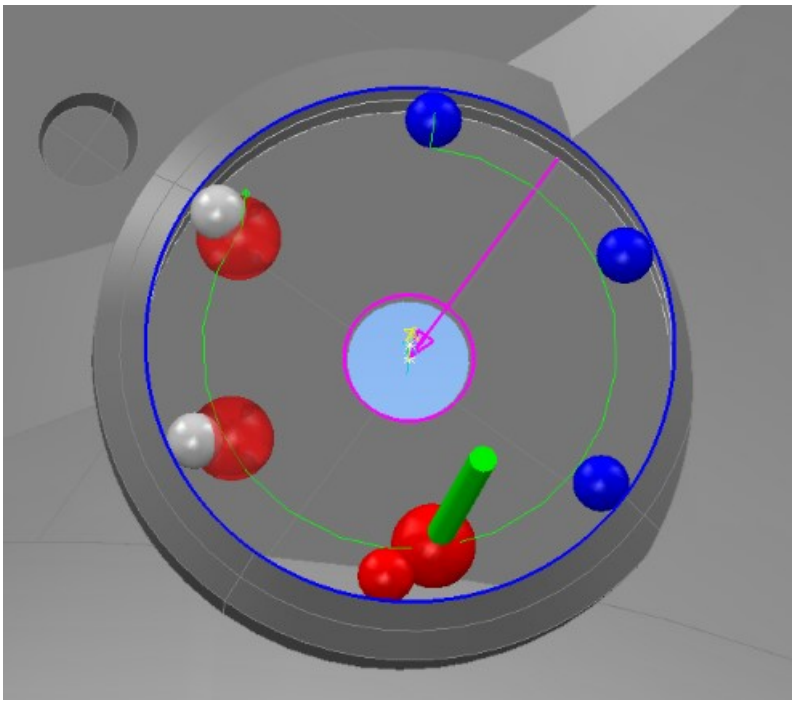
8 Click  to add the slot and plane items to the inspection sequence.

9 Click  to close the geometric group.

8.3 Bouncing Ball

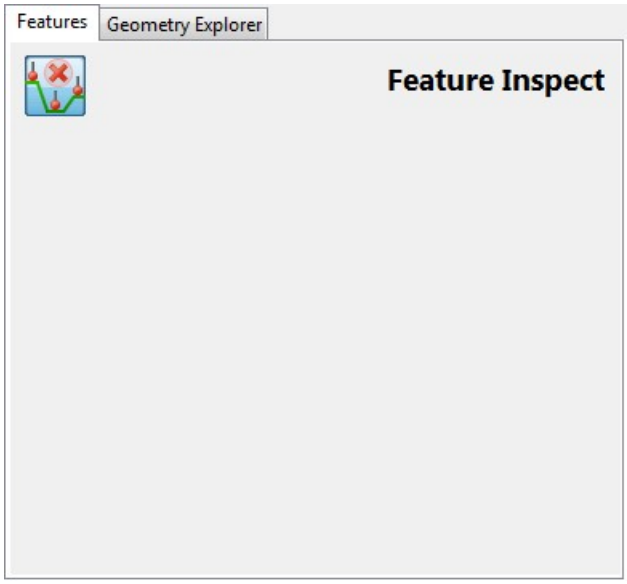
Bouncing ball is an on-screen guide which shows you what has been measured and what is to be measured next. Points are displayed as spheres in different colours to guide the user during the inspection. The following colours are used by default.

- **Blue**: Unmeasured point.
- **Red**: Next point to be measured.
- **Grey**: measured point.
- **Transparent Red**: Actual position of point measured.



To illustrate this functionality, the bouncing ball will be applied to the two geometric features defined above.

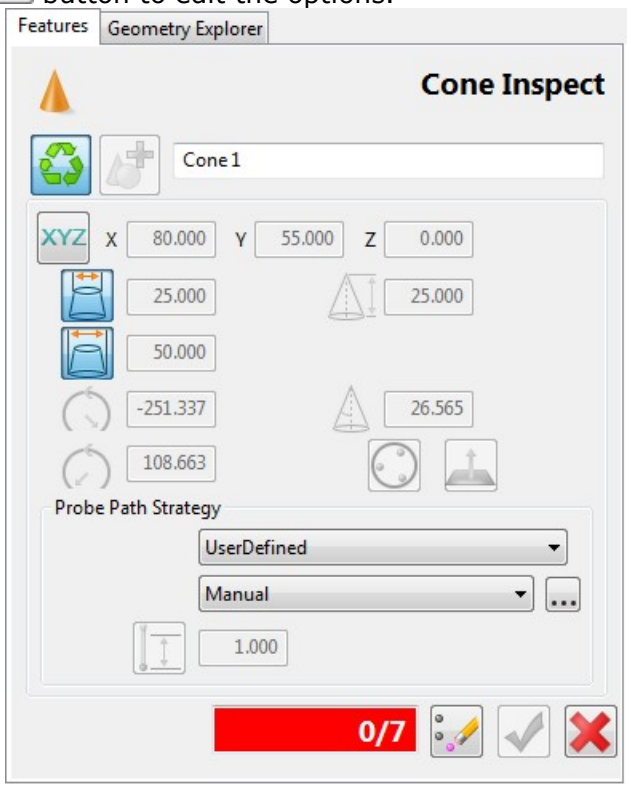
1. Open **Geometric Group 2** in the Sequence Tree to display both geometric features.
2. Select the **Features** tab at the bottom of the Sequence Tree.



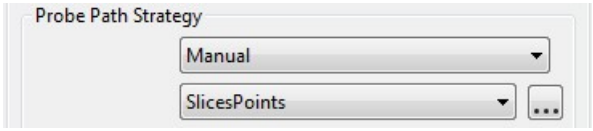
3 Bouncing ball is disabled by default. Click the **Probe path generator** button to enable it. The button changes to .

4 Select **Cone 1** in the Sequence Tree. Its details are displayed in the **Features** tab.

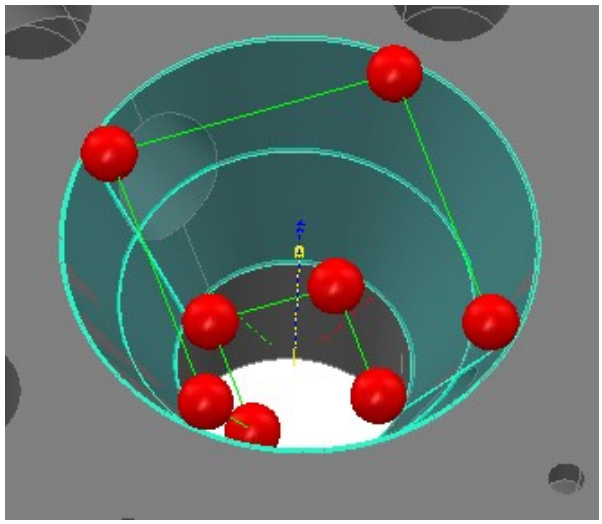
5 Click the **Recycle** button to edit the options.



6 In the **Probe path strategy** area, select **Manual** in the top drop-down list (strategy), and select **Slices points** in the bottom drop-down list (method).



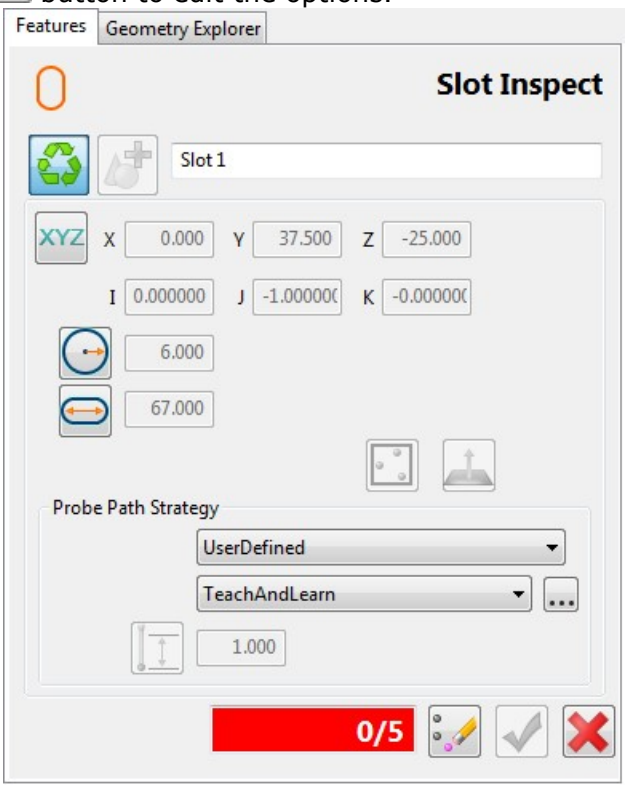
The cone displays the probe points as red spheres. Green lines/arcs show the shortest distance between the points.



7 Select  to accept and save the geometric feature.

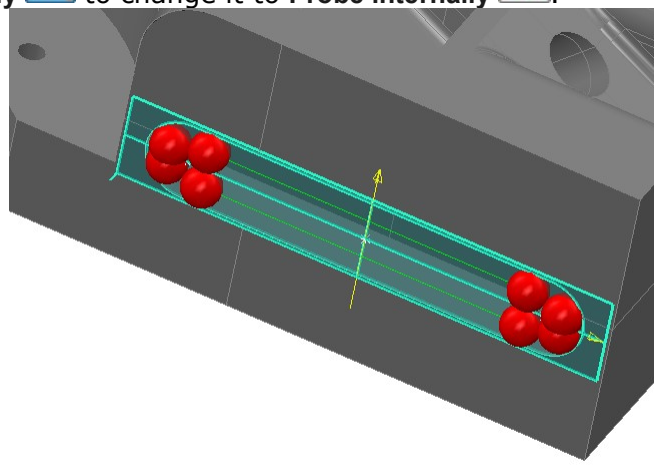
8 Click **Slot 1** in the Sequence Tree. Its details are displayed in the **Features** tab.

9 Click the **Recycle**  button to edit the options.



10 In the **Probe path strategy** area, select **Manual** in the top drop-down list (strategy), and select **SlicesPoints** in the bottom drop-down list (method).

11 Click **Probe externally**  to change it to **Probe internally**  .



12 Click  to accept and save the geometric feature.

13 Click  to close the **Slot Inspect** page.
The Bouncing Ball feature is now applied to the two geometric features. This will be activated in the Sequence Tree during Play mode.

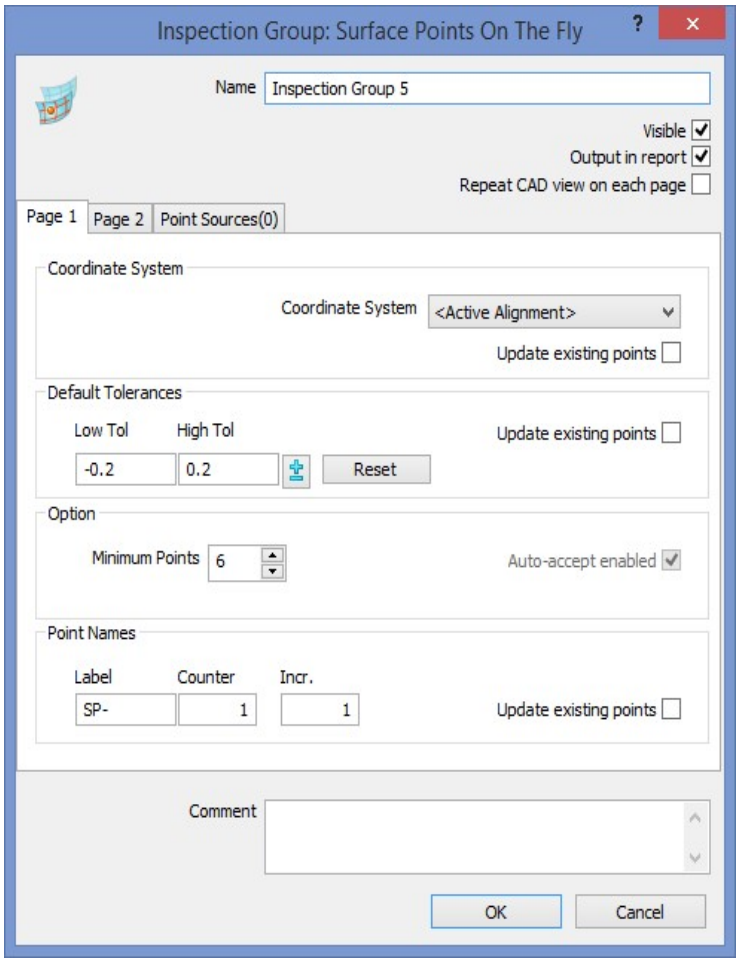
14 Click the **Probe path generator** button to disable the Bouncing Ball feature. The button changes to  .

15 Click  to close the geometric group.

8.4 Surface Inspection

The next stage is to perform a surface inspection.

1 Click the **Inspection Groups** icon  and then select the **On the fly Surface Points** icon  .



2 Set the **Minimum points** as **6.**

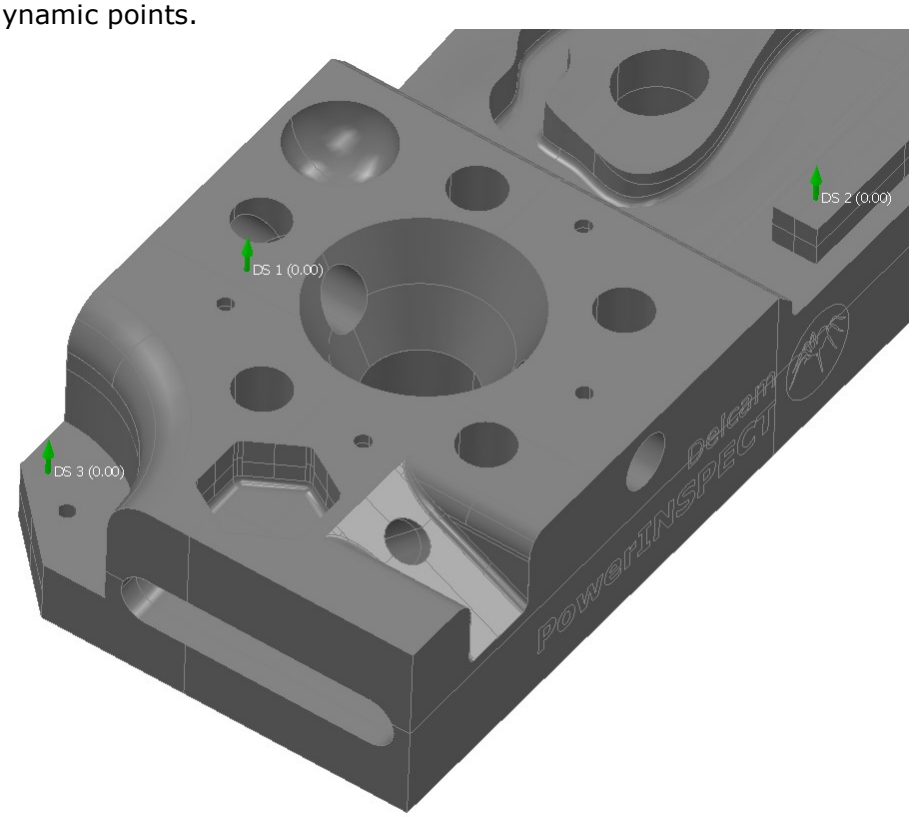
3 Click **OK.**
The inspection group has been added to the inspection sequence.

8.5 Guided Surface Inspection

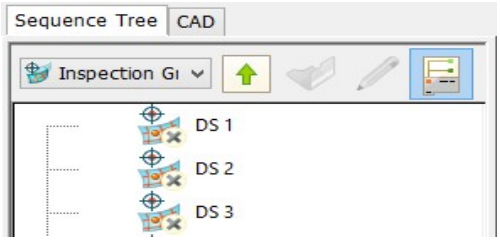
This example creates another surface inspection but instead using guided points, the **Dynamic Points Editor** is used to specify the points to be probed.

1 Select the **Dynamic Points Editor**  from the right **Mouse Context** toolbar.

2 Double-click on the model surface approximately in the areas shown below, to specify the three dynamic points.



3 Click **Create inspection points**  from the Dynamic Point toolbar.
In the Sequence Tree, an inspection group is created including the three dynamic points.



4 Click **Up One Level**  to close the inspection group.

5 Click **Dynamic Points Editor**  to deselect it.

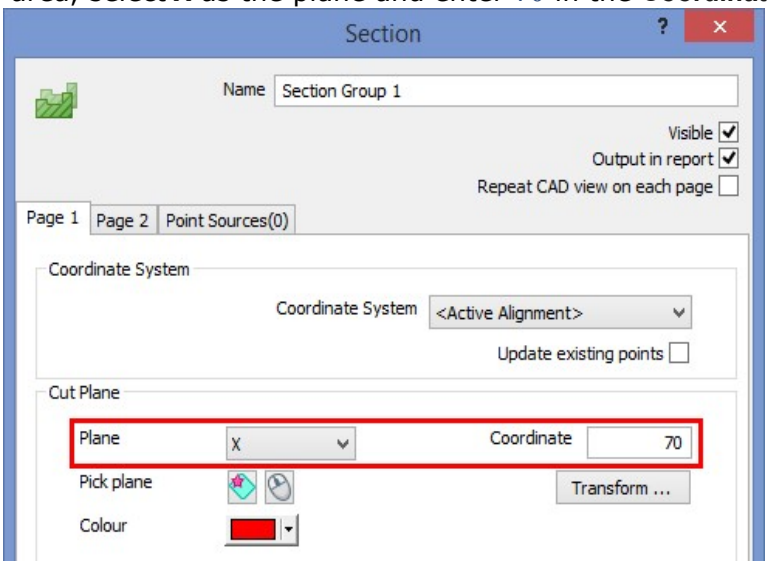
8.6 Section Group Inspection

Finally, a **Section Inspection** will be included in the inspection sequence.

