Delcam

PowerINSPECT

Training Course

Delcam

Talbot Way, Small Heath Business Park, Birmingham, B10 0HJ.
www.delcam.com

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Training Centre
Tel: 0121 683 1050
Fax 0121 7665511

Customer Support
Tel: 0121 683 1010
Fax: 0121 7665542

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1. Introduction

PowerINSPECT 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**)
With a standard Windows application layout, **PowerINSPECT** allows the user 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.
The software gives the user instantaneous results during the inspection and the onscreen reaction and detailed graphical displays give immediate feedback for each measured point.

PowerINSPECT supports models created by other Delcam products or from neutral formats such as IGES. If the relevant **PS-Exchange** translators are purchased **PowerINSPECT** will directly import data created by the majority of non-Delcam packages, such as CATIA, Unigraphics, SolidWorks, IDEAS etc.

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

It must be noted that the examples given in these training notes are specific to the models used in the training course. Therefore, it is the techniques that should be learnt, and adapted to the users own requirements.

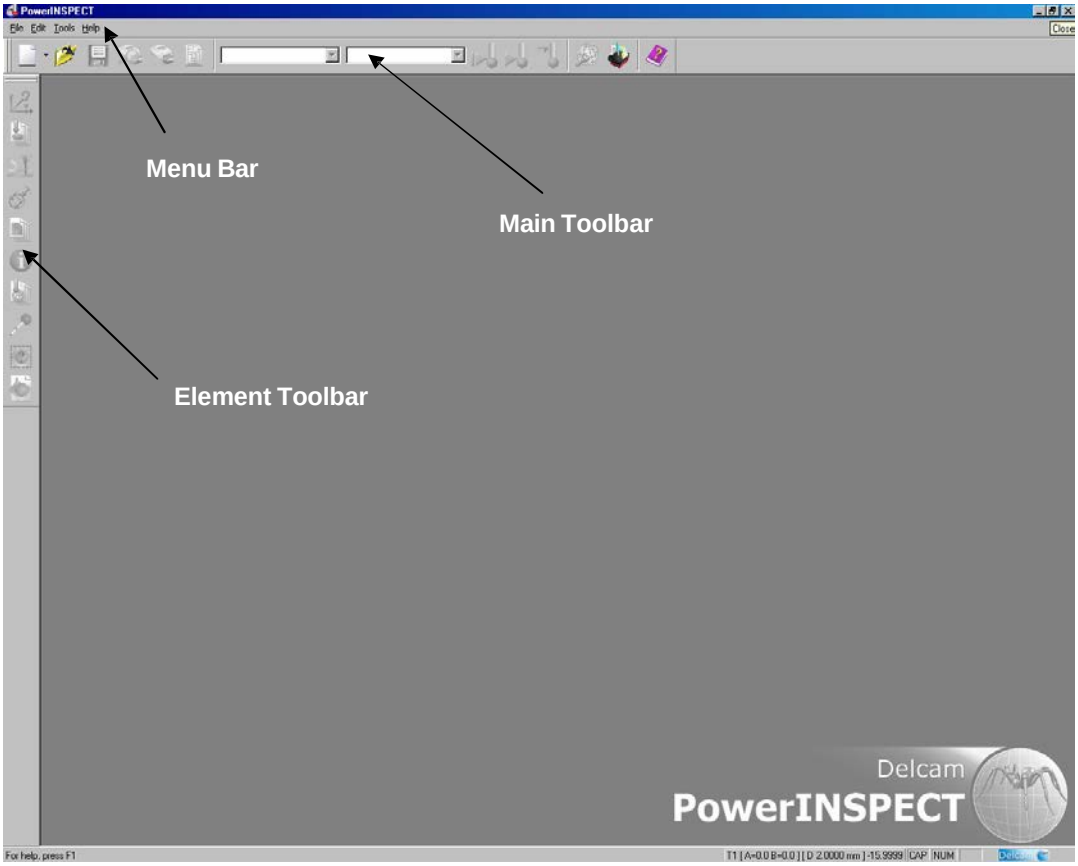
PowerINSPECT gives the User:

- Comparison against all mainstream CAD formats.
- Rapid alignment even for complex freeform shapes.
- Inspection of 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.
- A step-by-step Geometric Dimensioning & Tolerancing (GD&T) Wizard.
- Point, wireframe and surface export for measured entities.
- CAD manipulation - including surface offsetting (e.g. reverse side for sheet metal, spark gap for electrodes).

Starting PowerINSPECT

- To load **PowerINSPECT**, double-click on the **Desktop icon**.

Once loaded the screen should look as follows:



When **PowerINSPECT** loads up, the **Work Environment** remains empty, until the user starts a **new** session. When a **new session** begins more options become available in the **Menu Bar**, and a **Graphics Window**, **Sequence Tree** and **Context-Sensitive Toolbar** are automatically generated. These are summarized below.

Menu Bar

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

Clicking on a menu item opens a pull-down menu that contains submenus and commands. If a menu item is dimmed then it does not apply to the currently active document. Some sub- menus contain additional sub-menus, indicated by a small arrow to the right of the text.
Clicking on an arrow generates a further list of command options.
Clicking outside a menu box will cancel any command

Main Toolbar

The **Main Toolbar** is displayed at the top of the project window under the menu bar. The tools allow the user easy access to the most frequently used **PowerINSPECT** menu items.

Each **Icon/Button** corresponds to different functions, the definitions of which are given in the
Glossary (Chapter 14), and by clicking on them their assigned task will be performed.

Additional functions can be added to the **Main Toolbar** by either right clicking in the **Main Toolbar** area and choosing other menu items, or by selecting other functions (e.g. the CAD File Manager – see Chapter 3).

Element Toolbar

The **Element Toolbar** (left hand side toolbar) is used for the creatio
Alignments and **Geometric/Inspection Groups**.

This toolbar can vary according to the function that is chosen (this will be expanded upon in later chapters).

The **Comment Icon**  can be used, for example, to provide another user with detailed instructions on how to perform an inspection, or how an inspection was performed.

For instance, combining images and videos inside a comment, can guide other users to specific areas for inspection.

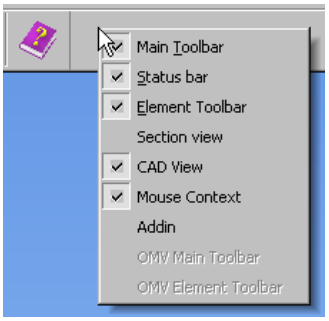
Please note that the Toolbars are *Dockable*, as they can be dragged a new screen position if the user requires.

CAD Toolbar

The CAD Toolbar (right hand side) contains the options to orientate and shade the CAD model.

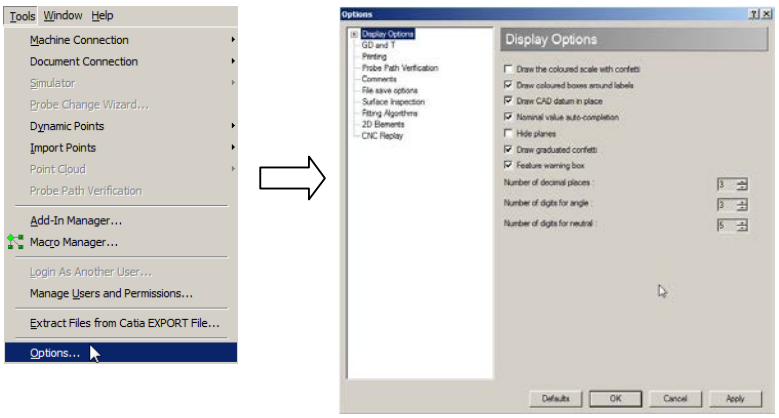
Labels displaying measured/selected data can be controlled from thi olbar.

Right-mouse button selecting in the Grey menu bar areas can toggle the visibility of all Toolbars.



Small black arrows under icons can be selected to display further options.

Selecting **Options**... from the **Tools** menu allows the user to specify settings for various PowerINSPECT features.



Creating a New Session

A new session can be started either from the **File menu** or directly accessed from the **Main Toolbar**. There are two options available.

 Creates a **new** inspection document **without** loading a CAD model.

 The **New Document Wizard** allows users to choose whether they want to open no CAD models, one model or multiple models.

If a **PowerINSPECT** session **(*.pwi)** *already exists*, then the **Open** option can be used in the file menu, or the **Open Button**  on the **Main Toolbar** can be pressed.

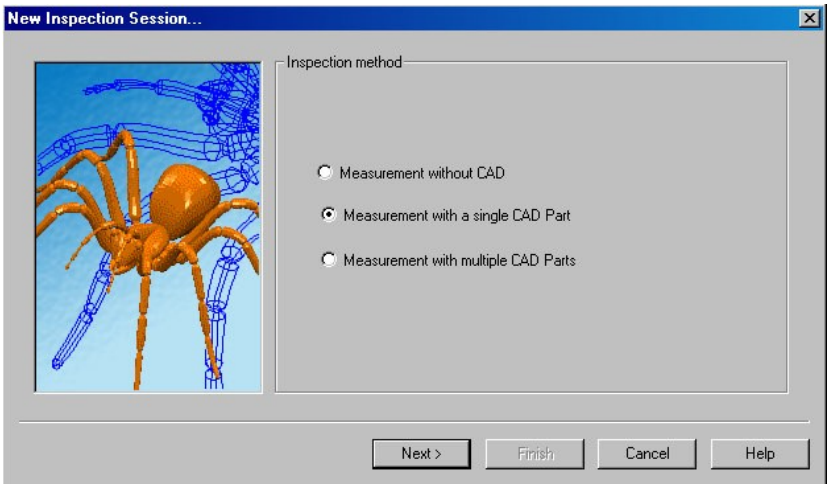
Note that this icon corresponds to that in the file menu.

When a new session begins using the **New Document Wizard**, the user is guided through a step-by-step instruction.

New Document Wizard 

- Select **New Document Wizard.** 

The first section opens the New Inspection Session window



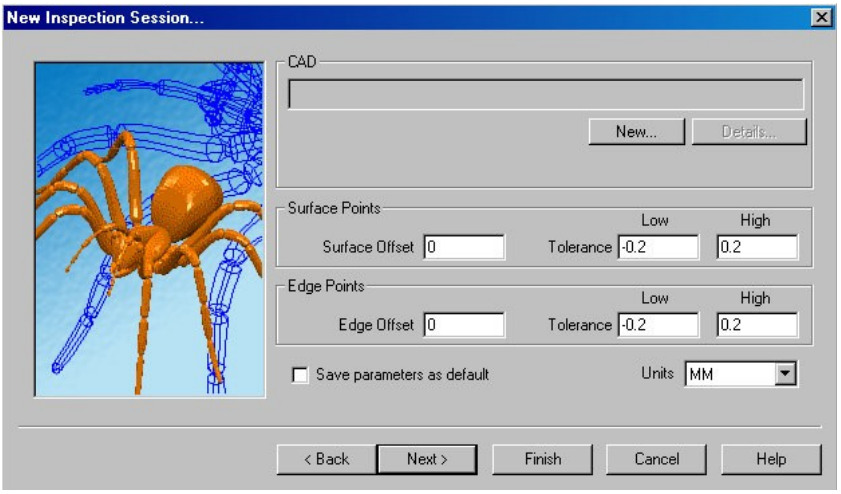
The user can select from the options displayed.