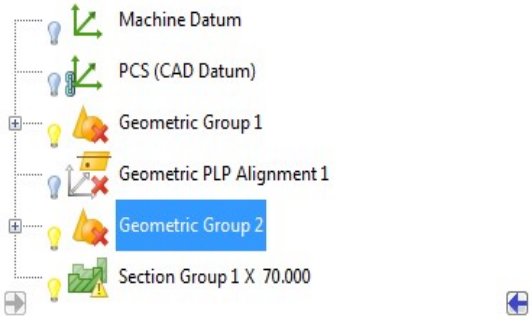
1 Click **View from Top (Z)**  to reorientate the view down Z.

2 From the **Item** toolbar click the **Section Group**  button. The **Section** dialog is displayed.

3 In the **Cut plane** area, select **X** as the plane and enter 70 in the **Coordinate** box.



4 Leave the other options unchanged, and click **OK**.
The inspection sequence is now complete and the inspection sequence contains all the items you created.



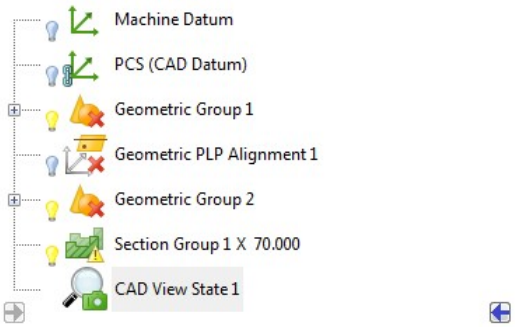
Two extra features (inspection aids) will be added to the inspection sequence to assist the inspector.

8.7 CAD View State

CAD View State items allow you to save the current CAD view (orientation and zoom) and add it to the inspection sequence.

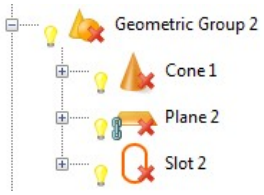
During the inspection, the CAD view item will load automatically, allowing the inspector to continue probing without having to pause and manipulate the CAD model manually, thus speeding up the inspection process.

1 Zoom and orientate to a suitable view around the slot feature, then select **CAD View State**  from the **Miscellaneous**  toolbar.

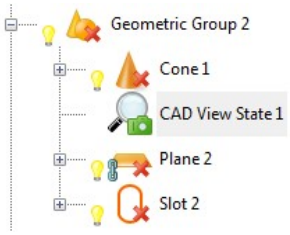


The **CAD View State** item is added to the bottom of the sequence.

2 Expand **Geometric Group 2** by clicking the ⊞ symbol adjacent to the name, to show the three geometric features.



3 Select the new **CAD View State** item. Keep the mouse button pressed, and drag and drop the CAD view below Cone 1 feature.



The view is now placed just before the Slot feature, displaying the view just prior to the inspector probing the feature.

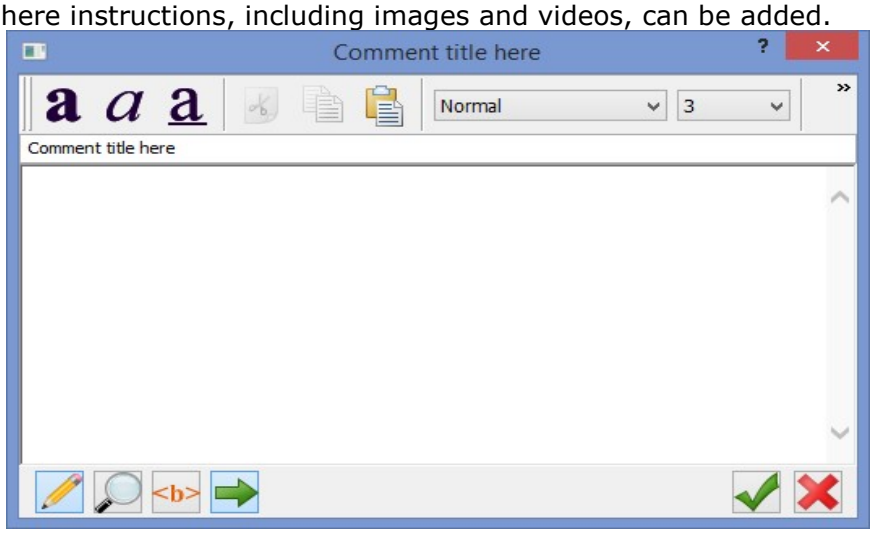
*Dragging the **CAD View State** item would not be required if the insertion cursor*  *was positioned here initially.*

8.8 Comment

Another method of assisting the operator during inspection is to include a **Comment** in the inspection sequence.

As with **CAD View State** items, **Comment** items should be placed into the inspection sequence at the point where the information needs to be seen.

1 Select **Comment**  from the **Miscellaneous**  toolbar. The **Comment** dialog is displayed where instructions, including images and videos, can be added.



2 Enter the following text.
Probe 10 points in and around the sphere feature.

3 Select  to save your changes. The stored comment is added to the end of the sequence.

4 Drag and drop the new **Comment** item below **Geometric Group 2**. The comment will display just before the surface inspection.
The sequence is now complete and ready for inspection.

5 Click **Play All**  on the **Main** toolbar.
The screen changes to Full Screen measure mode, indicating what needs to be measured and the minimum number of points required.

6 Measure all the requested points.

7 Select the **Report** tab to check the measurement results.

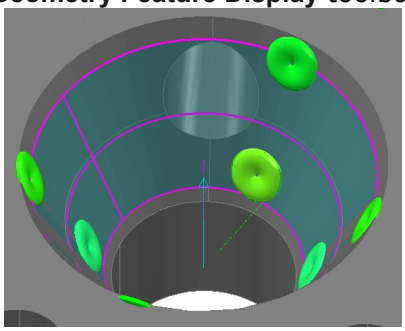
8.9 Play Resume

This feature allows you to measure more probed points for the selected element.

1 Expand **Geometric group 2**.

2 Highlight and zoom into the cone feature.

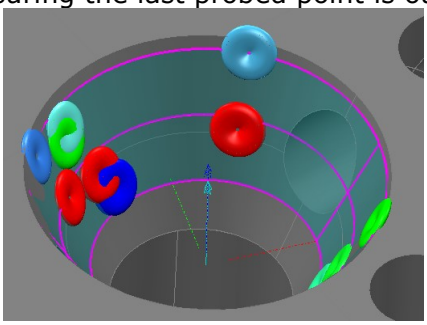
3 Select **Discs**  from the **Geometry Feature Display** toolbar.



Further measured points will now be added to the cone feature. PowerInspect will include these new points and then recalculate the cone.
A deliberate bad point will be included in this measure to illustrate the ability of PowerInspect to choose, select, and discard points afterwards. This enables you to look for rogue points and to discard or replace them before creating the report.

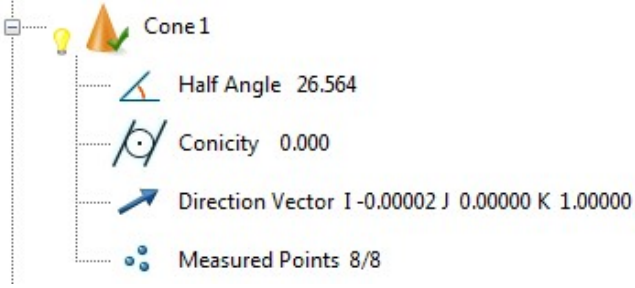
4 Select the **Play with Resume**  button from the **Main** toolbar.

5 Probe a further 5 points, ensuring the last probed point is out-of-tolerance.

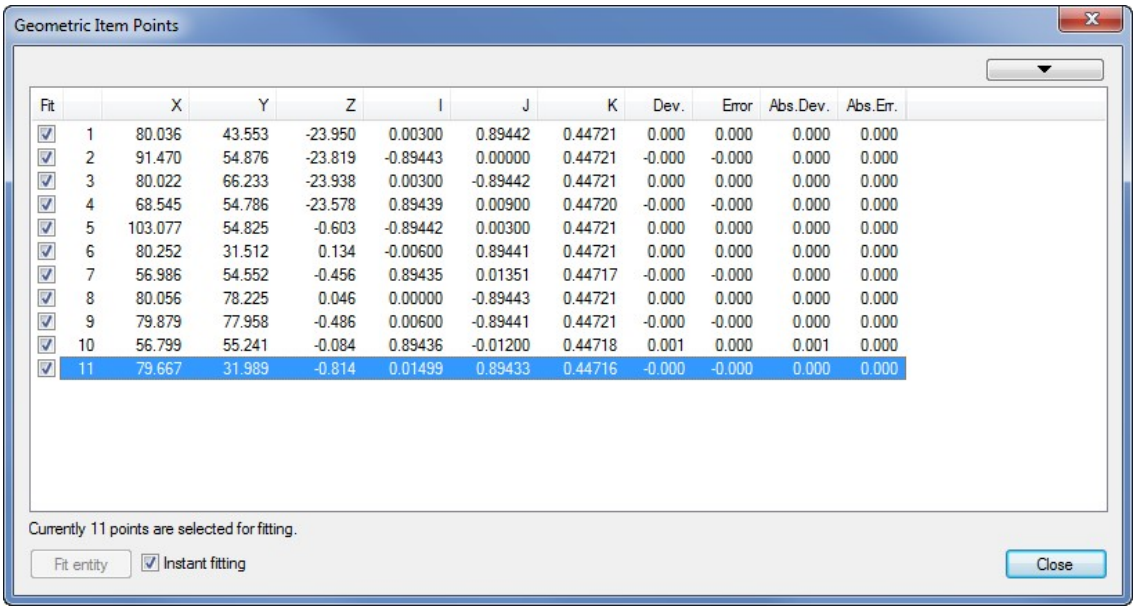


Every additional point taken forces PowerInspect to recalculate/refit the cone. The image above shows the refitted cone to include all original and the new 5 points taken.
The out-of-tolerance point taken has forced the cone to be out-of-tolerance. This rogue point will now be excluded.

6 Expand the cone feature  in the Sequence Tree

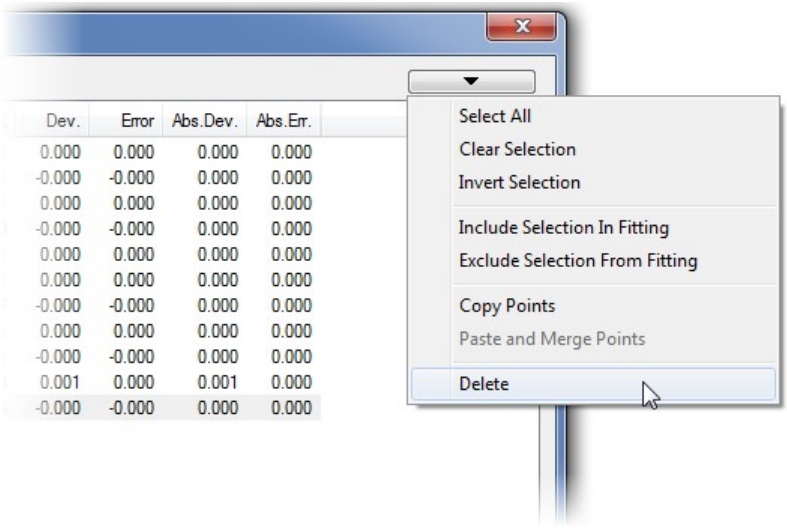


7 Double click on **Measured points 11/11.**



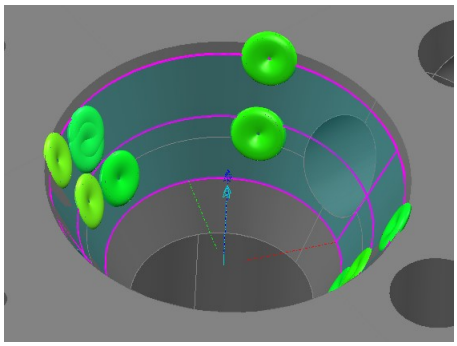
All the measured points are displayed. Point 11 is the rogue point. This point will now be excluded from the list.

8 Highlight point 11 and select > **Delete.**



The point has been deleted permanently, and the cone has been recalculated without it.

The cone is now within specification as shown below.



Points do not have to be deleted to exclude them from the fitting calculation.

You can deselect the check box next to a point to exclude its point from the calculation without losing it. This has the benefit of providing an audit trail for all measurements.

8.10 Custom Actions

PowerInspect includes three buttons that enable you to run conditional tests and custom applications as part of the inspection sequence.

You can add the custom actions to the **Definition** level and within a geometric group from the **Miscellaneous**  toolbar.

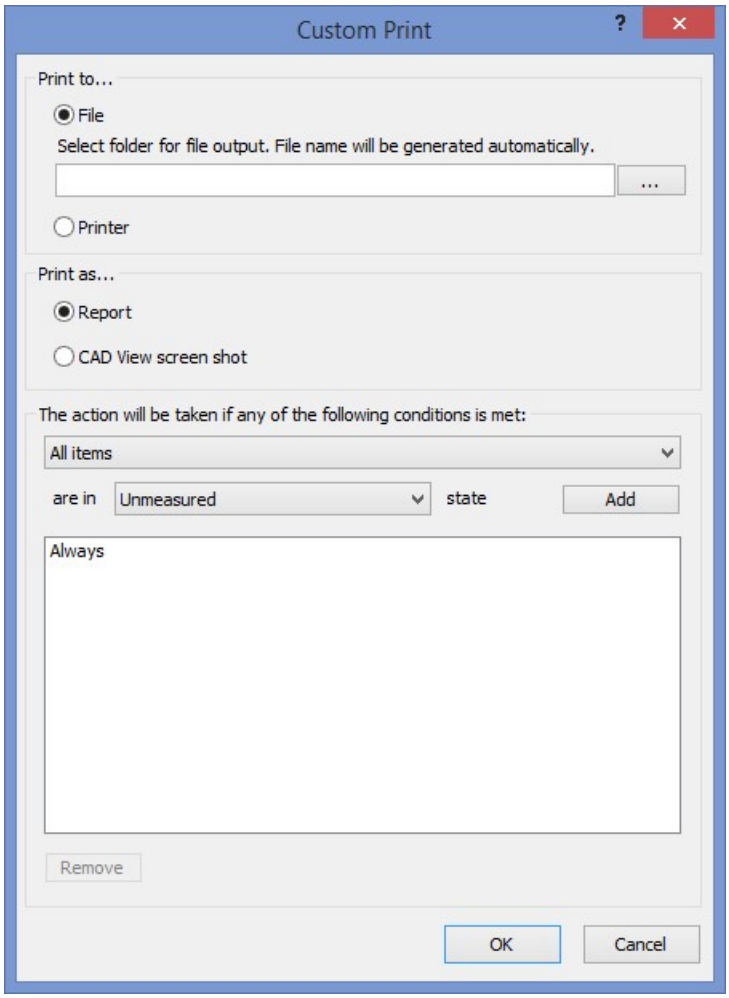
Custom Stop Replay

Use **Custom Stop Continue**  items to automatically stop the inspection sequence if any of the specified conditions are met. For example, a **Custom Stop Continue** item can be used to automatically stop the inspection if an element's measurement is out-of-tolerance.

When the inspection sequence is played, processing stops and a message is displayed if any specified condition is met.

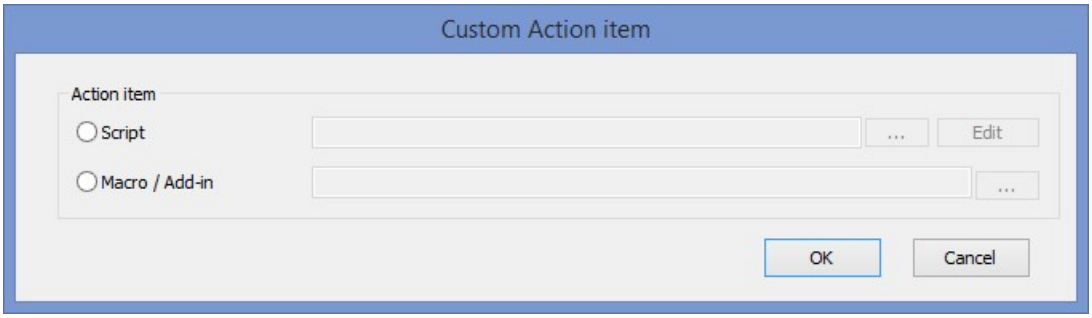
Custom Print

Use **Custom Print**  items to automatically print a report if any of the specified conditions are met. For example, a **Custom Print** item can be used to output results only when an item is unmeasured or out of tolerance.



Custom Action

Use **Custom Action**  items to automatically run an internal script or macro during the inspection sequence. For example, a **Custom action** item can be used to automate further processing or archiving of your measurement data.



9. Modifying Elements

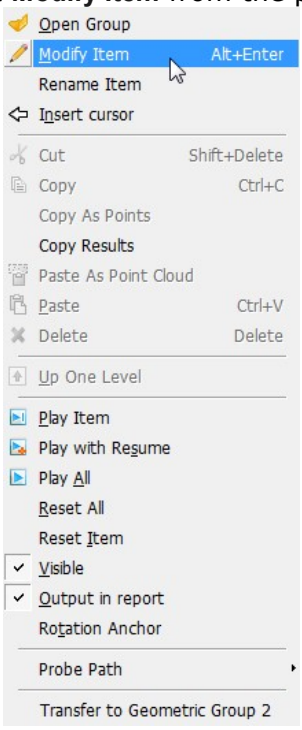
Many elements created within PowerInspect can be modified in one way or another. Modifications can be made to items, probed points and even PowerInspect itself.

This chapter briefly goes through the deletion and reordering of points, and some display modifications.

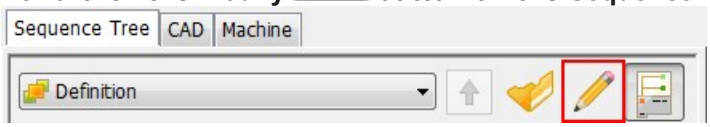
9.1 Accessing Modification

You can edit or modify items in three ways.

- Right-click an item and select **Modify Item** from the popup menu.



- Select the item and click the **Modify**  button on the **Sequence Tree** toolbar.



- Use the short-cut key combination of **Alt+Enter**.

When you modify geometric items, the dialog that specified the item's parameters is opened. This allows you to change the constraints that set the item.

For example, in the **Geometric PLP Alignment** where a line is specified using the centres of two circles, it can be altered to pass through different elements.

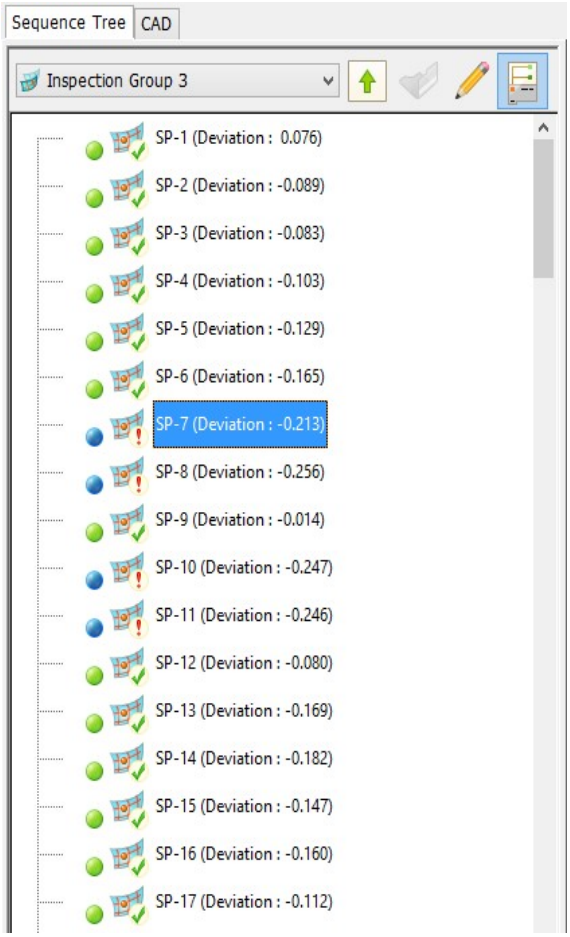
Most modifications work on similar principles, but these might affect the results. The next example shows how to delete an unwanted result, and demonstrates how to rectify the point numbers to maintain a continuous set of results.

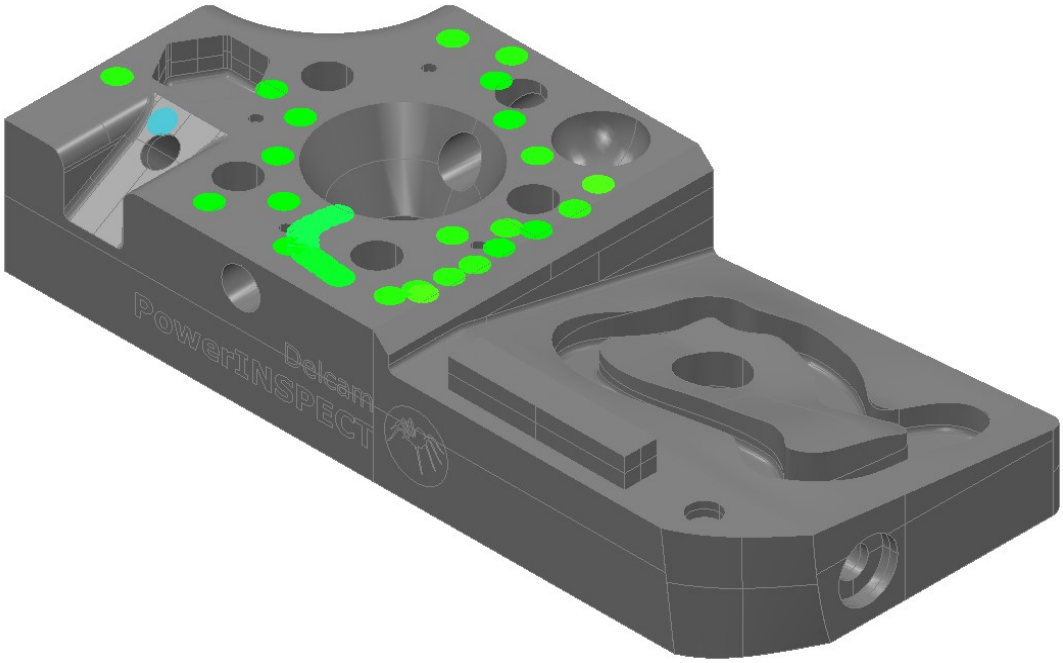
9.2 Anomalous Result Example

When probing a point, it is possible to create an unwanted or anomalous result by probing too hard, taking too many points, taking points in the wrong place, false triggering and so on. These results could affect a line of best fit or an inspection report, for example, so it is often best practice to delete them.

In this example, the DemoBlock2008 part was probed and during the inspection an anomalous result was taken. This anomaly was verified by taking a point at the same position and comparing the results. As can be seen from the Sequence Tree, the deviation of the anomaly is much greater than that of the surrounding results, so it should be removed from the group.

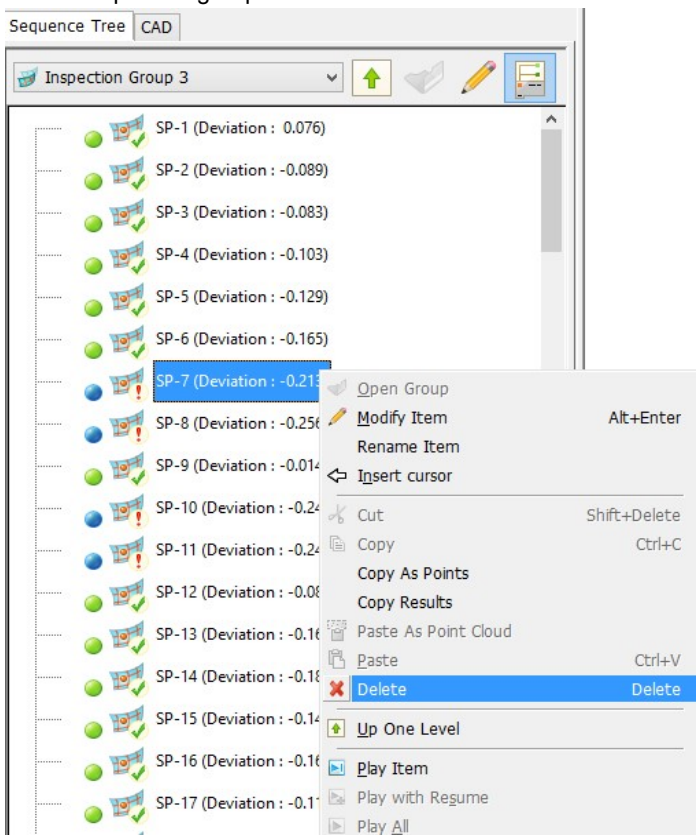
The inspection group is open to display the points.



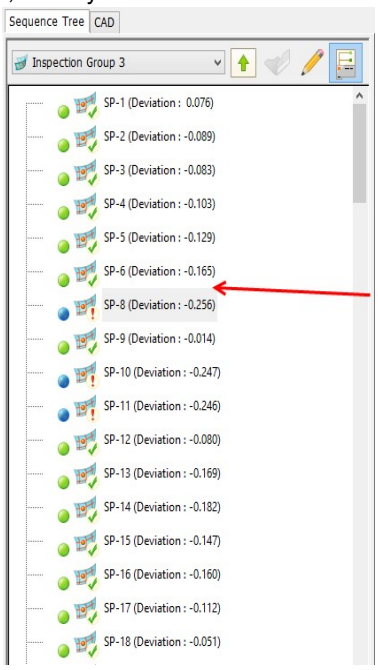


1 Click  to open the affected group.

2 Right-click on the anomalous result (in this case **SP7**), and click **Delete** on the popup menu.
The point is removed from the inspection group.

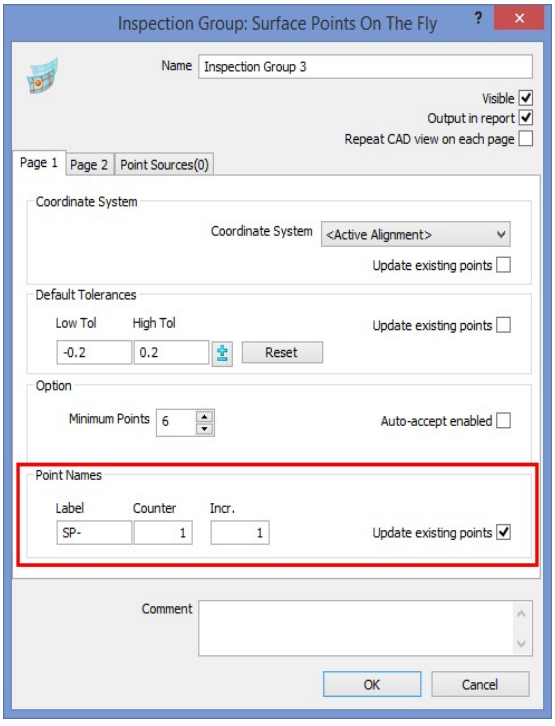


This leaves a gap in the point numbers, so they need to be reordered.



3 Click **Move Up One Level**  . The numbering of the surface inspection group can now be modified.

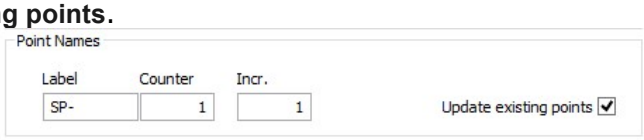
4 Select **Inspection Group 2** and click the **Modify**  button. The **Surface Inspection Group** dialog is displayed which allows you to reset the creation parameters.
To reset the numbers for an inspection the following changes need to be made in the **Item Naming Rules** area.



5 Enter a **Counter** value of **1**.

6 Enter an **Incr.** (increment) value of **1**.

7 Select **Update existing points**.

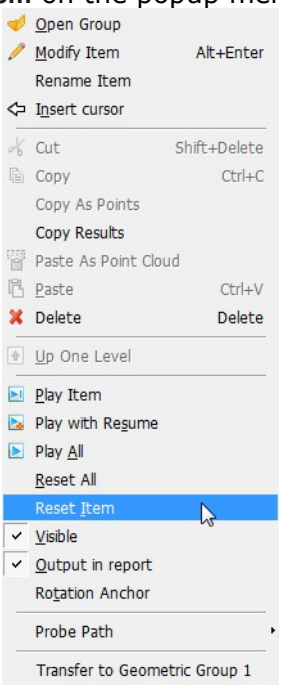


8 Click **OK**.
The points are updated in the Sequence Tree; points **SP11** and **SP12** have been updated to become points **SP10** and **SP11** respectively.

9.3 Replaying Probed Items

Individual features and points can be replayed.

1 Right-click **SP10** and click **Reset Item** on the popup menu.



The point is reset and the icon displayed in the Sequence Tree changes.

*Selecting **Reset All** resets all the measures in the group.*

2 Select the point and then click **Play Item**  . The point is shown as a dynamic point  in the CAD view.

3 Probe the point.
The Sequence Tree and report are updated with the new measure.

10. Exporting to CAD

If you do not have a CAD model of a part, you can use PowerInspect to take measurements which can then be exported to CAD and used for reverse engineering with a CAD system such as PowerShape.

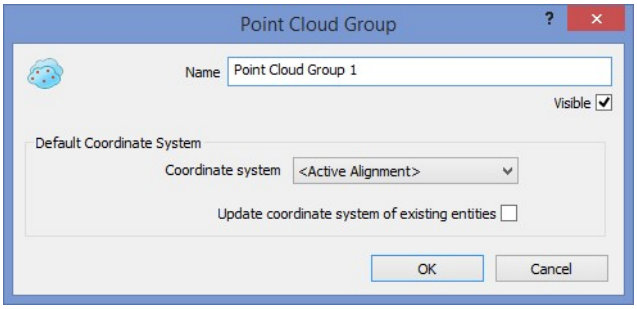
10.1 Point Cloud

Point cloud data (3D digitised data) can be obtained in two ways.

- **Contact** — Using devices such as touch probes attached to portable arms or CMM machines. At point of contact, the X, Y and Z coordinates are taken.
- **Non-contact** — Using devices such as line or scanning point lasers.

To include point cloud data in an inspection sequence.

1 Click the **Point Cloud Group**  button on the **Item** toolbar to create a point cloud group.



2 Leave the settings unchanged, and click **OK**. The **Item** toolbar is replaced by the **Point Cloud** toolbar.

Select an option from the toolbar which is appropriate for the action you want to perform:

 **Create a point cloud by probing points on the fly**. Use this option to probe points on the part and save them as a point cloud. You can also use this option to delete previously probed points so you can reprobe them. If you have a CAD model, you can project the point cloud on to it to view any deviation.

 **Curve flyout**. This flyout contains three options:

 to probe points on the part and fit a curve through them. The curve is saved in the point cloud group and can be exported for use in CAD applications.

 to create curves from sections through a CAD model. These curves can be used to create reference lines for marking out a clay model.

 to import a digitised curve from IGES or VDA files. These curves can be used to create reference lines for marking out a clay model.

 **Import a cloud of points from a file**. Use this option to import points that have been probed using a different application. The cloud of points is imported as a single element in the point cloud group and can then be projected onto the model to find the deviation between the probed points and your CAD data.

 **Miscellaneous** toolbar. This flyout contains two options:

 to add a comment.

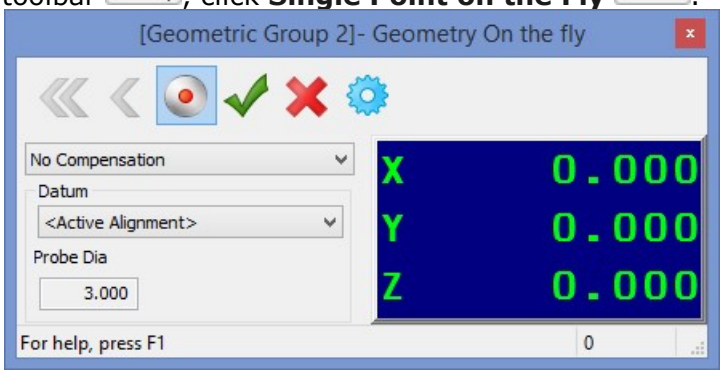
 to add a CAD View Report to the Point Cloud group.

Single Points on the fly

Single points can be taken as geometric items anywhere on the part. Single points on-the-fly are the same as guided single points, except that a nominal value does not have to be entered.

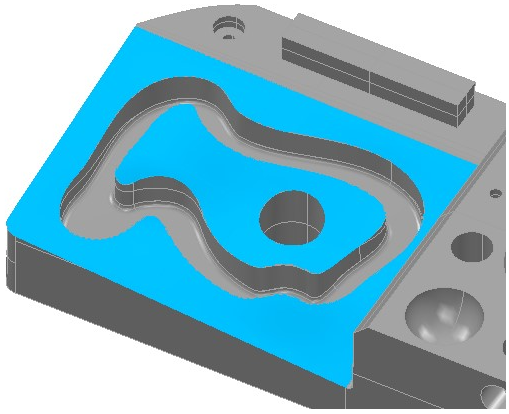
1 Click  to create a new Geometric Group.

2 From the **Points** toolbar  , click **Single Point on the Fly**  .

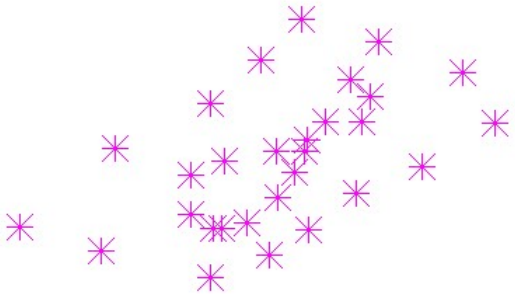


*The **Compensation** drop-down list controls whether probe compensation is required on the points taken.*

3 Probe a number of points on the sloping surface on the block (as shown in blue).



4 Click  to save the points. The points are displayed in the CAD view.



10.2 Digitised curves

Digitised curves enable you to probe a curve across the surfaces of a part. Use them to measure sections around a part and to build up a picture of a part for use in reverse engineering.

In addition to the ability to fit a curve through points probed on a part, you can also:

- Identify edges and gaps as you probe a part.
- Manage curve items using the **Digitised Curve** dialog.
- Generate curves from CAD models.
- Import curves generated using other software packages.
- Rotate, translate, mirror, and scale curves.
- View and edit point measurements.