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

(This is the same as selecting new session  *from the Main Toolbar.)*

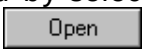
Measurement with a single CAD part creates an inspection session using one CAD model.
Measurement with multiple CAD parts creates an inspection session using several CAD models.

- Ensure the option **Measurement with a single CAD part** is selected, then select **Next** to display the second page of the wizard.



Within this window the user can specify the **units** that they wish to work in, **applying offsets** and **specifying** the tolerance for surface and edge points. Primarily, the **CAD** file/model location is loaded here.

The **CAD File** is located by selecting 

, browsing for the file, and then selecting 

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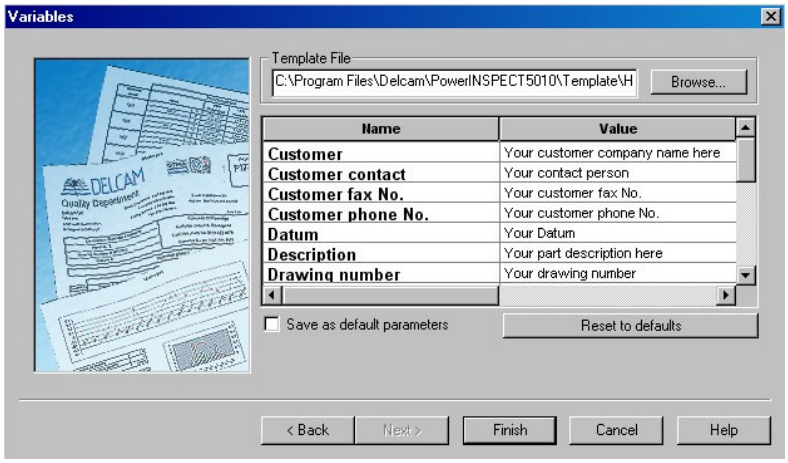
Once the CAD File has been selected, it can be **transformed** (see Chapter 3), or further CAD files can be added (i.e. for an assembly). Further models can be selected from within a session using the **CAD** manager tab, or from the first page of the wizard.

The user can then move on to the next section of the Wizard using 

Button.

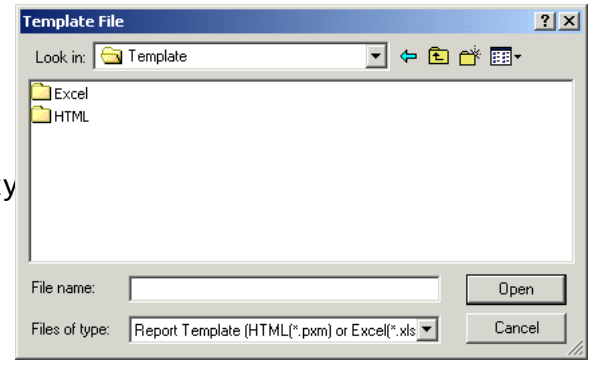
Variables Window and the Report Template

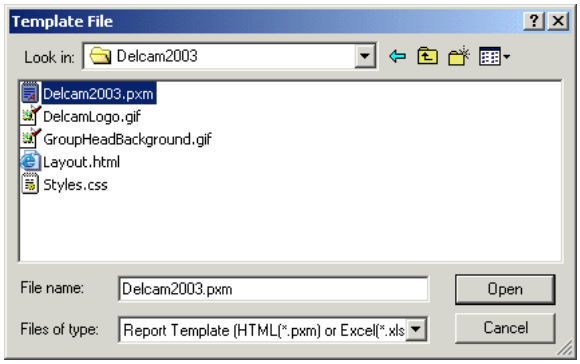
The **Variables Window** allows the user to select the **Report Template** they wish to output to. As with the CAD File selection the user chooses the **Report Template File** via the **Browse Button**.



When browsing, the user has 2 options.

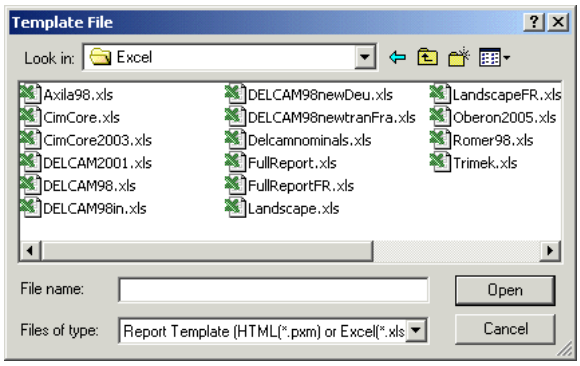
PowerINSPECT 2010 has the ability **Reports** in 2 formats.

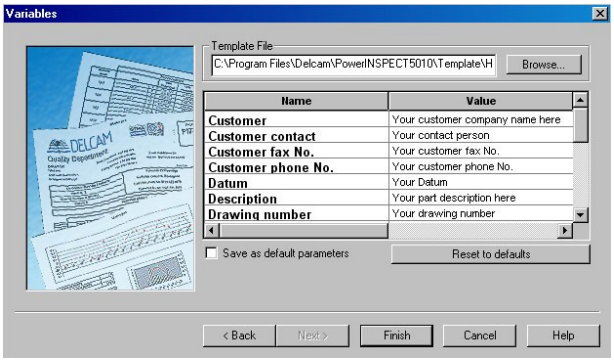




The first allows the user to create a report using an HTML format. These reports are created directly inside the **PowerINSPECT *.pwi** file, and are accessible through the **Report** tab in the **Area Selection Tabs** area of the **Graphics Window**.

The second allows the user to create a report using **Microsoft Excel**, thus allowing backwards compatibility for older versions.





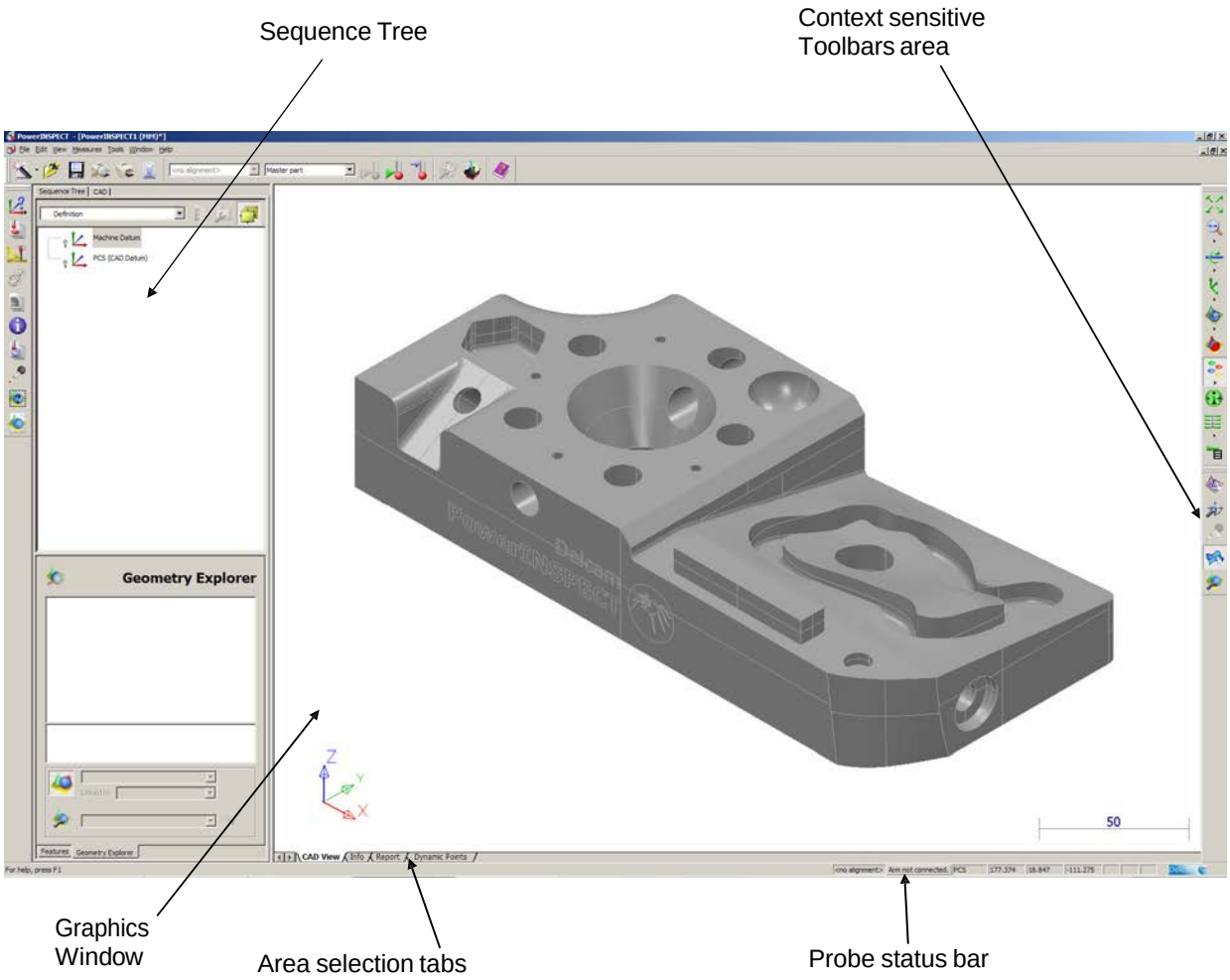
The entries can be directly edited by simply clicking on the required field, and editing the contents.

To finish the process and begin the session, the 

Button is selected.

Graphics Window

With a new session now open, some changes can be seen in the **PowerINSPECT Work Environment**. These include several new menus, a **Graphics Window** (seen here as the default **CAD View**, with a series of area selection tabs) and a **Sequence Tree** (pictured here with the CMM and PCS Datum Icons). To the right of the **Graphics Window** is the **Context-Sensitive Toolbars** area (pictured here with the CAD view toolbar) whose toolbars change according to the selected area tab the user is operating in. At the base of the screen is the Probe Status Bar (see Chapter 2). *Note: Screen layout is for manual licence.*



The **Graphics window** displays a number of tabs, including the **CAD View** tab where the user can view the CAD model and visualise the inspection they are carrying out. There are five display tabs available:

CAD View	Where the CAD data, against which the user is inspecting, is displayed.
Info	Allows the user to view information about a selected **Sequence Tree** element.
Dynamic Points	**PowerINSPECT** displays this tab when the user uses dynamic points to create a **Free Form Fit Alignment**.
Report	Allows the user to view the **Report** in **HTML** format.
Section	**PowerINSPECT** displays a Section tab each time the user creates a **Section Inspection Group**.

Views and View Manipulation

Now that the new session has been created the user may 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.

Firstly though, the use of the mouse functions must be established.

Mouse buttons

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

Mouse button 1: Picking and selecting

This button is used for selecting items from the pull down menus, options within forms, and entities in the graphics area.