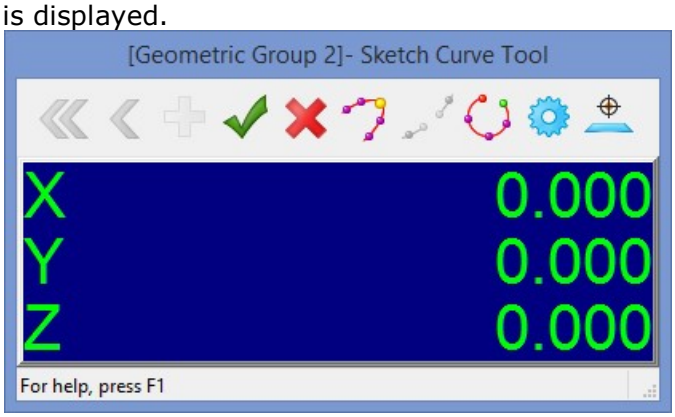
Creating a digitised curve

To create digitised curves.

1 Ensure the part is secured to the CMM bed.

2 Click  to create a new Geometric Group. Alternatively, you can create a Point Cloud group.

3 Click  to display the **Curve** flyout, then click the **Curve**  button. The **Sketch Curve Tool** dialog is displayed.



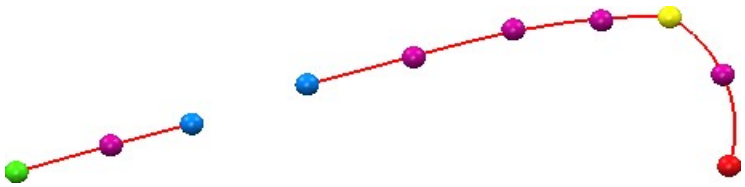
 *The dialog contains the following buttons:*

- **Reset All**  to reset all points taken for the currently active item.
- **Reset**  to reset the last probed point.
- **New Curve**  to save the probed points to a new Digitised Curve item and start a new curve.
- **OK**  to save the probed points to a new Digitised Curve item and close the dialog.

- **Cancel**  to discard any unsaved points and closed the dialog.
- **Edge point**  to identify a discontinuity, such as an edge or a corner in the surface.
- **Gap point**  to insert a gap before the next probed point.
- **Close/Open Curve**   to connect the first and last point and close the curve. Click again to open the curve.
- **Curve Sketching Options**  to display the **Curve Sketching Options** dialog. Use this dialog to change the curve fitting tolerance or use a guided plane to control probing.
- **Toggle Guided Plane**  to toggle the guided plane.

4 Probe the points on the part to create a curve. You can probe as many or as few points as you want.

5 Gaps and edges in the curve can be marked using the buttons on the **Sketch Tool** dialog, extending your ability to describe the surface of a part while it is being probed.



Probed points are colour coded depending on their function:

- Green indicates the start point of the curve.
- Red indicates the end point of the curve.
- Blue indicates a gap point.
- Yellow indicates an edge point.
- purple indicates other points in the curve.

 *You can change the point colours using the Curve page of the Options dialog.*

6 When you have finished probing the curve, click  to save the points to a new **Digitised Curve** item in the Sequence Tree.

Alternatively, click  to save the points and start a new curve.

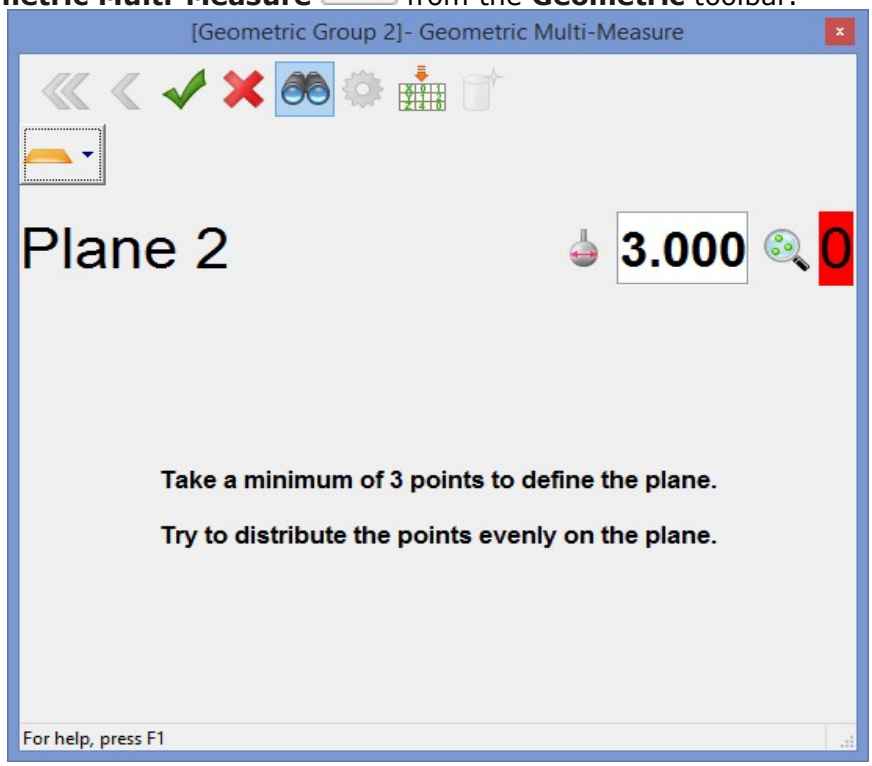
10.3 Geometric multi-measure

Geometric Multi-Measure allows you to measure multiple geometric features without having to create items first. The Multi-Measure feature is available from the **Geometric** toolbar.

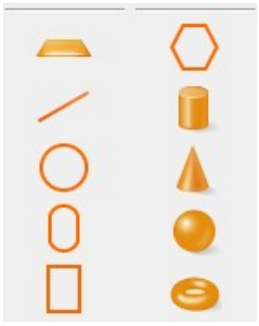
1 From the **Item** toolbar, click the **Geometric Group**  button.

2 In the **Geometric Group** dialog, leave the settings unchanged and click **OK**.

3 Select **Geometric Multi-Measure**  from the **Geometric** toolbar.



4 By default, the dialog is ready to probe plane features. To probe another type of feature, select the **Item selection**  button, then select the required feature from the selection list.

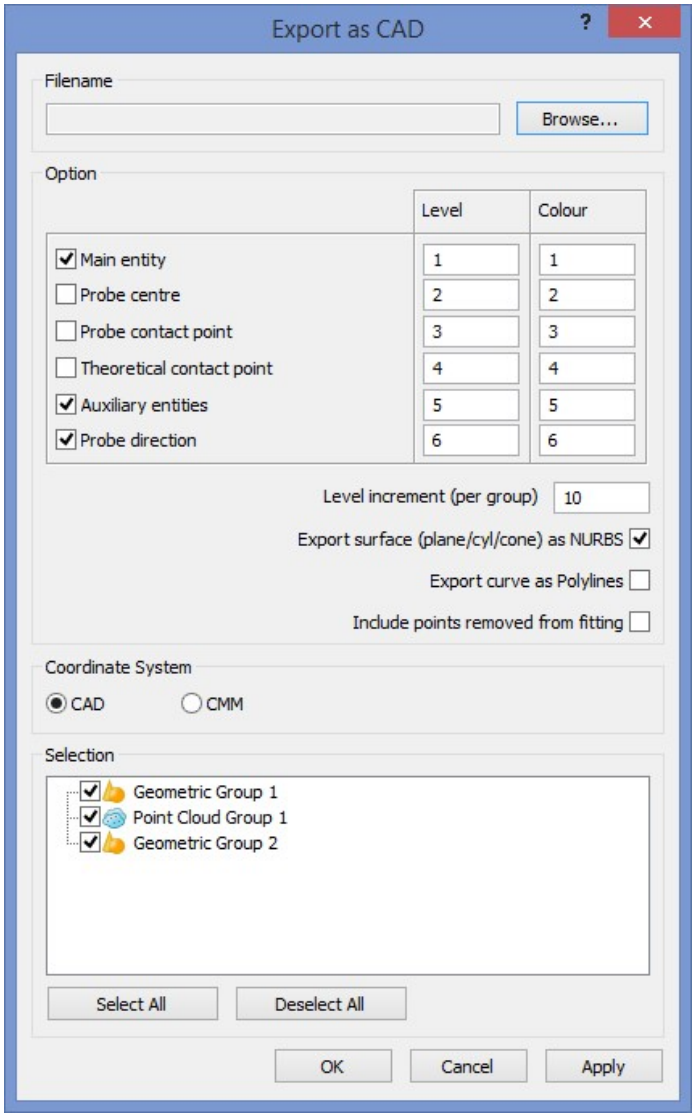


5 When you have probed all the required features, select  to close the **Multi-Measure** dialog.

10.4 File Export

When you have measured geometric items or surfaces, you can use the data as a basis for reverse engineering by exporting it in a format that can be used in modelling applications. You can export to IGES, VDA-FS 2.0, ddz, ddx and dgk formats.

1 From the **File** menu, select **Export > Export as CAD**. The **Export as CAD** dialog is displayed.



2 Click **Browse**. The **Save As** dialog is displayed.

3 In the **Save As** dialog, enter a name and choose a file type, then click **Save**.

4 In the **Options** area, select the check boxes of the attributes that you want to export.
By default, a unique identification and colour number is assigned to each attribute for use in differentiating the different types of information associated with exported items. If you do not want to differentiate between the selected attributes, enter the same number in all the **Level** and **Colour** boxes.

5 Select the **Export surface as NURBS** check box to export the data in NURBS format; deselect the check box to export 3D items in Surface of Revolution format, and 2D items as points or Ruled Surfaces.

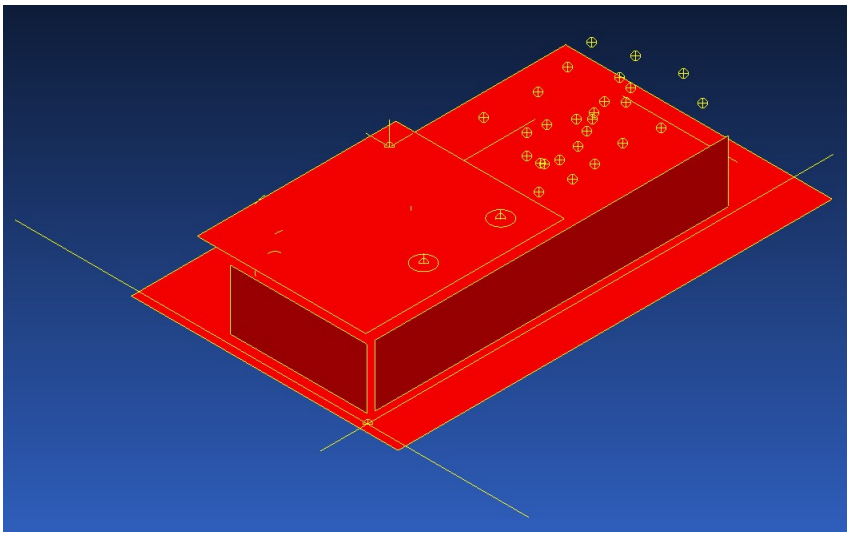
6 Select the **Export curves as polylines** check box to export curves as a series of straight lines connecting the measured points; deselect the check box to export curves in NURBS format.

7 In the **Coordinate system** area, select **CAD** to export the measurements in CAD coordinates, or select **CMM** to export the measurements in machine coordinates.

8 In the **Selection** area of the dialog, select the check boxes of the groups whose contents you want to export

9 Click **OK** to close the dialog and save the selected data to file.
The data can be now be imported into a CAD system and manipulated as required.

The picture below shows the data imported into PowerShape, where the probed planes and features are now represented as NURB surfaces.



11. Device Repositioning

The repositioning wizard in PowerINSPECT provides the ability to move the part and/or measuring device without losing the alignment. This allows you to inspect a part that is larger than the device's measuring envelope.

The wizard works by creating a new repositioning datum by specifying at least three elements. These elements can be a combination of spheres and single points. The part or device can then be relocated and the same elements measured again. PowerINSPECT matches the two positions and so can relate the new measurements to the original alignment.

This allows you to carry on inspecting the part using the same co-ordinate system.

After creating a position, it can be edited and deleted from the session. The position can also be reactivated if you are certain that inspection device is in the exact same position.

If needed the part can be moved instead of the device providing the reposition points can be probed.

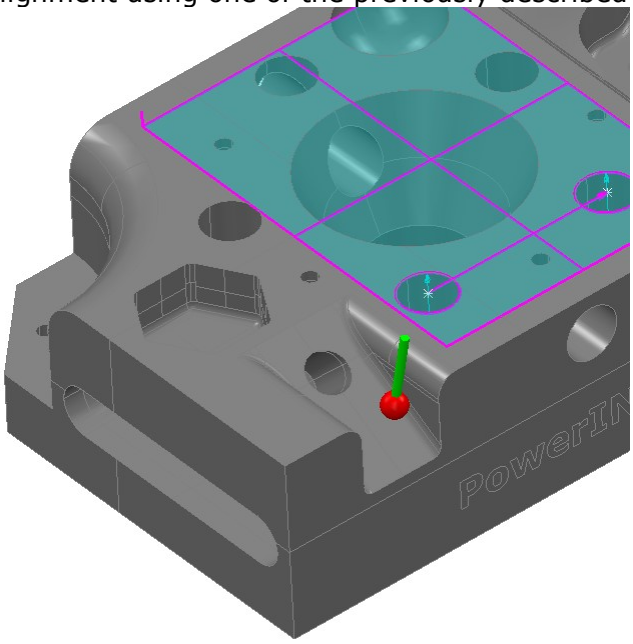
1 Click **New Document Wizard**  to open the **New Inspection Session** dialog.

2 Select **Measurement with a single CAD Part** and click **Next**.

3 Click **New** and browse to **DemoBlock2008(CMM+Arm).dgk**.

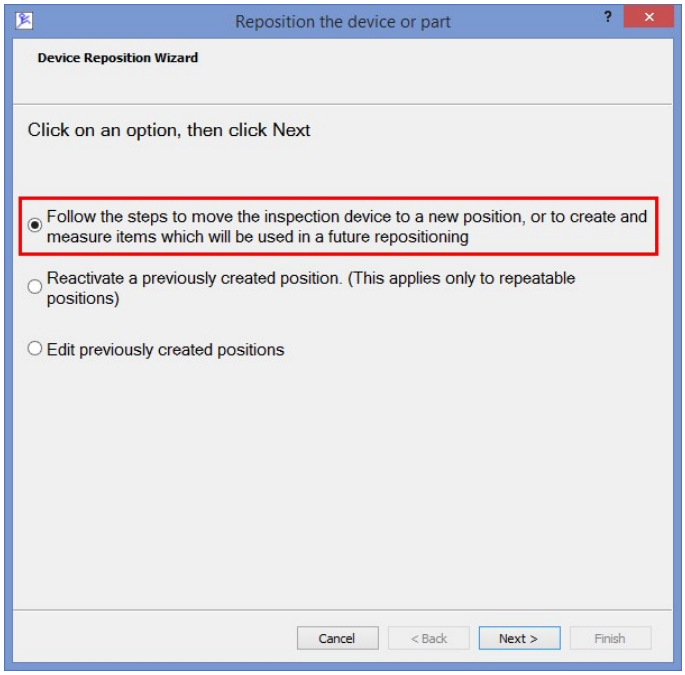
4 Leave the **Offset** and **Tolerance** settings unchanged, and click **Finish**.

5 Create and play an alignment using one of the previously described methods.

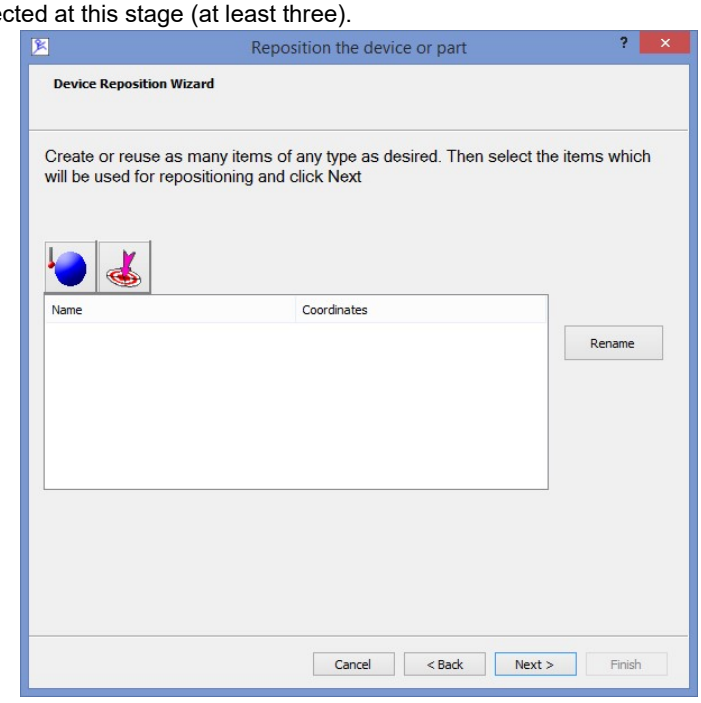


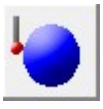
6 From the **Machine** toolbar  , click **Repositioning Wizard**  .

7 Select the first option, then click **Next**.



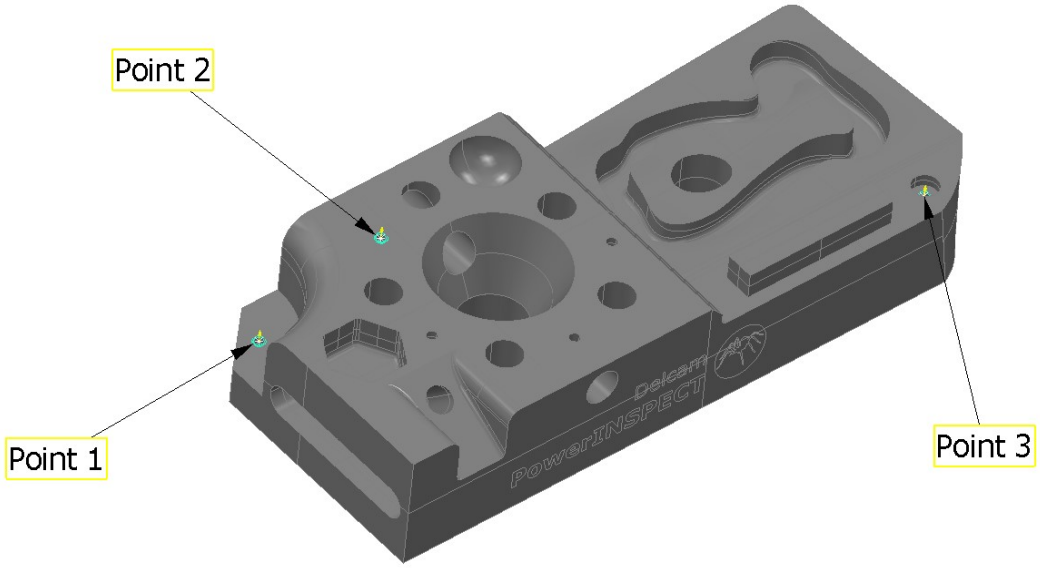
The elements are selected at this stage (at least three).



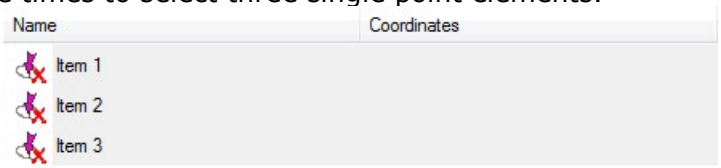
- Click  to create a sphere element.
- Click  to create a single point element

 *Only a hard probe should be used.*

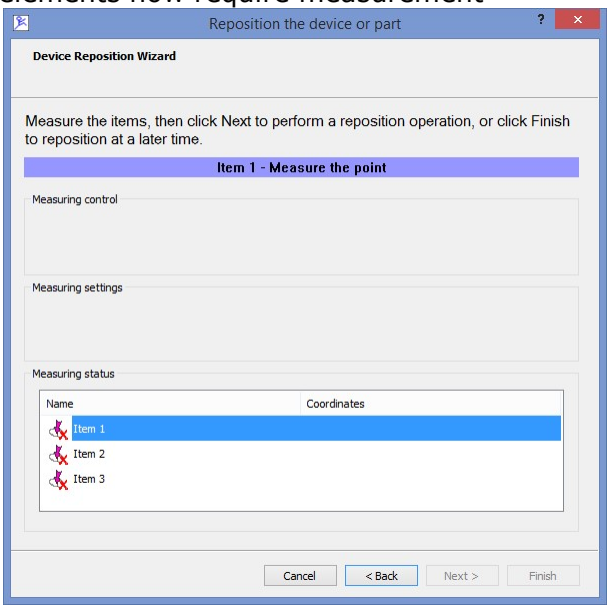
For this example, three single point elements are selected on the DemoBlock2008 part, as shown:



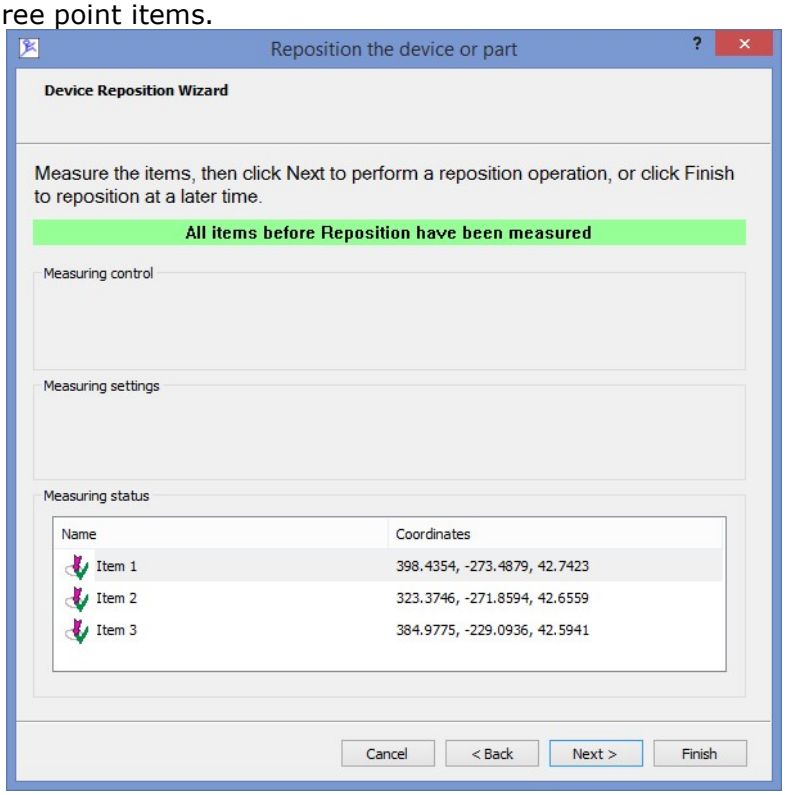
8 Click  three times to select three single point elements.



9 Click **Next.** The three elements now require measurement

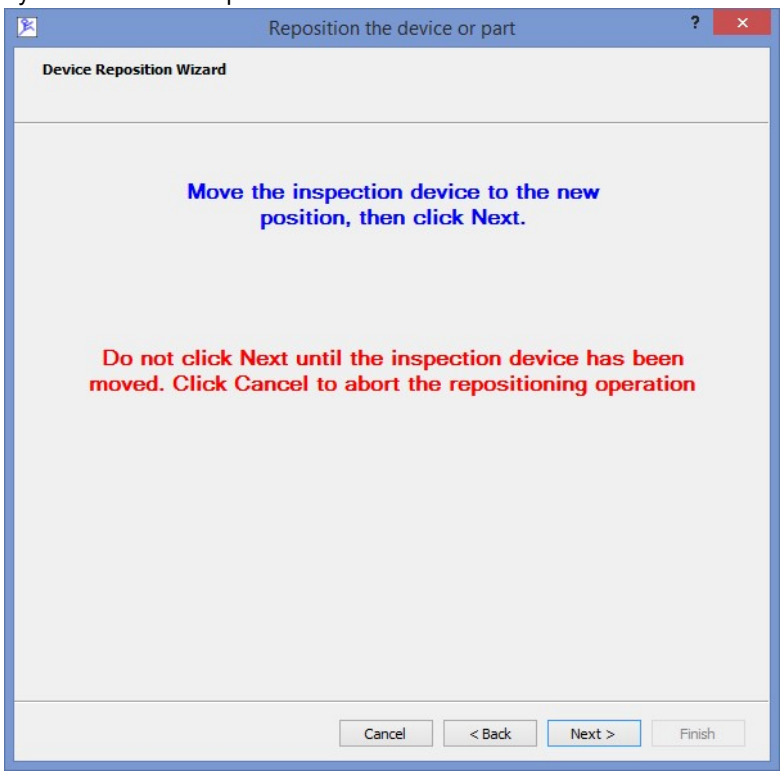


10 Measure the three point items.



The measured coordinates for the three points are displayed.

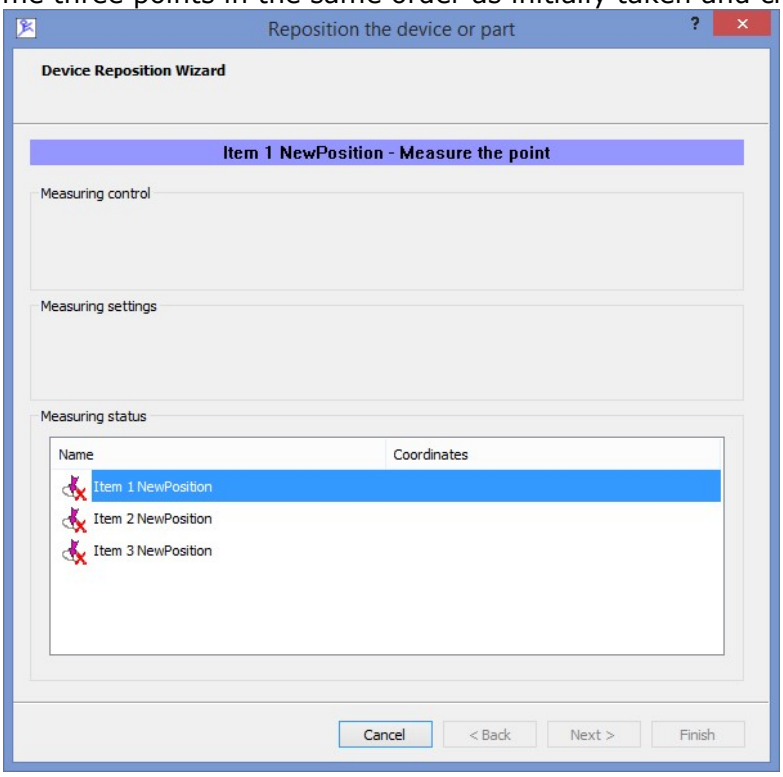
11 Click **Next.**
The wizard instructs you to move the inspection device to a new location.



12 Move the inspection device (or to simulate the same effect, move the component block to a new location).

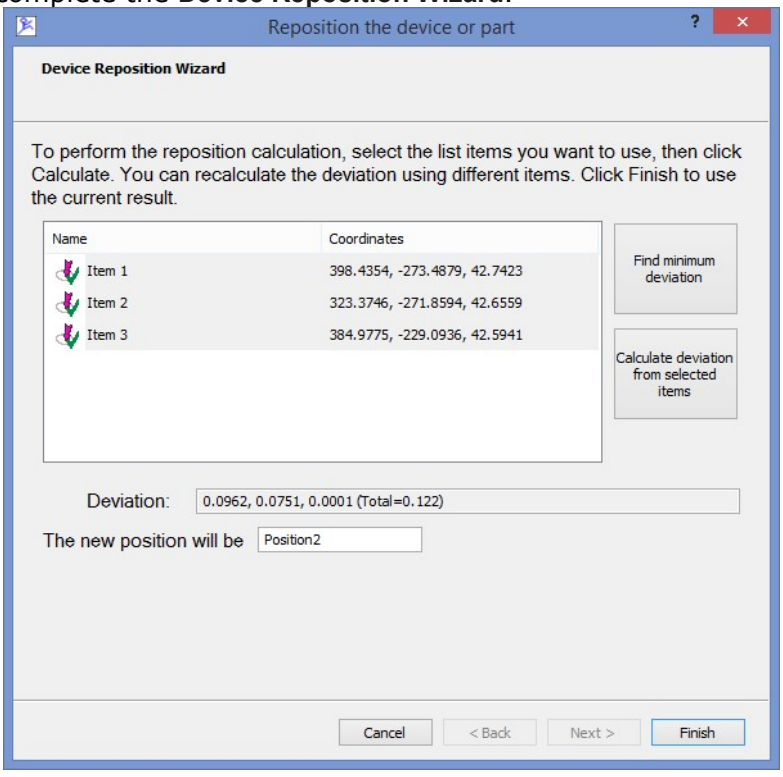
13 Click **Next.**

14 Measure the same three points in the same order as initially taken and click **Next.**

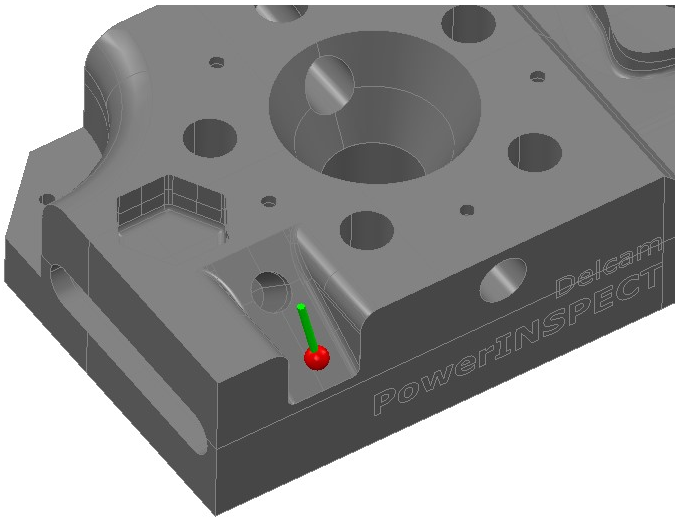


The deviation from the transformation is shown. **Calculate Deviation** can be used to calculate the deviation from selected items.

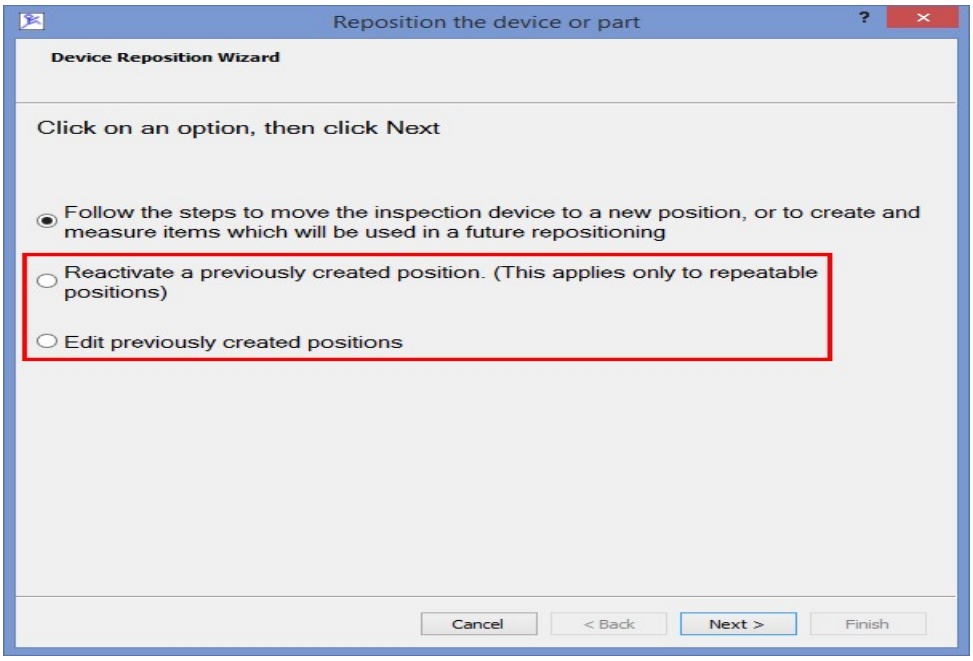
15 Click **Finish** to complete the **Device Reposition Wizard.**



The new position is saved. Name it **Position 2**.



16 Previously saved positions can be reactivated or edited from the opening page of the Wizard.

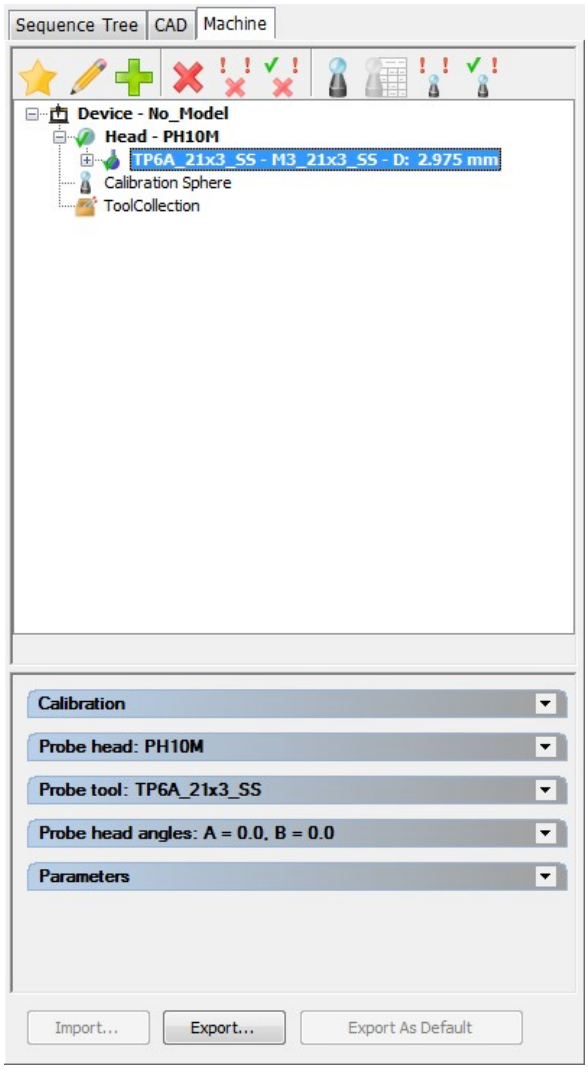


12. Machine Tab (Manual CMM)

The **Machine** tab is used to manage the measuring device when PowerInspect is connected to a **CMM**.