It is also responsible for view manipulation depending on the setting used in the **Set Mouse Button 1 View Mode**  .

Mouse button 2: Dynamics

Zooming in and out: - Hold down the **Ctrl** key and mouse button 2. Move the mouse up and down to zoom in and out. Or alternatively, rotate the **Scroll Wheel** (if available).
Pan around the model: - Hold down the **Shift** key with mouse button 2. Move the mouse in the required direction.
Zoom Box: - Hold down the **Ctrl** and **Shift** key, drag a box around the area to zoom into using the middle mouse button.
Rotate mode: - Hold down mouse button 2 and move the mouse, the view now rotates.

Mouse button 3: Dynamics, Special Menus & PowerINSPECT Sequence Tree Options

Zooming in and out: - Hold down the **Ctrl** key and mouse button 3. Move the mouse up and down to zoom in and out.
Pan around the model: - Hold down the **Shift** key with mouse button 3. Move the mouse in the required direction.
Zoom Box: - Hold down the **Alt** key and mouse button 3, drag a box around the area to zoom into.
Rotate mode: - Hold down the **Ctrl** and **Shift** keys, and mouse button 3, to rotate the view.

When this button is pressed on its own it brings up a context sensitive menu based on whatever the mouse is over, such as the Sequence Tree, or the toolbar menus.

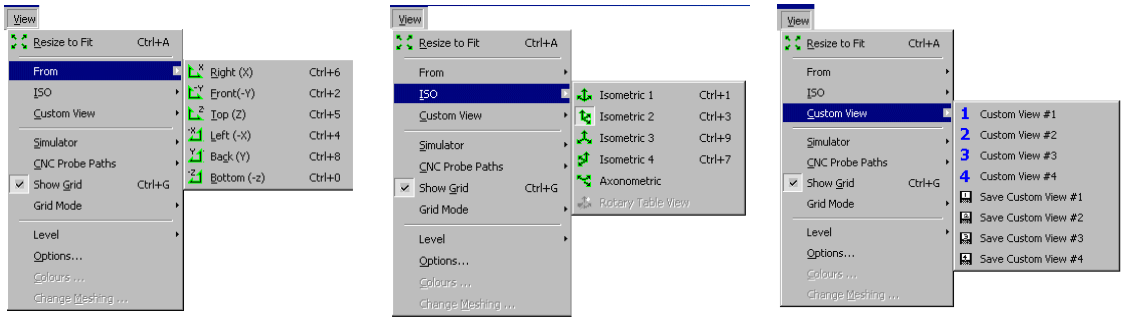
Choosing Views

View choice can be made in several ways using the **View Menu**, **View Palette** and in some cases using the **Context-Sensitive Toolbar**. In all cases the icons/buttons are the same and definitions of these can be located in the **Glossary** (Chapter 10, Page 74)

The **View Menu** is accessed from the top menu bar and contains the **Resize to Fit** option, and a number of **View Sub-Menus**. Each of these sub menus corresponds to a different group of views.

The **From** views look at the model using particular axes, the **ISO** views are isometric and the **Custom Views** are user defined (which can be saved in this sub-menu).

The **Simulator** and **Probe Path** menu options are for the **CNC** version of the software, with the other options relating to the **CAD View Grid**, and various **Colour**, **CAD Mesh** and **Level** changing options.

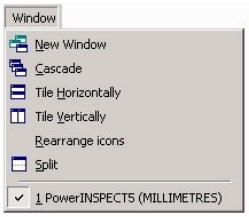


The **View Palette** shares these view-changing functions.

This **Palette** 'pops-up' by clicking the arrow attached to the **Select View Button**  in the **CAD View Toolbar** and contains a series of buttons, which 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** pop-up toolbar  . The rotation axis is selected from the toolbar and the view then rotated incrementally by clicking on the icon

Finally, if more than one session has been opened, the user can view all sessions in a single screen, using the **Window** menu. The sessions can then be arranged according to the users preference.

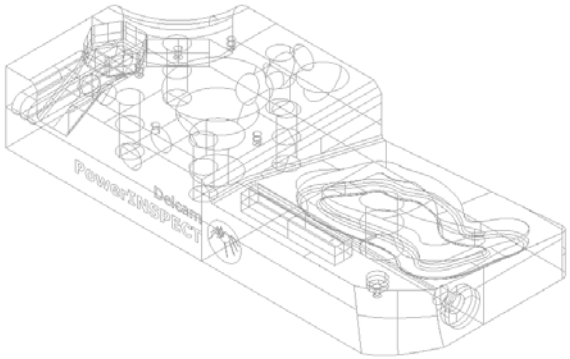


The CAD View

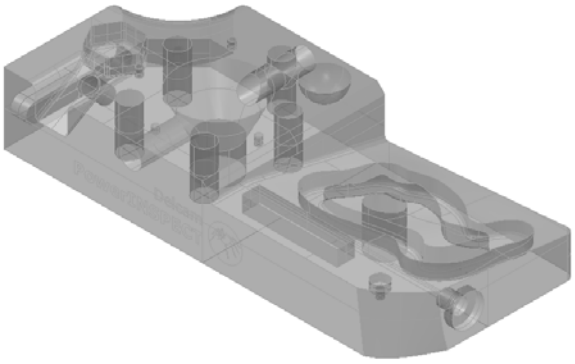
The **CAD View** within **PowerINSPECT** is where all the **Inspection** and **Geometric** information is viewed. **CAD** inside this view can be seen in a variety of different guises.

Models can be seen either with **No Shading**, **Transparent Shading**, or **Solid Shading**, combined with or without **Wireframe**.

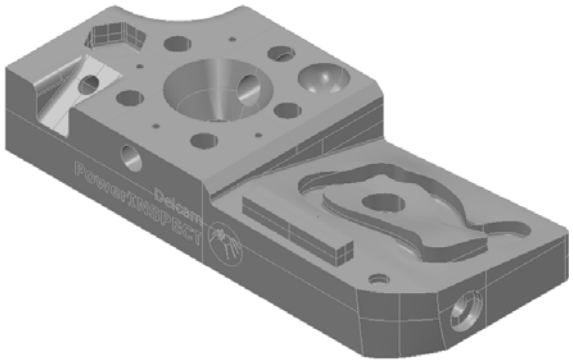
Shading changes are made through either the **Shading Mode For Model** pop-up toolbar button  , which opens this toolbar  , or by using the shortcut keys (**W=Wireframe on/off**, **S=Shading on/off**). Some examples of these modes can be seen to the right and below.



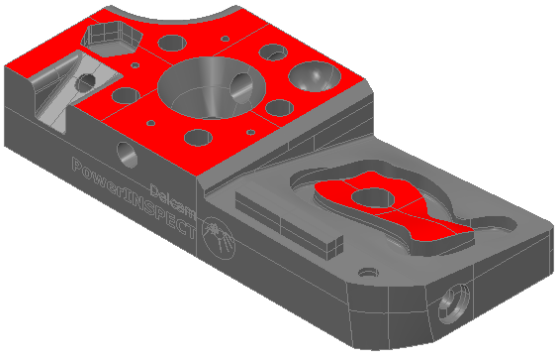
Wireframe Only



Transparent Shading

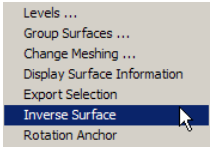


Solid Shading



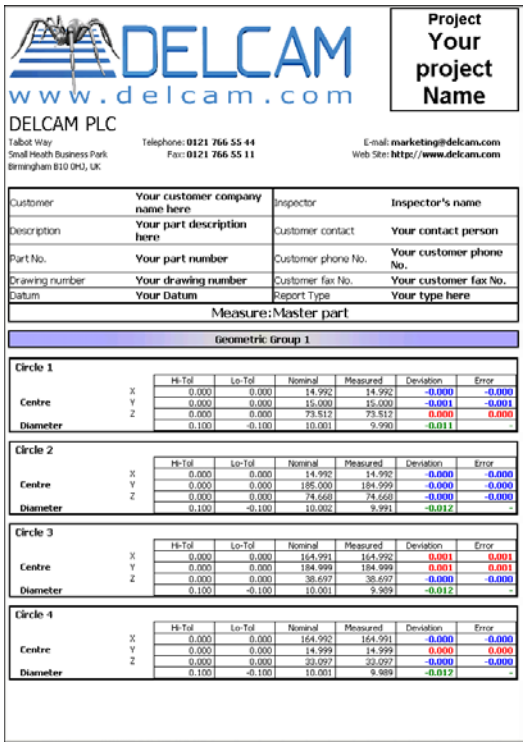
PowerINSPECT also has a function which allows the user determine the surface normal direction.
Using the **Highlight Back-facing Surfaces** button  , in combination with the shading, will show all reversed surfaces as bright red. This is useful for determining the direction needed for Surface Off-sets.

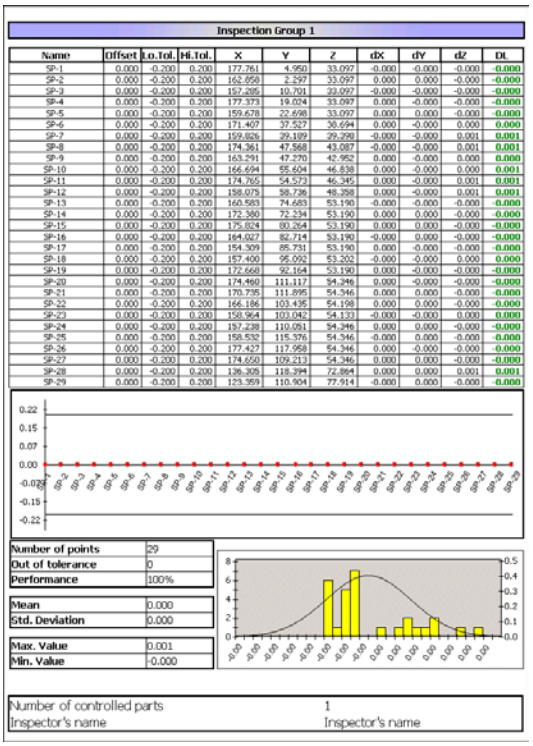
The Surface can be **reversed** by using **Surface selector**  to select it first and then selecting **Inverse Surface.** (Right mouse click)



The Report Template

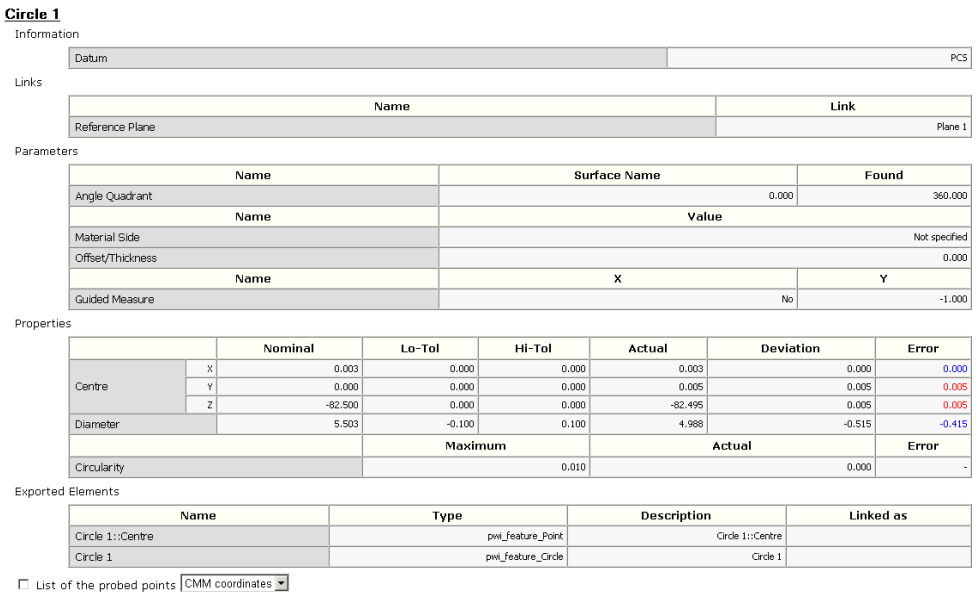
As highlighted in the variables section, **PowerINSPECT** allows the user to select a **Report Template** they wish to output their data to. If using the **HTML** format, the report is integrated into the **PowerINSPECT** work environment, and can be accessed at any time using the Report **Tab**. These reports can be **customised** to suit the customer's needs.





The Information Tab

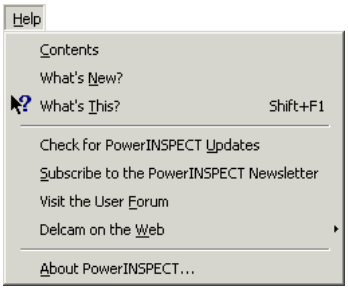
The Information Tab allows the user to view specific items for positional data, tolerances, deviation, error etc. This can be displayed by selecting the item to be investigated from the **Sequence Tree**, and pressing the **Info Tab**. The following is then seen:



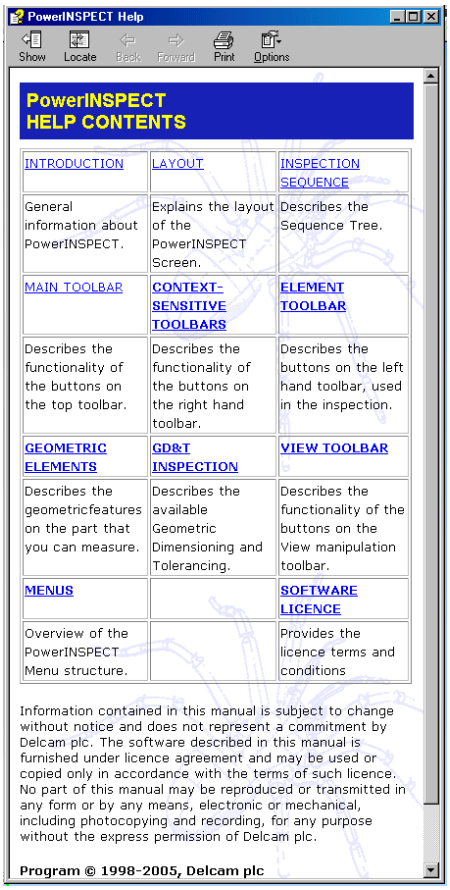
PowerINSPECT Help

PowerINSPECT comes complete with it's own **Help Document** which is accessed via the **Help Menu**. When faced with a problem, it is often best practice, to make the help menu your first port of call.

- Select **Contents** from the **Help Menu**.

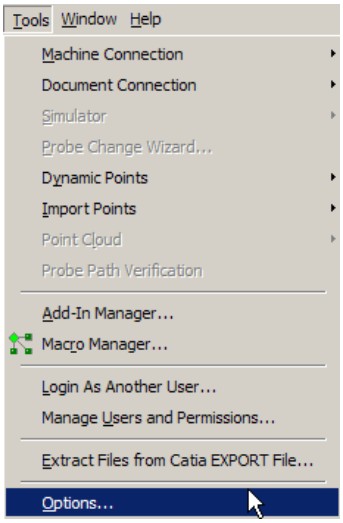


- The Contents can be viewed, with each area having a specific sub-menu. Required definitions could also be found using the **Search** tab.

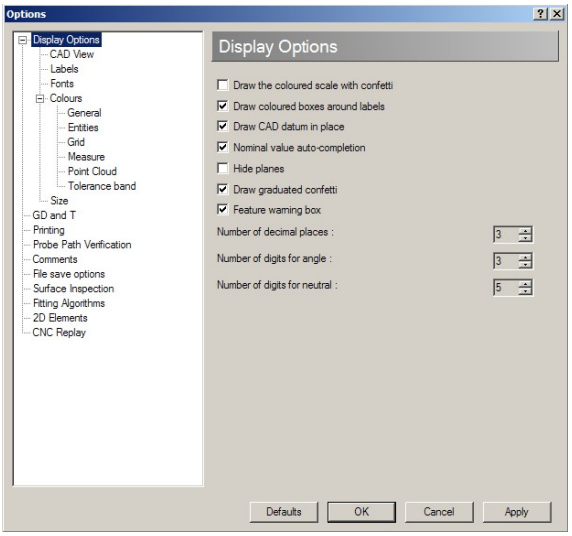


Changing The PowerINSPECT Appearance

From the **Tools Menu** (right), the **Options Menu** allows the user to change the **Working Appearance** of **PowerINSPECT**.



- From the **Tools Menu** select the **Options Menu**.



Within the **Display Options** dialogue box the user can specify a number of items, including, the **Colours** of **Entities**, the **Sizes** of **Points**, **Confetti** or **Edge Points** and toggle on or off **Displayed** items such as the **CAD Datum**.

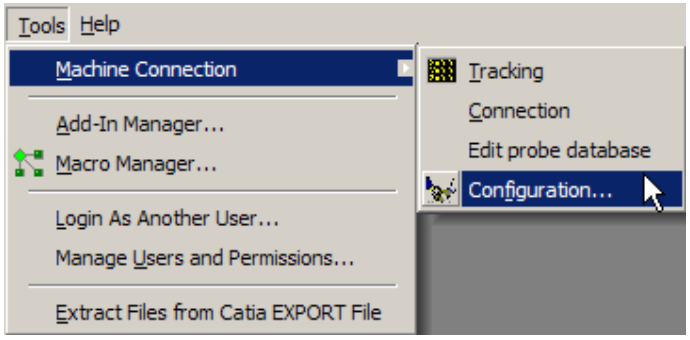
Device Connection

Users starting PowerINSPECT for the first time may need to establish a protocol connection to the inspection device

When starting **PowerINSPECT** for the first time, the **Main Configuration Protocol** dialogue box appears. Before loading up, an active connection needs to be made to some form of inspection device.

The protocol dialogue box can be opened manually from the File menu.

- Select Tools ► Machine Connection ► Configuration.



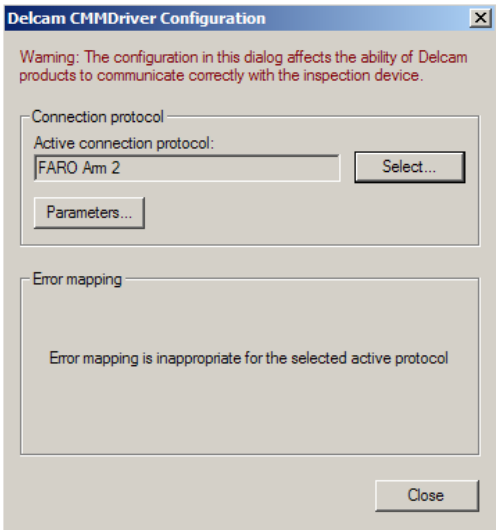
Configuration and Error Mapping

If a connection has not been set up:

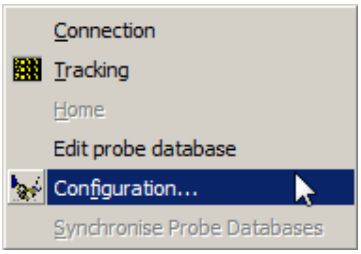
- Choose the **Select Button** to open a list of **Connection Protocols**.

- Select the required **Protocol** for the system that **PowerINSPECT** is to connect to (if unsure of this contact **Delcam Support**) and choose **Ok**.

The correct **Connection Protocol** should now be set up.



Although most users have no need to access this once initially set up, it should be noted that if the probe position is not as expected, or a connection cannot be made, checking the **Configuration** might provide some help.



Once opened the form can be checked for the **Protocol Connection** and for **Error Mapping**. Error mapping allows the CMM machine to compensate for Variations in Measurement over extended distances. For the training exercises however, the small distances moved, means the error mapping is disabled.

Note: Do not alter the connection protocol unless an experienced user. The original installation should have a connection already set up for the specific machine.

Starting Set-Up

- Select the **Toggle Machine Toolbar**  from the **Main Toolbar**.

The **Machine Toolbar** appears. Within this toolbar the user can **Connect** to  and **Home**  the machine, open the **Tracking Box**  and **Change** the **Probe** tool  . This toolbar also has a status bar, indicating the status of the CMM. In the case above it can be seen that the CMM is "**Not Connected**".

Making a Connection

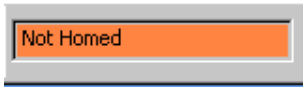
The **Connection Icon**  , also indicates **the connection** status to the CMM, and is located at the right of the **Windows Task Bar**.

- Select the **Connection Button**  from the **Machine Toolbar** (or alternatively, choose the **Connection** option from either the **Tools► Machine Connection ► Connection**.

This will attempt to connect to the CMM, if successful the Not Connected with change to a Green **Machine OK** as above. (*Note: The user will be prompted to move the probe head to **A = 0**, **B = 0** for a manual CMM*).

The user may also be prompted to move the CMM to the home position (in most cases, this means all axes moved to their positive end stops, although machines do vary).

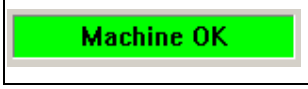
- If prompted, **Home** the **CMM** by clicking on the **Home Button** 



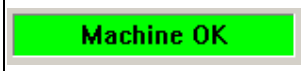
There are a number of conditions displayed in the Status Box on the Machine Toolbar.

These are summarised below.

When no document (session) is open, the **Status** box shows the status of the connection between PowerINSPECT and the measuring device:

Status	Description
	PowerINSPECT is not connected to the machine.
	PowerINSPECT is connected to the machine.

When one or more documents are open, the **Status** box shows the status of the connection between the active document and the measuring device:

Status	Description
	Neither PowerINSPECT nor the active document are connected to the machine.
	PowerINSPECT is connected to the machine, but the active document is not connected.
	PowerINSPECT and the active document are connected to the machine.

The **Status** box can also display the following information messages:

Status	Description
	The machine is not homed.
	The homing procedure has just completed successfully.

It should be noted that the exact procedure will vary from CMM to CMM and Portable Arm. You should follow the prompts given on your own machine. Once connected the **Probes** can be changed and edited.