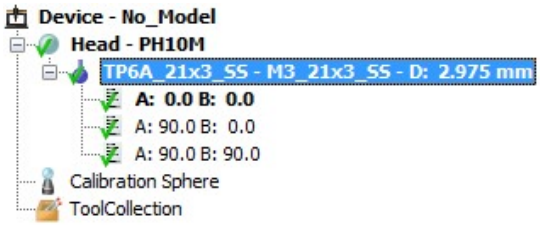
*No information is displayed in the **Machine** tab if PowerInspect is connected to an arm.*

The **Machine** tab is available from the document window to enable you to manage the probe heads, probes and calibrations in one place.



12.1 The Probe view

The probe view lists the details of the measuring device and probing tools for the document.



The active probe head, probe, and probe position are shown in bold. The view also indicates the calibration status by displaying a calibrated tick or uncalibrated cross for each entry in the list. The following options are available:

 Change active probe or probe position.

 Edit the selected probe tool assembly.

 Define a new probe position.

 Delete the selected item in the tree.

 Delete all uncalibrated probe positions in the tree.

 Delete all probe positions in the tree.

 Calibrate the selected item.

 Calibrate all uncalibrated probe positions in the document.

 Calibrate all probe positions in the document.

The following exercises demonstrate the creation of a new probe assembly and calibration.

The exact procedure will vary from CMM to CMM. You should follow the prompts given on your machine. Ask your tutor if in doubt.

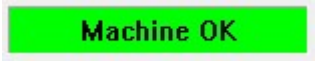
12.2 Creating a new probe assembly

For the purposes of this exercise, no CAD data will be used.

1 Click **New Document**  . A new empty session is loaded.

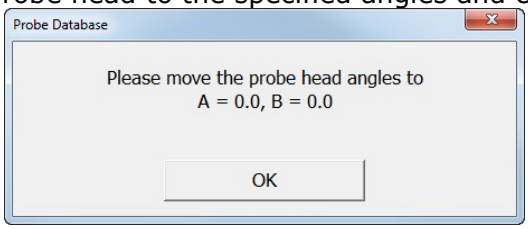
2 Click **Toggle Machine toolbar**  on the **Main** toolbar.

3 Click the **Connection**  button to connect PowerInspect to the CMM.

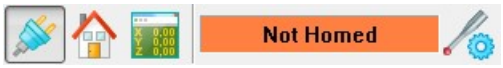
If successful, 

is displayed in the **Machine** toolbar.

4 If prompted, move the probe head to the specified angles and click **OK**.



The **Machine** toolbar displays the **Not Homed** message.



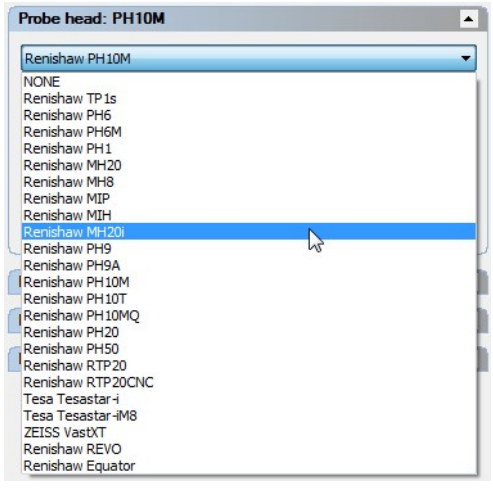
5 Click **Home**  .

6 Move the CMM to the positive end stops (machines vary) and then trigger the probe as requested.

The following assembly will be created as an example. Where applicable, substitute the components to match your own hardware.

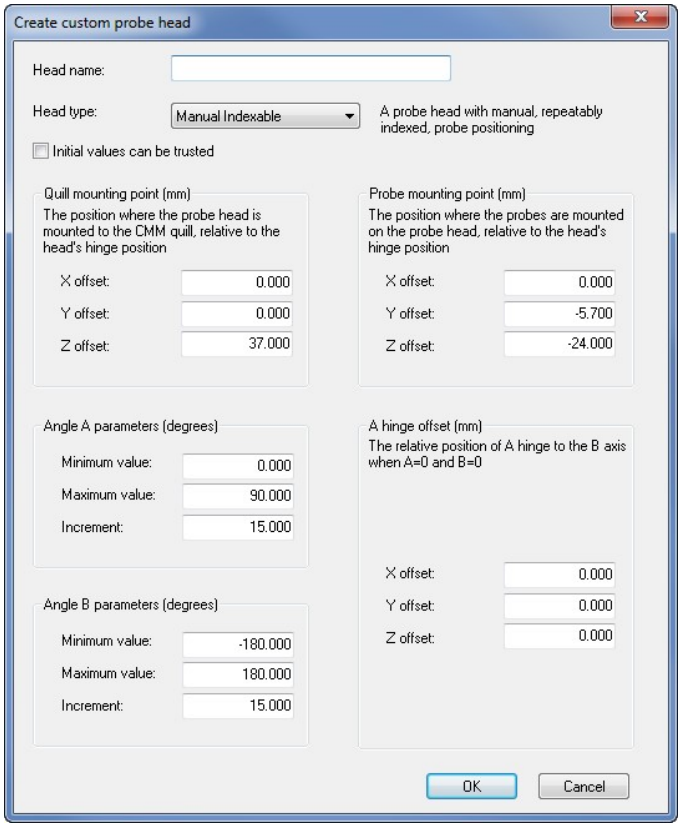
1 Expand the **Probe Head** tab by selecting the down arrow  ..

2 From the drop-down list, select the head **MH20i**.



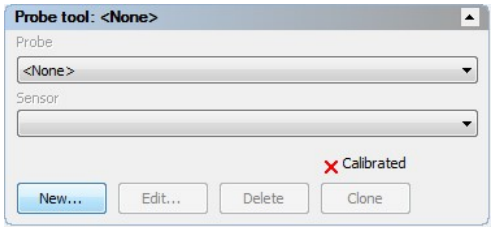
 *You can also create a new, custom probe head from this tab.*

*Select **Create** to display the **Create custom probe head** dialog and enter your custom values.*



The head type can be changed here (for example, Fixed, Manual Indexable).

3 Expand the **Probe Tool** tab.

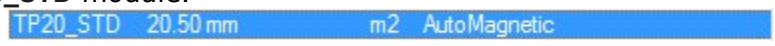


*The **Probe tool** tab provides the ability to create, edit and delete probes. **Clone** allows you to quickly copy an existing probe assembly and change/modify the required details to create a new one*

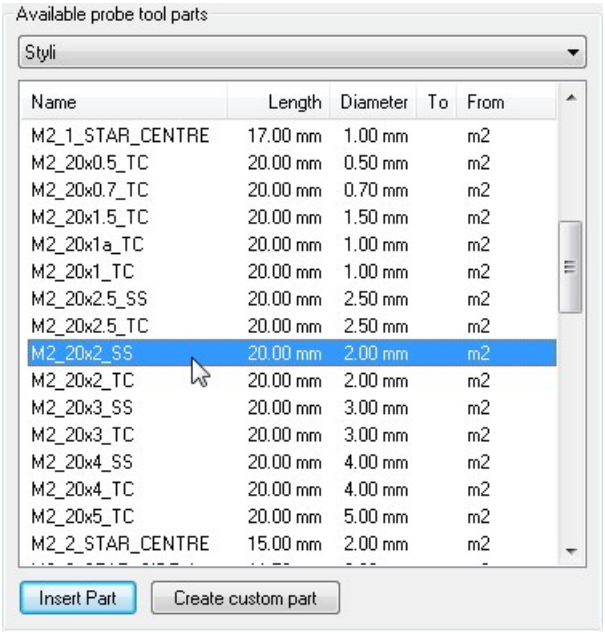
4 Click **New**.

5 At the top of the tab, enter a name for this assembly (e.g. **New Assembly**).
At the bottom the tab, a list of modules is displayed.

6 Select the **TP20_STD** module.

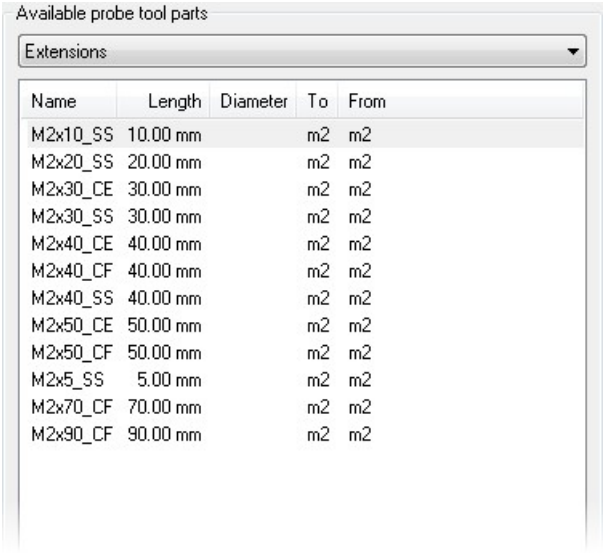


7 Click **Insert Part**.

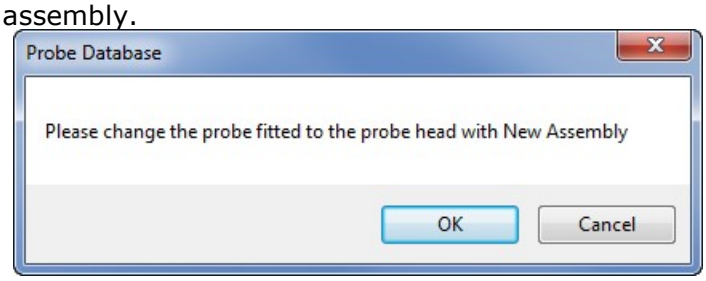


8 Select the **M2_20x2_SS** stylus.

9 Click **Insert Part**.



10 The final component to be selected is an extension. No extension is required for this example, so click **Save**. A message is displayed prompting you ensure the fitted probe matches the new assembly.



11 Ensure the correct assembly is fitted, and then click **OK**.

The defined probe and probe tool are kept in the .pwi file when saved, therefore this procedure does not need to be repeated for each new .pwi session.

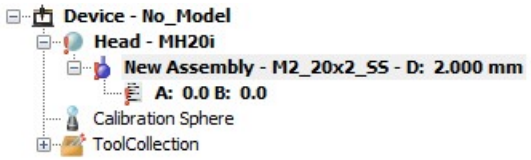
There are three options at the bottom of the **Machine** Tab.

These settings can be imported or exported (saved) to be used in other .pwi files. Therefore, this procedure does not need to be repeated for each new .pwi session.

You can also choose to export these settings to a definition .pdb file, which can then be imported into other .pwi files. In this way, you do not need to set up the same settings for each .pwi file.

To specify the default settings, click **Export As Default** on the **Machine** tab. This saves your settings to a default.pdb file, which is automatically loaded each time you create a new inspection session.

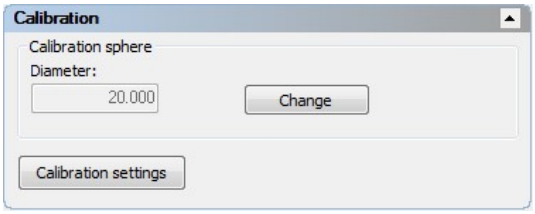
The new assembly is now displayed at the top of the **Machine** tab, summarising the tool setup.



12.3 Calibration

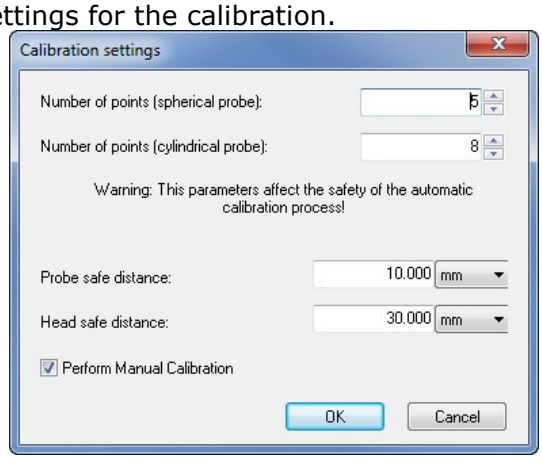
After you have created a new assembly, it will need to be calibrated.

1 Click ▼ to expand the **Calibration** tab.



The current diameter of the calibration sphere is displayed here. This can be changed after the head is calibrated.

2 Click **Calibration Settings** to display the **Calibration settings** dialog. This dialog can be used to change the overall settings for the calibration.



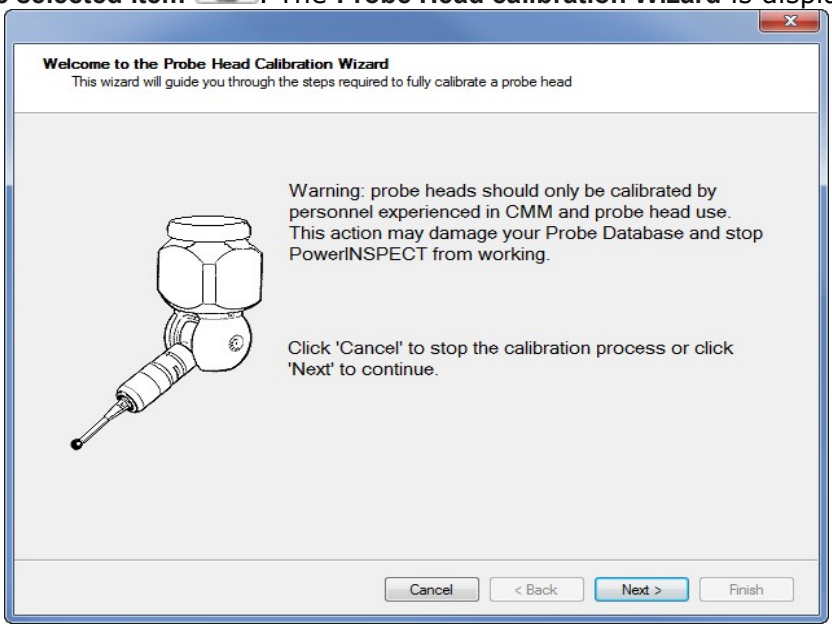
3 Leave the settings as default and click **OK**.

The Head must be calibrated first.

4 Click **Head-MH20i** in the assembly tree to highlight it 

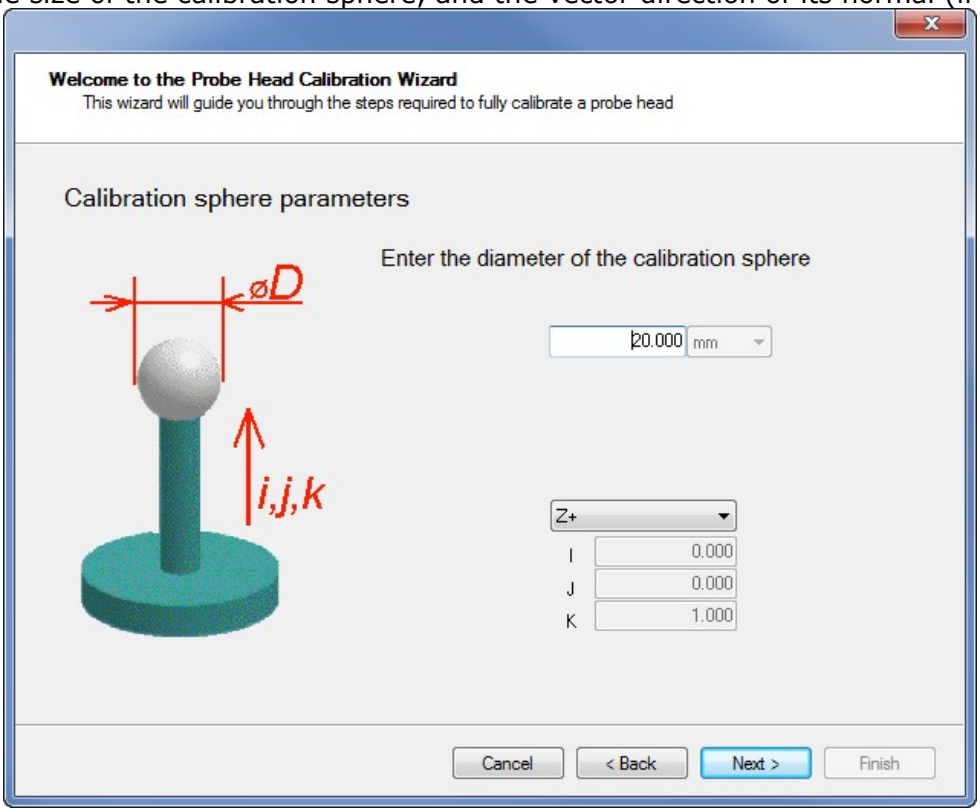
.

5 Click **Calibrate selected item**  . The **Probe Head calibration Wizard** is displayed.



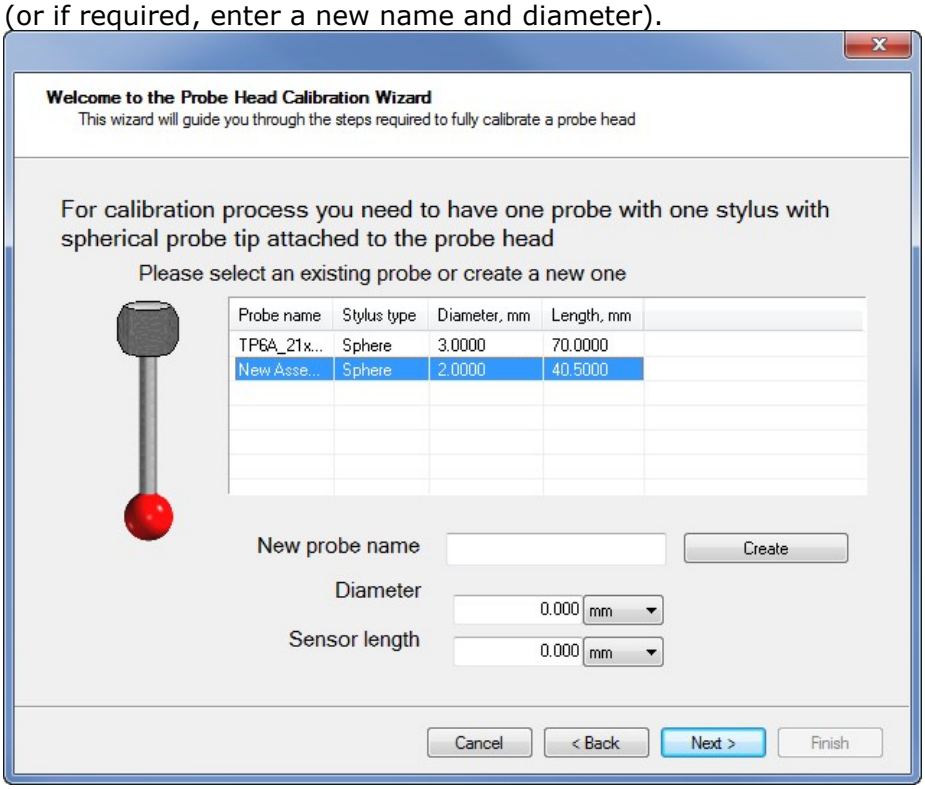
6 Click **Next**.

7 Enter the size of the calibration sphere, and the vector direction of its normal (if asked).



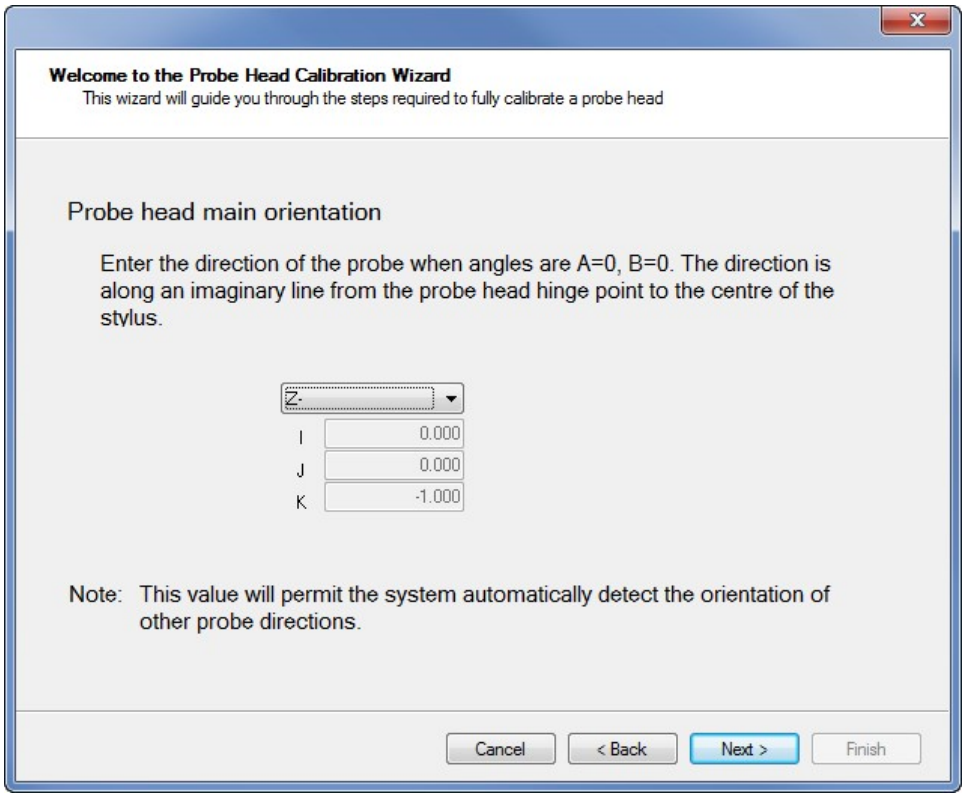
8 Click **Next**.

9 The next stage involves the creation of the Probe tool fitted to the head. Select the **New Assembly** (or if required, enter a new name and diameter).

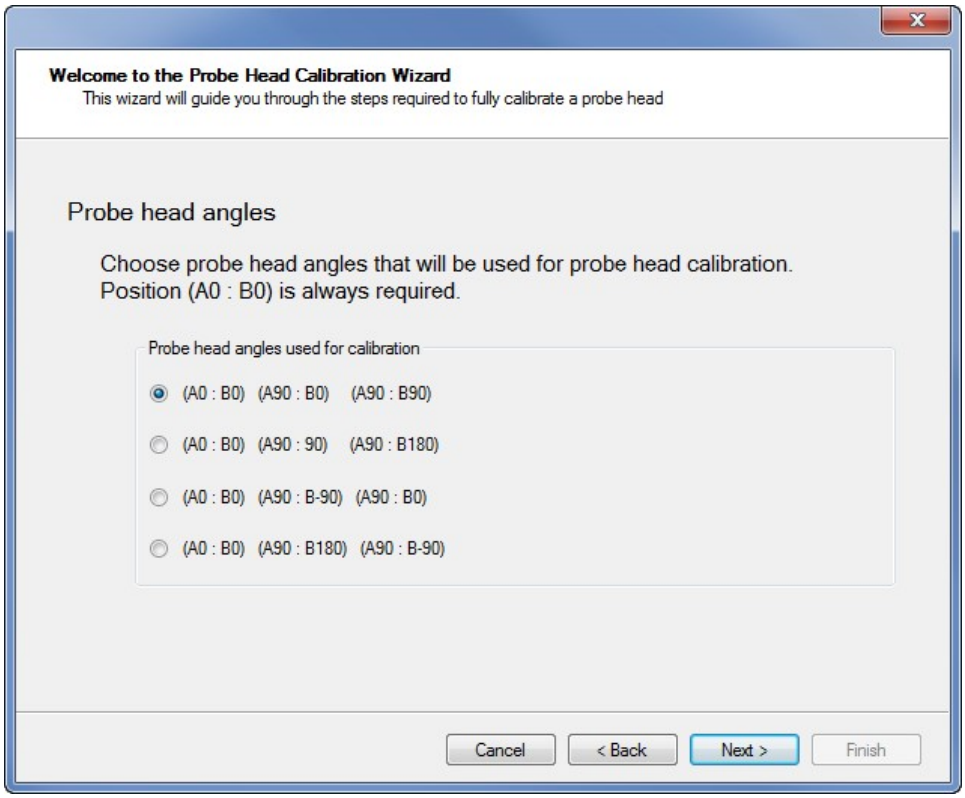


10 Click **Next**.

11 You are prompted to enter the direction of the probe when it is at the **A=0**, **B=0** position. In this case it is directly down Z.



12 Click **Next.**

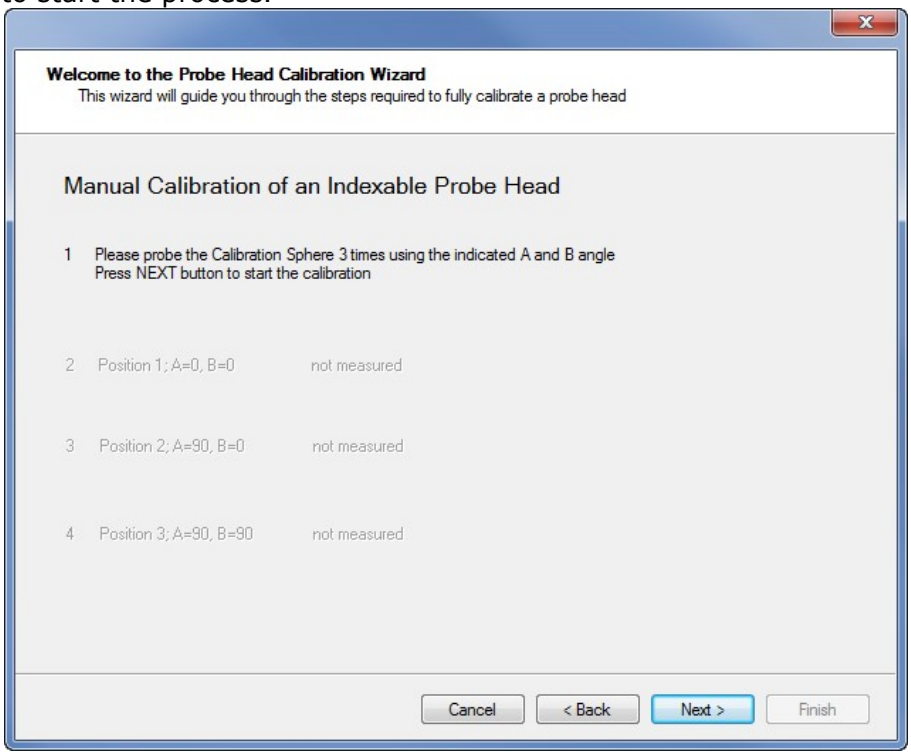


You have the option to select angles you wish to use in the calibration process. This is useful if the default (first option) angles are difficult to apply.

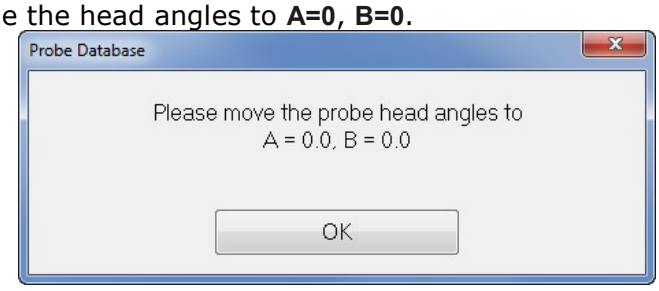
13 Leave the first default option selected.

14 Click **Next.**
The wizard prompts you to probe the calibration sphere.

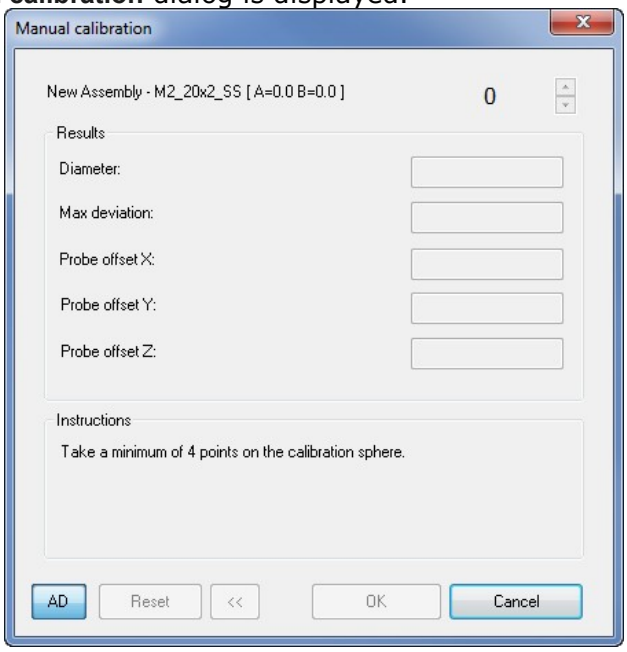
15 Click **Next** to start the process.



16 If prompted, change the head angles to **A=0**, **B=0**.



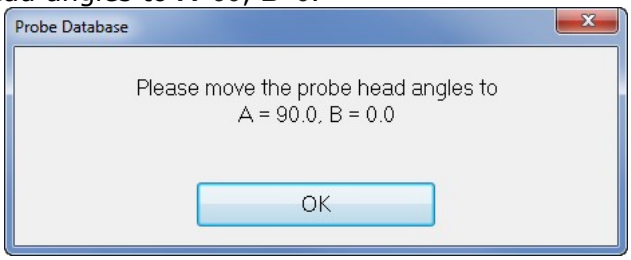
17 Click **OK**. The **Manual calibration** dialog is displayed.



18 Take at least 4 points around the sphere as shown.

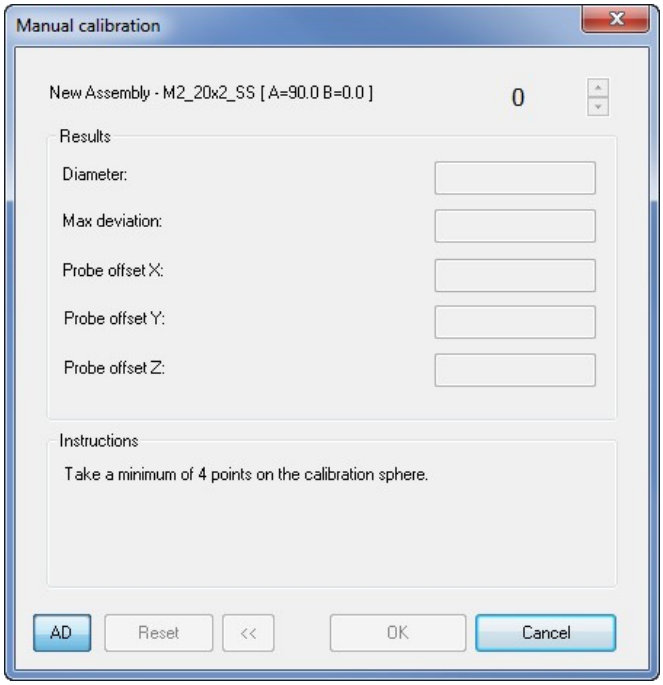
19 Click **OK**.

20 Change the probe head angles to **A=90, B=0**.



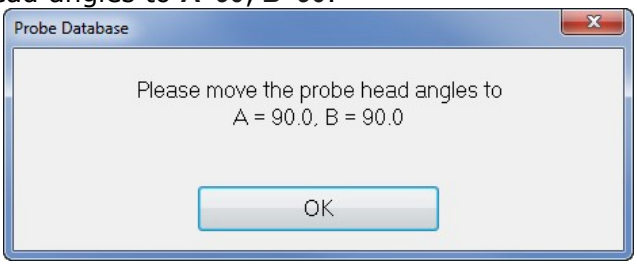
21 Click **OK**. The **Manual calibration** dialog is displayed.

22 Take at least 4 points.



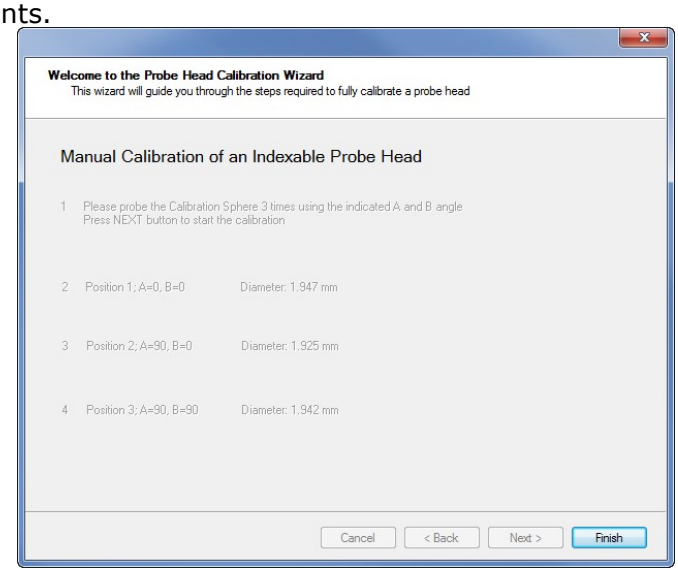
23 Click **OK**.

24 Change the probe head angles to **A=90, B=90**.



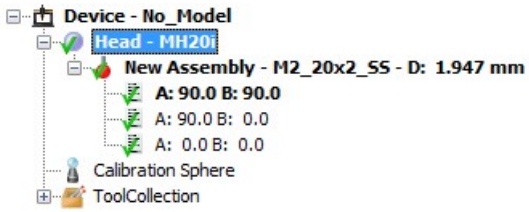
25 Click **OK**.

26 Take at least 4 points.

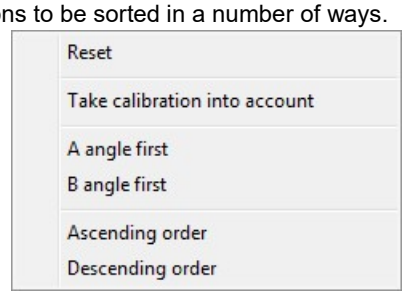


27 Click **Finish** to complete the calibration process.

The assembly tree is updated to reflect the calibration. The head and three positions used are calibrated and identified with a green tick.



The list is currently displayed in the order the probe positions were created. Right-click on **New Assembly** or a probe position allows the positions to be sorted in a number of ways.



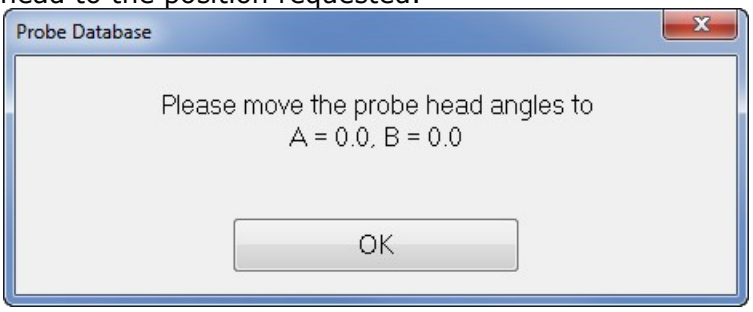
Currently, the position **A: 90 B: 90** is highlighted in bold. This identifies it as the active position.