2. CAD Management

CAD Data

The manipulation of **CAD Data** within **PowerINSPECT** is very important for the user. Using the **CAD File Manager** the user can **Add** or **Remove** CAD files, **Transform** CAD data in relation to **Workplanes** and indicate **Levels** to be included in the inspection.

Workplanes

Users, who are unfamiliar with the CAD environment, need to understand the use of **Workplanes**. The **Workplane** is the **Datum** for all the CAD creation, and positional values are taken from this **Datum** location.

While these values might be what are expected for a component with a local workplane, components created in say the **Aerospace** or **Automotive Industry** might be created in relation to a **General** or **World Workplane** (e.g. the centre of the front axle of a vehicle – car line) and values could be metres away from the Datum. This might not seem much, but because **PowerINSPECT** works in **mm**, a value of say, only **2m,** would register **2000mm**.

A graphical representation of this **Workplane** can be seen on the right.

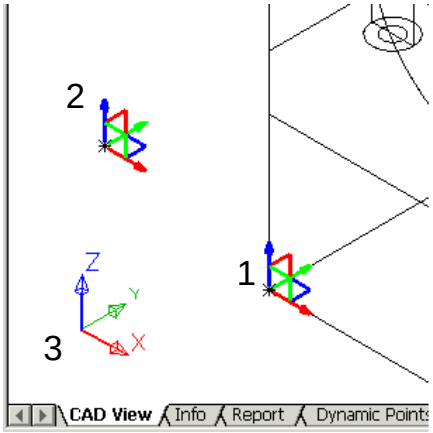
The **Workplane**(**s**) seen within the CAD view can be one of two:

1. The first is the **PCS Datum Workplane**. This is the actual world workplane that the CAD data was created in relation to. This will always lie in the same position relative to the CAD, even after Transformation.

In the picture on the right, this workplane can be seen at the corner of the part.

2. The second relates to the Machine Datum for the CMM machine being used.

This Datum appears move (relative to the CAD) once the alignment is created.

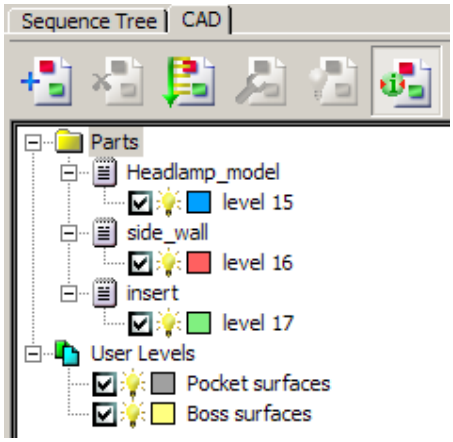


3. The third is merely a **Graphical Representation** of the workplane and, even if the world workplane lies metres away and off the screen, this workplane is always visible. This allows the user to identify the axis directions of **X**, **Y** and **Z**.

Note: All Transformations are made with respect the first Workplane.

CAD File Manager

The **CAD File Manager** is located using the **Tab** to the right of the **Sequence Tree**.



When selected the window opens up to show the **Parts** folder and the **User Levels** area. Double clicking on the item, or pressing the + icon, opens the item to a lower level.

To the right it can be seen that a **Headlamp model assembly** has been opened, and contains three parts with one level, and one part with three levels. Two further user levels have also been created.

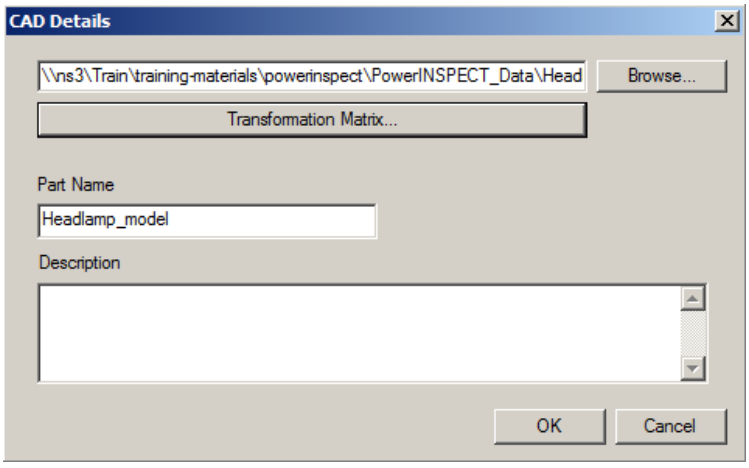
The coloured box indicates the **Wireframe** or **Shaded Colour** of the part and can be changed by left clicking on it, and assigning a new colour from the palette.

The light bulbs indicate whether the level is displayed or not in the **CAD View**, and the tick 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 corresponding toolbar appears at the top of the tab, with a number of functions that can be used to manipulate the CAD data.

This toolbar enables the user to **Add** or **Remove** CAD files, **Reset** the **User Levels**, **Edit** the **CAD Details**, **Show** or **Hide** CAD Files and **Detailed View**.

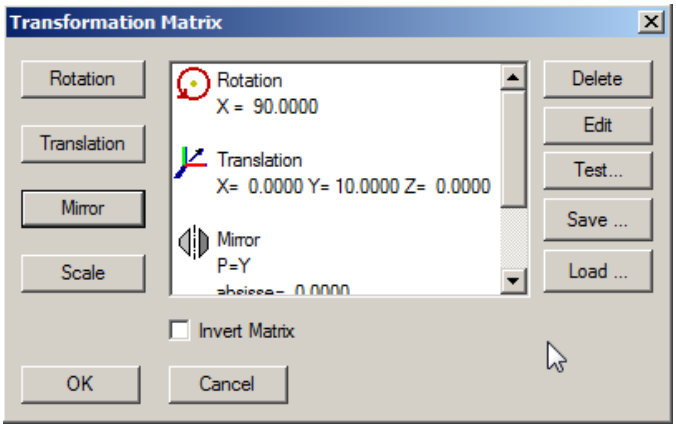
Choosing the **Add** button, or the **Cad Details** button brings up the **Cad Details** dialog box. This box allows the user to locate (using the Browse button) CAD files and perform **Transformations** on them.

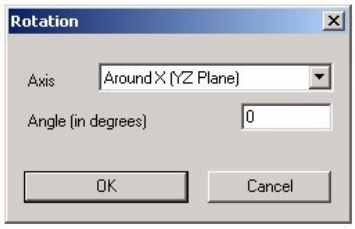


The **Transformations** are used to orientate parts within the **CAD View**.

To **Transform** any CAD data, the part must be defined in the **Filename** box.

Once the part is specified, pressing the **Transformation matrix** button will call up the **Matrix** dialog box. Within this box the user can specify any transformation (**Rotation**, **Translation**, **Mirror** and **Scale**), by clicking on the desired button and entering the desired values.



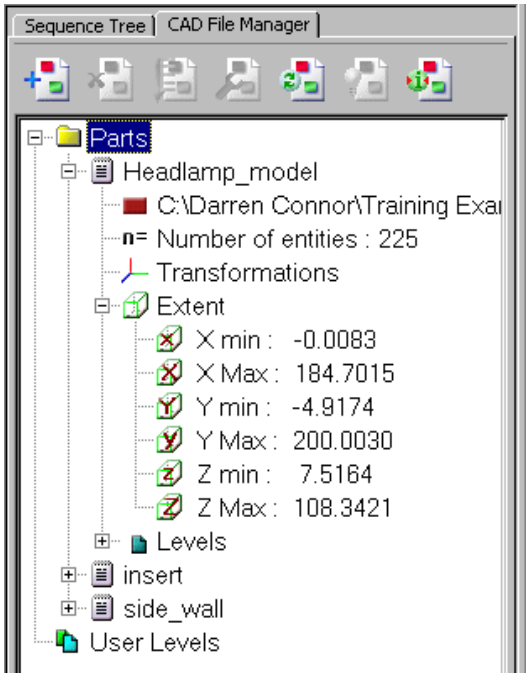


For example: Clicking on the **Rotation** button will bring up the **Rotation** box. The rotational axis can be specified (e.g. the X-axis) and the user can enter the angle they wish to rotate the part by (e.g. 90 degrees).

When the user is happy with the transformation, pressing **Ok** will enter it into the central area of the **Matrix** dialog box.

These transformations, once created, can be saved and loaded later, deleted, edited and tested all within this same box. It must be noted that, to edit or delete a transformation it must first be selected in the central area of the **Matrix** dialog box (just by left clicking on the item).

Summary and Detailed CAD View



Using the **Summary/Detailed View**  button on the **CAD File Manager** toolbar, the CAD information area can be expanded to show more information about the part files (see left).

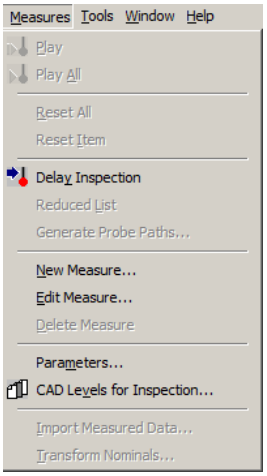
The information displayed includes any **Transformations**, the **Levels**, **Part Locations** and the **Number of Entities** that make up the part.

The **Extent** values indicate the dimensions of the part, in relation to the workplane origin.

Simply clicking on the  button again, will return to the data summary.

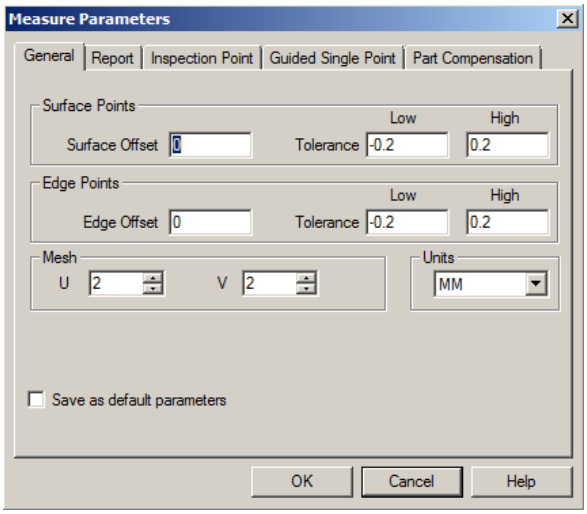
CAD Menu Options

As well as the CAD File Manager, CAD edits can be made using the some of the options in the Menu Bar.



Within the Measures menu lie two options that are relevant to the CAD management.

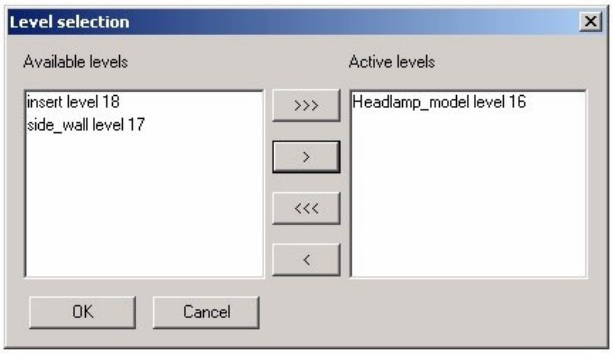
The first is the **Parameters** option, which opens up a **Parameters Edit** box, and the second is the **CAD Levels for Inspection** option.



Choosing the **Parameters** option opens up the **Parameters Edit** box. Here a series of tabs can be seen that relate to different areas of **PowerINSPECT**.

Choosing the **CAD Levels for Inspection** option, allows the user to specify which levels they wish to include in the inspection.

Choosing an item and clicking the **Selection Arrow Buttons** moves the item in the chosen direction.

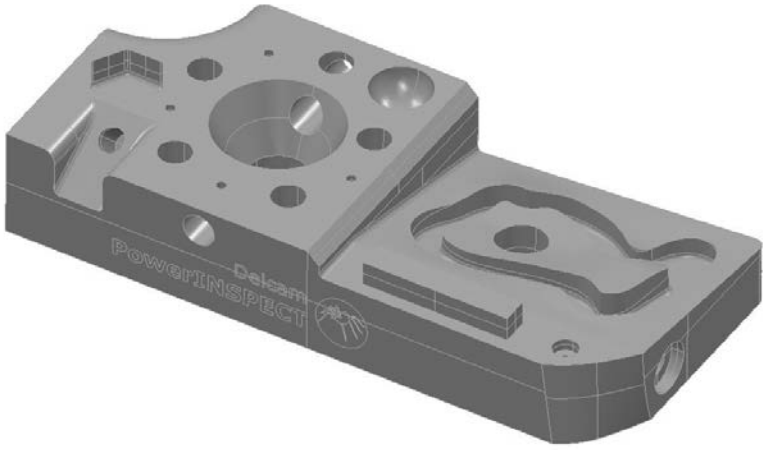


The < and > buttons move singular items, and the <<< and >>> buttons move all items in the chosen direction. The levels can be either predefined by the CAD or created within **PowerINSPECT**.

Level Edits

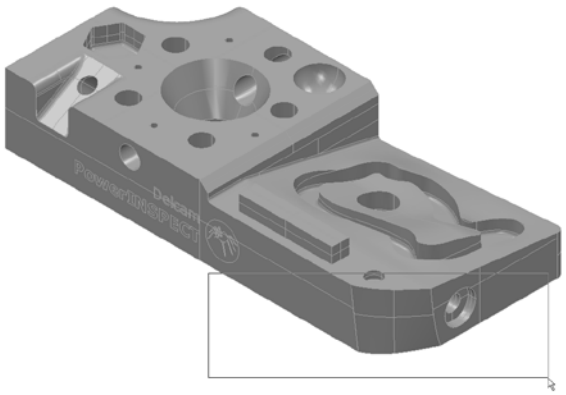
CAD data within **PowerINSPECT** can be move around to different levels. This is done using a combination of a selection tool and an option called group surfaces.

By choosing the **Surface Selector Button**  in the **Mouse Context Toolbar**, the user can specify which surfaces they wish to move and to which level.

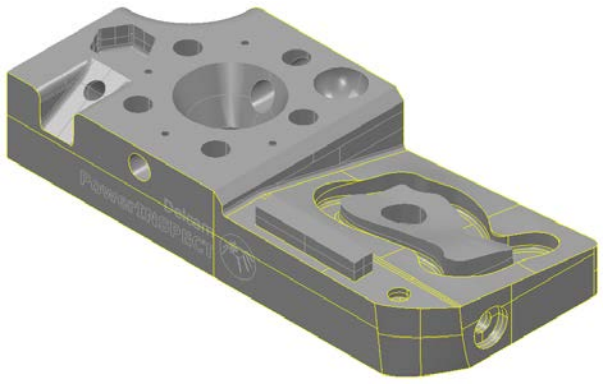


Pictured above is a standard view of a part, with the **CAD File Manager** open. Note it has a single level created in the file.

To create a new level, choose the **Surface Selector Button**  in the **Mouse Context Toolbar** and draw a box around, individually pick or Shift Click the surfaces that require moving (below left). These then become highlighted (below right).

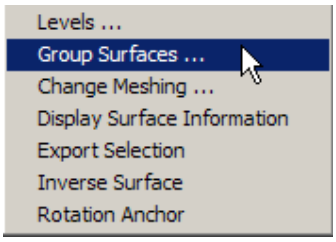


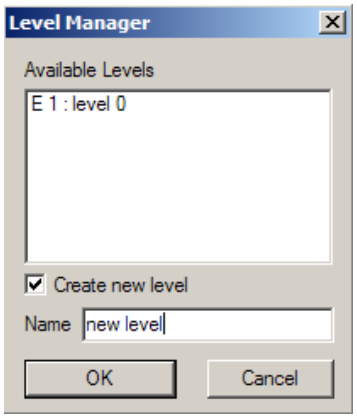
Create a Selection Box



Highlighted Surfaces

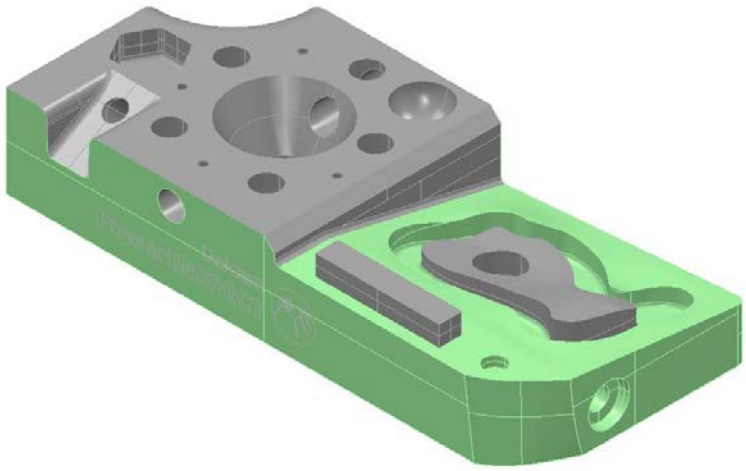
By right clicking on the selection and choosing **group surfaces** from the local menu (see right), the corresponding dialogue box (see below) that appears allows the user to place the selected surfaces on a different level.





The user can place the surfaces on either an existing level, or in this case, a fresh level. This fresh level (see left) has been called “New Level”, and the tick box confirms its creation.

Once created the level appears in the user levels area of the **CAD File Manager**, and can be switched on or off using the light bulb (see below).



This technique can be very useful if there is a large quantity of data in a view. Each layer has an associated colour, used when colour shading the model. The shading colour for a level may be changed by clicking on the small coloured box to the left of the level’s name. The new colour is then picked from the palette that appears.

3. Alignments

What is an Alignment?

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 normally dictated by the shape and which features can be used.

Alignments are accessed by selecting the Alignment Sub-Menu  from the Element Toolbar.

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

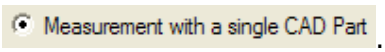
3.1 Geometric PLP Alignment 

Introduction to 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 – The Headlamp Example

For this example the Headlamp CAD file is going to be used. Using the methods, previously described in the course, for starting a new session:

- Create a **New Document using the Wizard**  and select the method 

.
- **Browse** and **Load** ***DemoBlock2008.dgk***.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**

In the Variables dialog box:

- **Browse** for any chosen **HTML Report Template** (**Excel** could be used, but for this example **HTML** is to be used).

- Select **Finish**.

The new session is now ready for inspection.

Choosing the Geometric Elements

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

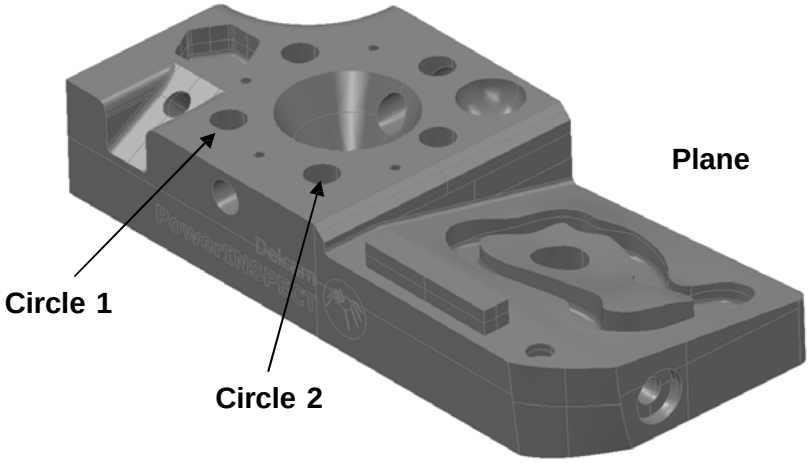
With any **Alignment** the results are better if a greater area spread is used to define the aligning elements. For this reason, the **CMM Bed** or **Table** will be used to define the **Plane**, as the model has a flat base, and all model planes are small.

The Line

Lines can be defined from square edges (**Probed Lines**) or from the **Connection**, or **Intersection**, of **Measured Elements**. In this particular example the line is to be defined in the **X**-direction, using the **Centres** of **Two Measured Circles** on the part. ***Note: The line direction in PLP must be axial.***

The Point

Because the **Circles** will have already been defined, the **Centre Position** of one of these will be used for the **Point Position**.



To the left is the **CAD File**.

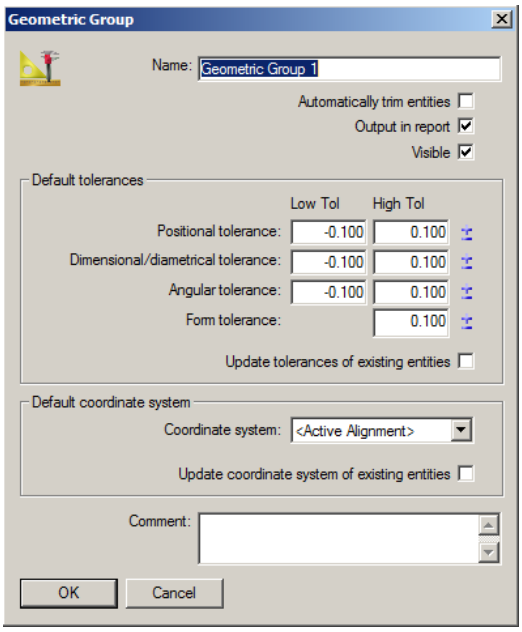
The **Two Circles** chosen are indicated (left), and the **Plane** will be the base area (shown). The **Line** will be between the **Centres** of **Both Circles**, and the **Point** will be taken from the **Centre** of **Circle 1**. *Both circles and line will therefore be projected onto the base (plane)*

Creating the Geometric Elements

With the position of the elements chosen it is now necessary to define the **Geometric Elements** to **Probe**.

- From the **Element Toolbar**, choose the **Geometric Group**  Button.

The Geometric Group dialogue box appears and prompts the user to name the group and add a comment if required.



- Name the group **Geometric Group 1**, and **Untick** the **Output in Report** box.

- Leave everything else as **Default** and choose **OK**.