If a different position is required, you must activate it first.

28 Click **A: 0.0 B: 0.0** to highlight it 
.

29 Click **Activate selected item**  .

30 Move the probe head to the position requested.



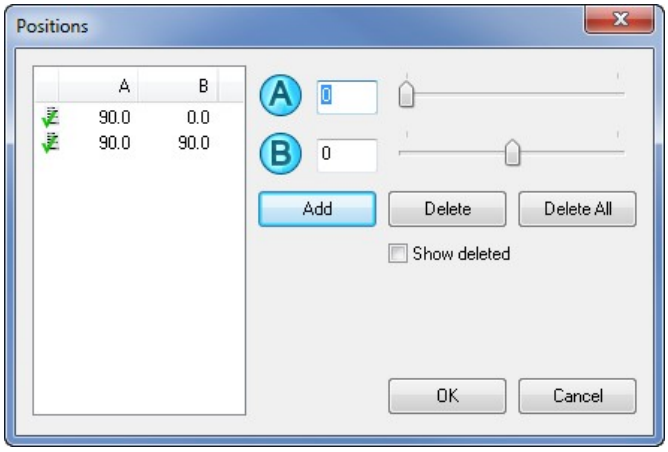
31 Click **OK**.
Position **A=0, B=0** is now ready to use.

12.4 Adding further probe positions

Further probe positions can be added using the **Positions** dialog.

1 Select the **New Assembly** in the tree.


2 Click **Add new position**  .



Use the slider bars or enter values directly in the boxes.

3 Enter **A=45, B=45.**

4 Click **Add** to add it to the current list of positions.

5 Select **OK** to close the dialog.
The new position is now listed in the assembly tree but it is marked by a red cross identifying it as uncalibrated.

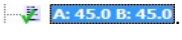
6 Highlight the position 

7 Click **Calibrate selected item**  .

8 When prompted, change the position of the probe to **A=45, B=45.**

9 Click **OK**. The **Manual calibration** dialog is displayed.

10 As with the previous calibration, take at least 4 points, and then click **OK.**

11 Click **Finish** to close the **Calibration Wizard.**
The new position is calibrated, active, and ready for use 
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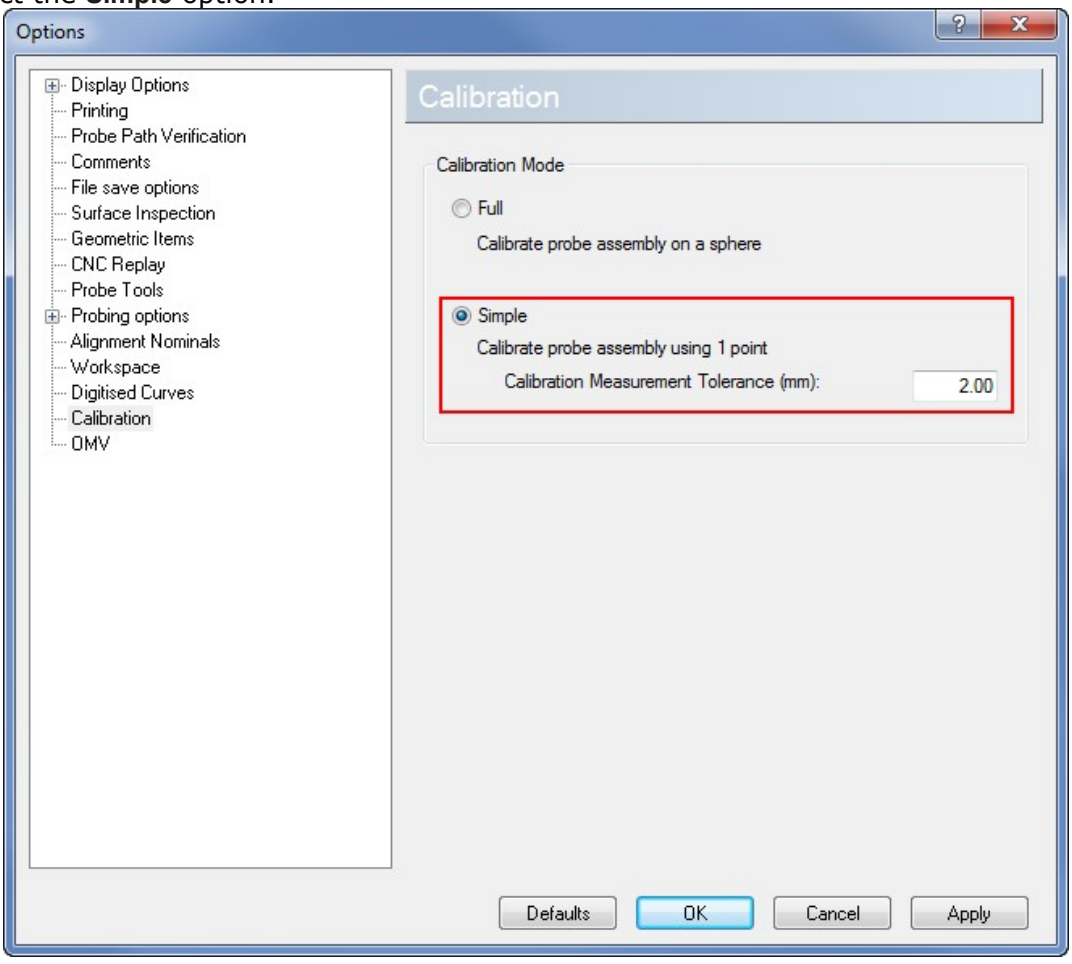
12.5 Single Point Calibration

Single point calibration is a method of determining the position of the probe tip. Instead of probing multiple positions on a calibration sphere, single point calibration uses the nominal offsets of the probe head and probe diameter, and one calibration point to calculate the offset of the probe tool. This enables you to quickly change the probe assembly and adjust the probe orientation without the need to go through a lengthy calibration procedure before running the inspection sequence.

Enabling single point calibration

To enable single point calibration:

1. Select the **Tools > Options** menu option. The **Options** dialog is displayed.
2. Select the **Calibration** page.
3. Select the **Simple** option.



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

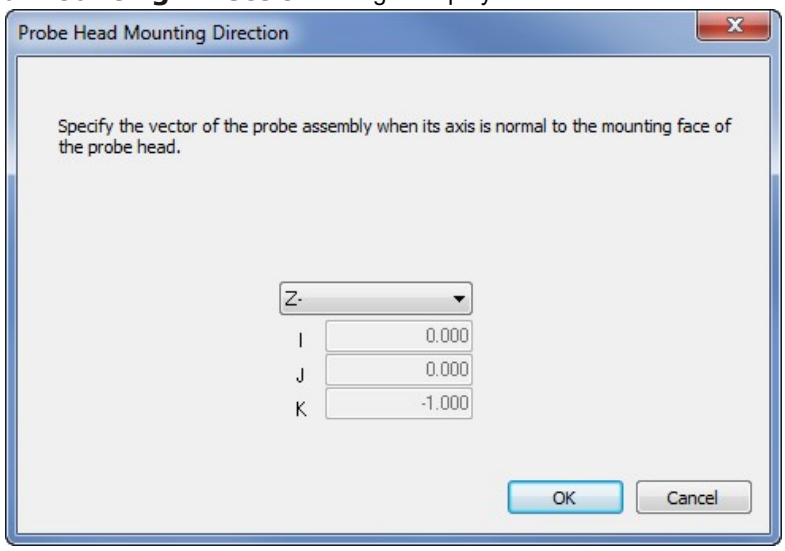
Calibrating the probe head

With the simple calibration method, PowerInspect uses the probe head nominal to calculate the position of the probe tip, so there is no need to perform any calibration measurements on it. However, before you can use the nominal to calibrate the probe head, you must specify the mounting direction of the probe head to specify the vector in which the offset is to be applied.

To calibrate the probe head:

1 Select the **Machine** tab.

2 In the Probe view, select the probe head and click  . Alternatively, in the **Probe head** area, select the probe head you want to use, and then click **Calibrate**.
The **Probe Head Mounting Direction** dialog is displayed.



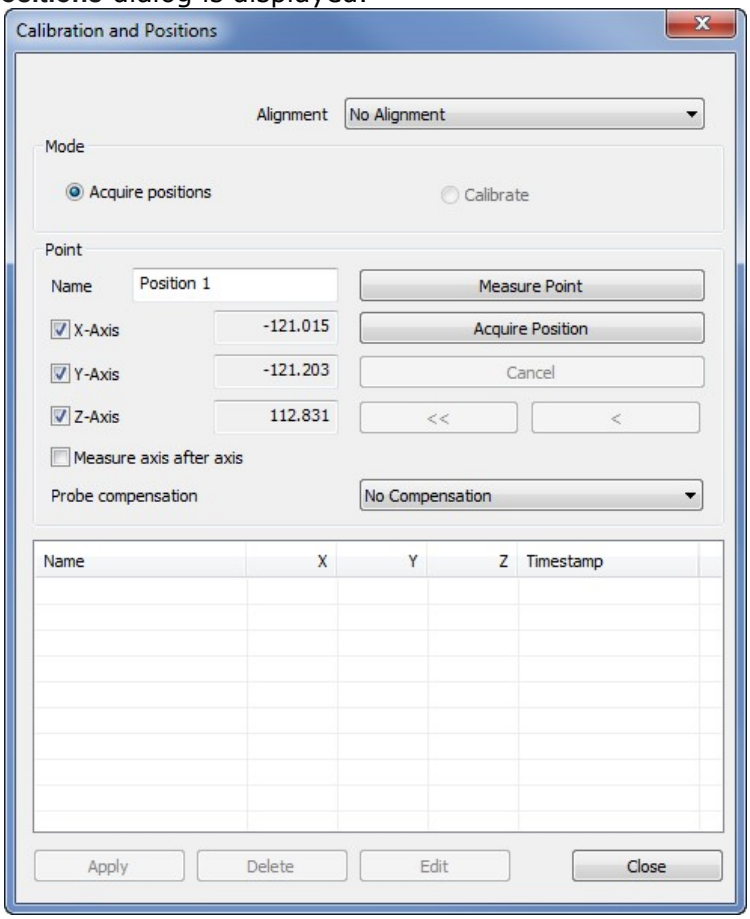
3 Select an entry in the drop-down list to specify which machine axis the head is parallel to. If you select **Custom**, enter the orientation vector for the probe head in the **I**, **J**, and **K** boxes.

4 In this case, leave the settings as default. Click **OK** to close the dialog and calibrate the probe head.

Calibrating the probe tool

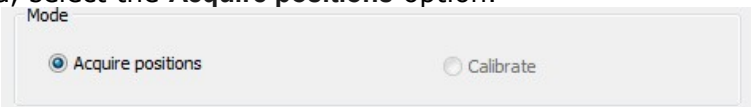
After calibrating the probe head, you can calibrate the probe tool. But first, you need to specify a calibration position.

1 Select the probe tool in the list of available tools and click the **Calibrate**  button. The **Calibration and Positions** dialog is displayed.



2 Use the **Alignment** list to select the alignment in which you want to report the calibration positions. In this example, there is no alignment available, so select **No Alignment** to display and specify positions in machine coordinates.

3 In the **Mode** area, select the **Acquire positions** option.



4 In the **Probe compensation** list, you can specify the direction in which you want to apply compensation for the diameter of the probe. Select:

- *an axis* to always apply compensation along the vector of that axis.
- **Automatic compensation** to apply compensation along the main axis nearest to the probing direction.
- **No compensation** to use the position of the probe centre without compensation.
- For this example, select **No Compensation**.

5 Enter a **Name** for the calibration position.

6 Specify the location of the calibration position:

- To create a position by probing a location in the workspace of the measuring device, click the **Measure point** button and probe the point.
- To create a position at a specified location, deselect the **X-Axis**, **Y-Axis**, and **Z-Axis** check boxes, enter the coordinates in the adjacent boxes, and click the **Acquire position** button.
- To create a position at the current location of the probe tip, click the **Acquire position** button.
- When you have created a position, its name and coordinates, and the time at which it was taken are displayed in the position list.

7 Repeat steps 4 to 6 to create more calibration positions.

*To change a point, select its entry in the list, click **Edit**, and enter new coordinates for the axes.*

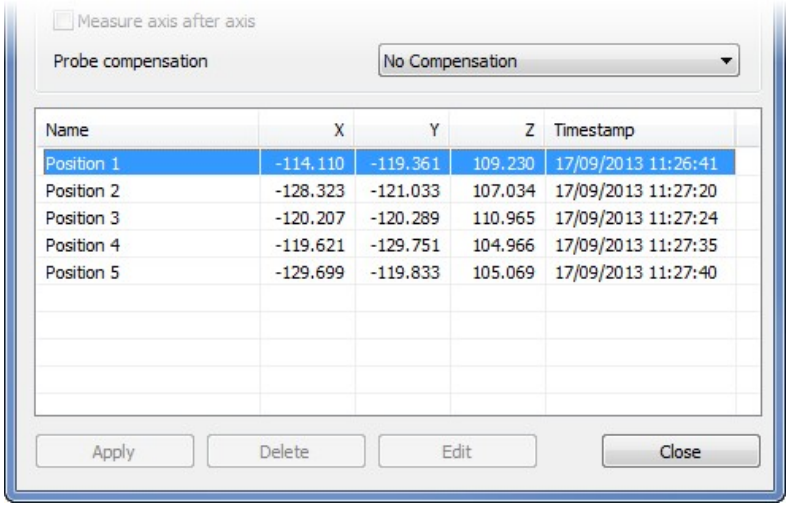
8 When you have created the positions you want to use, you can now calibrate the probe tool. Leave the **Calibration and Positions** dialog open.

Calibrating the probe tool

1 In the **Calibration and Positions** dialog, select the **Calibrate** option.

2 Orientate the probe.

3 In the position list, select the location with which you want to calibrate the probe.



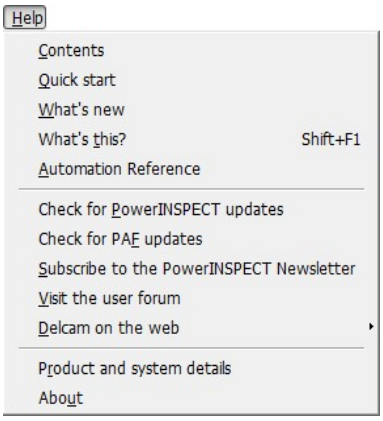
4 Move the probe tip to the selected position, and click the **Acquire position** button. Alternatively, click the **Measure point** button and probe the position.

5 Click **Close**.

13. Help & Shortcuts

This chapter contains information about the **Help** menu and useful keyboard shortcuts.

13.1 Help menu



Use the **Help** menu to access information about PowerInspect. Select:

- **Contents** to display the reference help file.
- **What's new** to display the What's New file. The What's New includes information on new and updated features for the current release of PowerInspect.
- **What's this?** to display help on a specific feature. The cursor changes to . Click on a button, toolbar or area to display the relevant page in the Reference Help.
- **Check for PowerInspect updates** to check if there are any updates available with your software licence.
- **Check for PAF updates** to check if there are any updates required for your PAF.
- **Subscribe to the PowerInspect Newsletter** to sign up and receive the latest information about PowerInspect by email.
- **Visit the user forum** to display the user forum page. You can use the forums to discuss any issues with other customers.
- **On the web** to visit the PowerInspect homepage.
- **Product and system details** to display a .txt file containing system information, details about your PowerInspect install and licensing information.
- **About** to display a dialog showing information about the application.

13.2 Keyboard shortcuts

There are keyboard shortcuts for many of the common operations within PowerInspect.

Opening, closing and printing files

File > New	Ctrl+n
File > Open	Ctrl+o
File > Save	Ctrl+s
File > Print	Ctrl+p

Editing files

Edit > Cut	Ctrl+x or Shift+Delete
Edit > Copy	Ctrl+c
Edit > Paste	Ctrl+v
Edit > Delete	Delete

Manipulating the view

View > Resize to Fit	Ctrl+a
View > Show Grid	Ctrl+g
View > From > Top (Z)	Ctrl+5
View > From > Front (-Y)	Ctrl+2
View > From > Right (X)	Ctrl+6
View > From > Bottom (-Z)	Ctrl+0
View > From > Back (Y)	Ctrl+8
View > From > Left (-X)	Ctrl+4
View > ISO > Isometric 1	Ctrl+1
View > ISO > Isometric 2	Ctrl+3
View > ISO > Isometric 3	Ctrl+9
View > ISO > Isometric 4	Ctrl+7
Toggle shading	s
Toggle hidden lines	w
Wireframe view	F2
Shaded view	F3
Wireframe with hidden lines view	F4
Zoom in	d
Zoom out	h

Working with surface inspection groups

Sort points by name	Shift+Ctrl+Alt+1
Sort points by deviation	Shift+Ctrl+Alt+2

Working with dynamic points

Reverse the direction of the normal projecting from the dynamic point	Ctrl+m
Convert a surface point to an edge point or convert an edge point to a surface point	Ctrl+b

Working with point clouds

Project points in the Point Cloud tab onto the CAD surface	Ctrl+Alt+p
Load point clouds from a file to evaluate a selected laser-scanned item	Ctrl+Alt+f

Working in full-screen measurement mode

Accept the points taken and close the current Feature Probing dialog	Enter
Cancel all the points taken and close the current Feature Probing dialog	Esc
Delete the last point taken	Backspace
Delete all points taken	Ctrl+Backspace