*Note: The **Output in Report** box was unticked because we are going to do an alignment which is not required in the report.*

The **Element Toolbar** automatically changes to the **Geometric Group Toolbar**, and it is from this that the **Geometric Elements** will be created. However, before the elements are created, the measurement should be delayed to allow the user to create ***all*** the elements first.

- Choose the **Delay Measure Button**  from the **Main Toolbar** or **Measures Menu**.

This allows the user to **Pause** the measurement until told. Otherwise, when the **Free Form Alignment** is created, **PowerINSPECT** will automatically move to the probing process.

The **Geometric Group Toolbar** (left) is split into eleven areas. For the purposes of this example only three will be used.

These are the **Planes**  , **Lines**  and **Features**  .

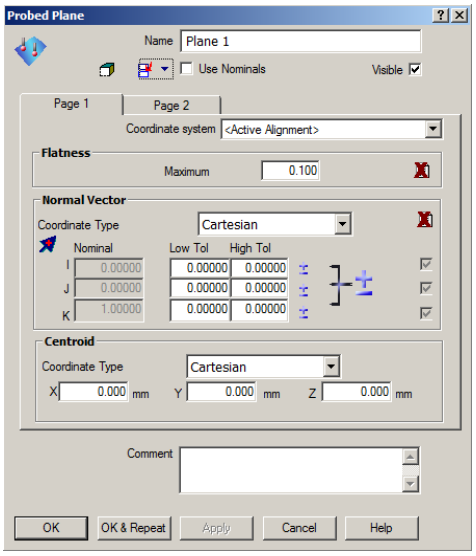
These satisfy all the criteria for the **Geometric PLP Alignment**.

The Plane

The first geometric item to be created will be the plane.

- Select the **Planes Button**  , and choose the **Probed Plane**  .

The **Probed Plane** dialogue box appears, and prompts the user to give the plane a **Name**, set the **Flatness Values** and **Normal Vectors**, and its **Centre Position**. The user can also add a **Comment** if required.



Because this plane is to be probed, these values can be left as default, and the probing process will determine them.

- Enter the name **Plane 1** leave the other settings as **Default**.

- Choose **OK**.

It can now be seen that a **Plane Icon**  has appeared in the **Sequence Tree** an **Exclamation Mark**  next to the **Plane**, signifies that the alignment has not yet been measured.

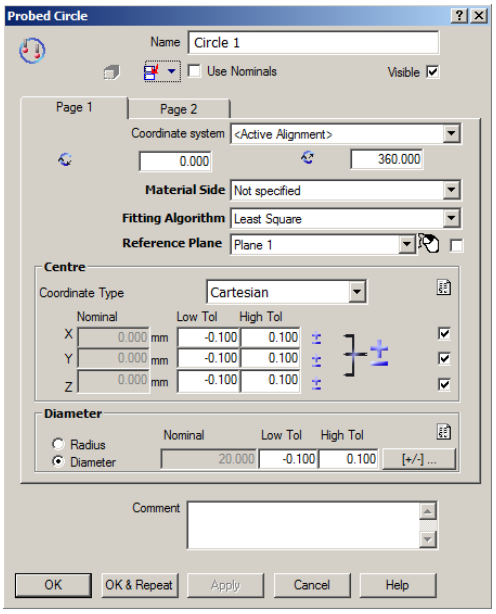
The Circles

The **Two Circles** are required next, and creating them follows much the same process as was used to create the **Plane**, only from a different menu.

- Select the **Features Button**  , and choose the **Probed Circle**  .

The **Probed Circle** dialogue box appears, and again prompts the user to give the circle a **Name**, its **Centre Position** and a **Comment** if required.

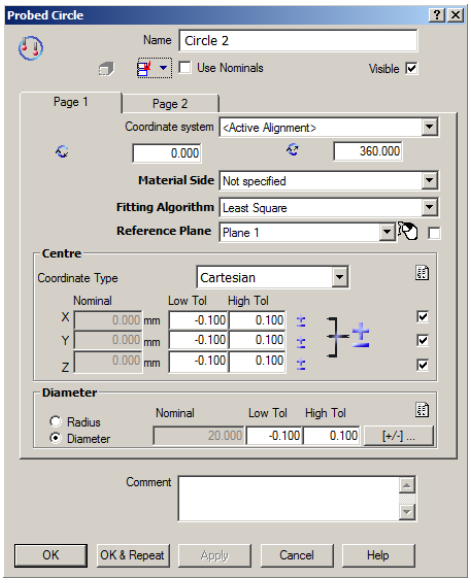
The **Circle** form also asks for an **Offset** or **Thickness** value, a **Plane** to **Reference** to and the **Diameter**, **Radius** and **Circularity**. (On page 2 of form)



- Enter the name **Circle 1**, and set the **Reference Plane** as **Plane 1**. Leave the other settings as **Default**.

- Choose **OK & Repeat**.

This option allows the user to accept the form and create a new item of the same type without the need to close the form and re-choose the item.



Once the new form comes up (it will be identical to the previous form except for the name) fill in the following values.

- Enter the name **Circle 2**, and set the **Reference Plane** as **Plane 1**. Leave the other settings as **Default**.

- Choose **OK**.

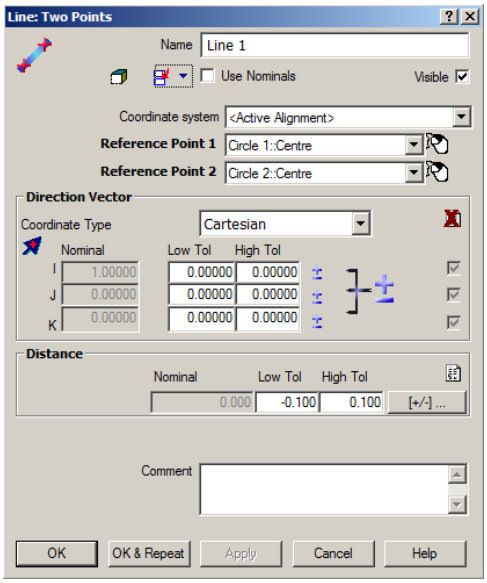
The two items have now appeared in the **Sequence Tree**, named **Circle 1** and **Circle 2**, and again they have the **Exclamation Marks** because they are unmeasured.

The Line

The final **Geometric Item** that will be created is the **Line**. As highlighted before, the **Line** is to be created between the **Centres** of the **Two Circles**, and therefore the **Line Between Two Points** option is required for the item.

- Select the **Lines Button**  , and choose the **Line: Two Points**  .

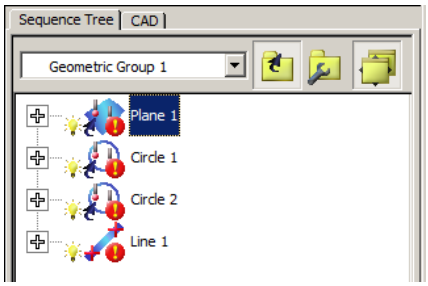
The **Line: Two Points** dialogue box appears, and is similar to the previous boxes. It prompts the user to give the line a **Name**, its **Direction Vector** and a **Comment** if required.



The form also asks for a **Distance** and **Two Reference Points** to pass the line through.

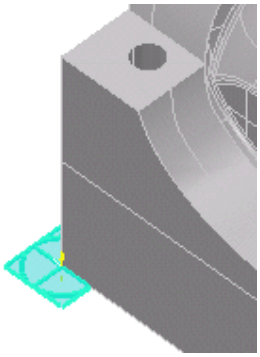
- Enter the name **Line 1**, and set **Reference Point 1** as **Circle 1: Centre**, and **Reference Point 2** as **Circle 2: Centre**. Leave the other settings as **Default**.

- Choose **OK**.



The **Line** has now been created and the **Sequence Tree** has been updated to include ***all*** the **Geometric Items** (left).

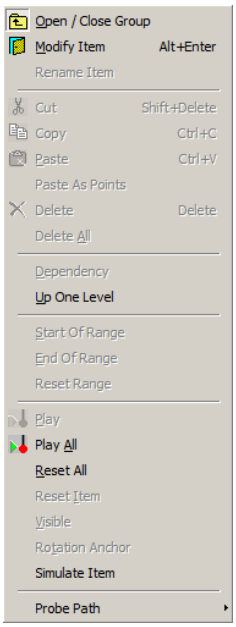
In the **CAD View**, all the items can be seen, located at the origin (see right).



*Note: This is the **Pre-Probed** default position set in the forms.*

Probing The Geometric Items

Once the **Geometric Items** have been created, and the **Part** is secured to the **CMM Bed/Table**, the probing process can begin.



- **Right Click** over ***Any*** **Item** in the **Sequence Tree** to bring up the **Local Menu** (left).
- Choose the **Play All** option.

Alternatively

- Choose **Play All**  from the **Main Toolbar**, or **Measures Menu**.

Note: Choosing the Play  *option will only play the selected element. Also, choosing Play All will only play* ***unmeasured*** *items.*

PowerINSPECT will progress into the **Probing Screen**, prompting the user to start probing the indicated **Geometric Item**.

Full Screen Options Toolbar

When moving into **Full Screen Measure Mode** a new toolbar appears at the right of the screen (by default – see right). This is called the **Full Screen Options Toolbar** and contains the functions that are common to all measurement modes.

The **Toggle CAD Context Display Button**  , switches on and off the CAD levels that are active for the user to probe, and the **Edit CAD Levels Used For Inspection Button**  allows the user to choose the levels used.

The **Toggle Previous Measure Display Button**  , switches on and off all previously measured items in the view, and the **Change Probe Button**  , opens up the **Probe Database Editor**, allowing the user to change probes and positions.

The **Tracking Box Button**  opens the **Tracking Box**, which displays the probe position in terms of CMM or CAD coordinates, and the **Help Button**  displays help about the **Probing Box.**

Feature Probing Dialogues

Also, when in **Full Screen Measure Mode** a **Feature Probing Box** appears which contains a series of button options across the top of it. At any stage during the probing process the user can utilise these options available to them, which vary depending on the items being probed. For Geometric items, the following functions are seen.

The first button is the **Reset All Button**  , and simply resets all the points taken for the currently active item. The second button is the **Remove Last Button**  and allows the user to delete the details of the last probed point, so that it may be probed again. This is useful when a mistake has been made and the point needs to be re-probed.

The third button is the **OK Button**  and allows the user (when they have finished probing the points or feature and are satisfied that they are accurate) to exit back to the main window, or simply accept the currently active item.

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

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

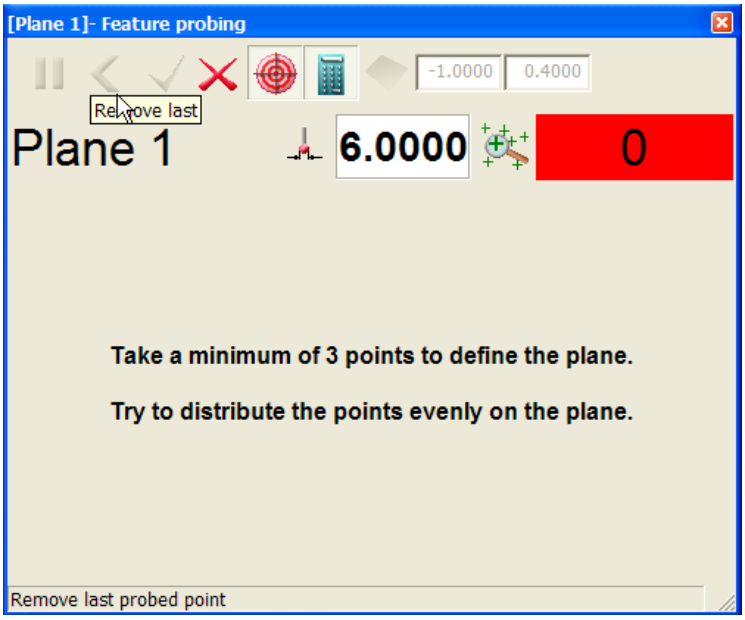
Auto calculate feature. 

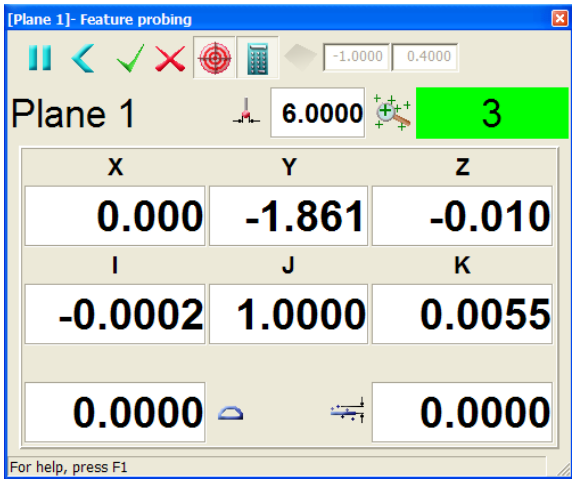
Feature Probing

The first **Feature Probing** box appears asking the user to take a minimum number of points to define that feature.

In this instance **PowerINSPECT** is asking for **Three Points** to define the **Plane**.

Note: The number of points taken is indicated at the top right hand side of the box.





Using the **CMM Bed** as the **Planar Surface**:

- Probe **Three Points** around the **Part**, on the **CMM Bed**.
- When the points have been probed select the **Green Tick**  to **Accept**.

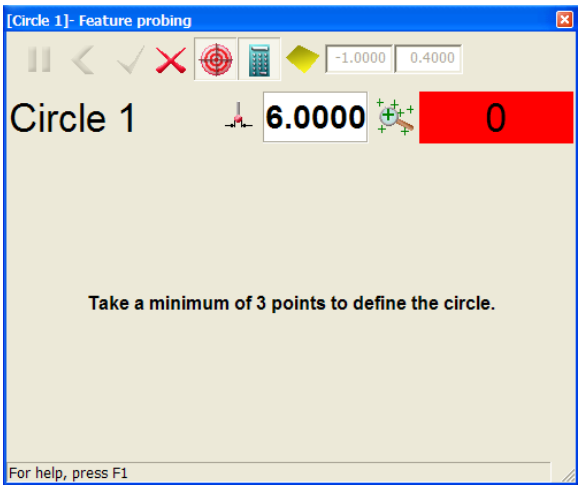
The process will then proceed to the next **Geometric Item**.

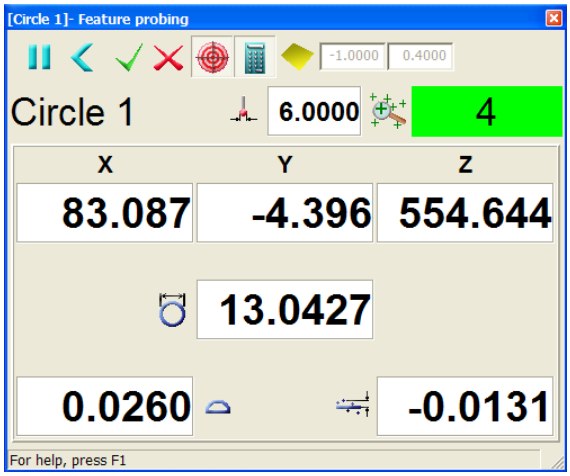
With the **Plane** probed, **PowerINSPECT** moves to the next item.

The second **Feature Probing** box appears asking the user to take a minimum number of points to define **Circle 1**.

However, rather than the minimum three points, **Four Points** will be taken for the circles, at the relative **North**, **East**, **South** and **West Positions**.

This makes it easier for the user to place the probe, and gives an even spread of points.





Using the **Part** itself:

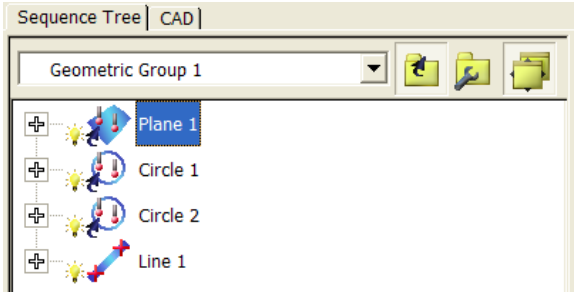
- Probe **Four Points** inside the **13mm Diameter Circle** highlighted in the screen.

- When the points have been probed select the **Green Tick**  to **Accept**.

*Note: The probed diameter of **13.0427mm** (approximately the **13mm** expected).*

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

Once all the probeable items have been probed, the **Sequence Tree** is updated (right), with all the items having lost their **Exclamation Marks**.

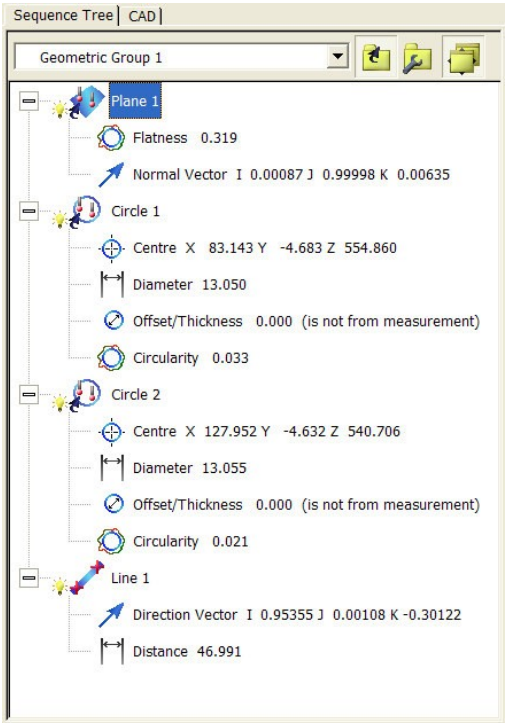


It should be noted that the **Line** was *not* probed but its **Exclamation Mark** has gone.

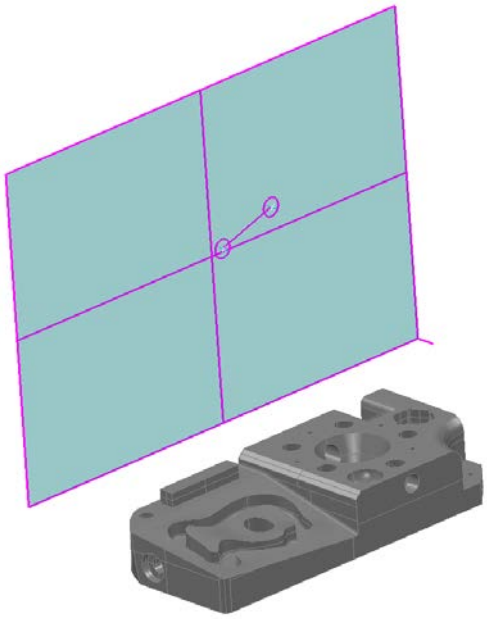
This is due to the fact that the **Centres of the Two Circles** determined the **Line's** position. These have been probed, and therefore satisfy the measurement conditions.

*Note: The Point will be defined as the **Centre** of **Circle 1**.*

Double Clicking on each item in the **Sequence Tree**, or clicking on the  **Icon**, will expand the item, revealing the basic information about it (see below left).



The un-aligned items can be seen below, with **Purple Wireframe**, and **Blue Shading**, in the **CAD View**.



Note: If the probed items can't be seen, press the  ***Button*** *to resize the screen to fit, because the items may lie some distance away at this point.*

A **Geometric Alignment** now needs to be created to align the **Probed Positions** with the **CAD File**.

Generating the Geometric PLP Alignment

In order 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 File**, and orientate them in terms of the **Cartesian X**, **Y** and **Z Coordinates**.

The deciding factors will be the **Plane Height** and **Normal Orientation**, the **Line Direction**, and the **Point Position**.

If these values are unknown then the **Geometry Explorer**  may need to be used to select the **Feature Locations** (more about this later in the course), however for this particular part these values *are* known.

The **Plane** is to be set as the **Z-Plane**, at a **Height** of **-45mm** *Note: All items have been projected on to this plane*), and the **Line** follows in the **X+ Direction**. Finally the **Point** is the position of **Circle 1** which is **X = 56.489, Y = 22.640, Z = -45**.

A **Geometric PLP Alignment** can now be created using these values.

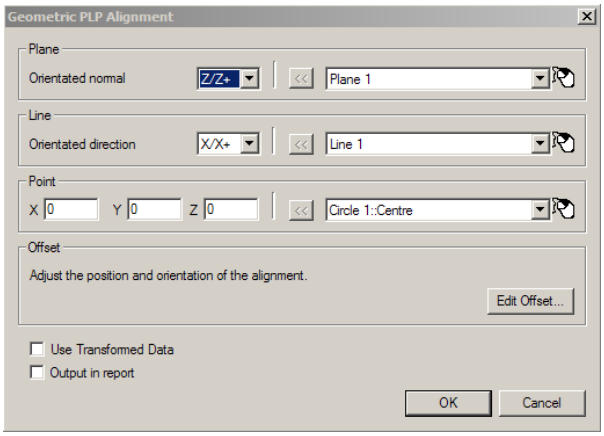
- Move up the levels  until **Geometric Group 1** is **Closed**.

- From the **Alignments Sub-Menu** icon  , in the **Element Toolbar**:

- Choose the **Geometric PLP Alignment** button  .

The **Geometric PLP Alignment** dialogue box appears and prompts the user to define the **Plane**, **Line** and **Point**.

Note: Where there is only one item (e.g. ***One Plane****), the pull down menu on the right hand side will only have the one option.*



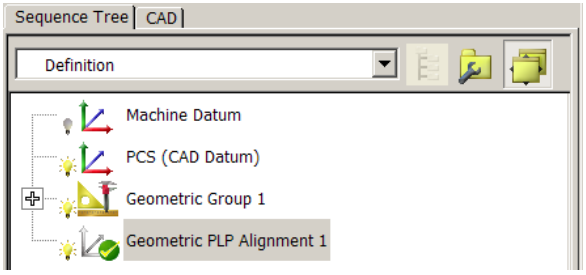
- From the pull-down menu on the right hand side, set the **Plane** as **Plane 1**, and set the **Orientated Normal** to the **Z/Z+ Direction**.

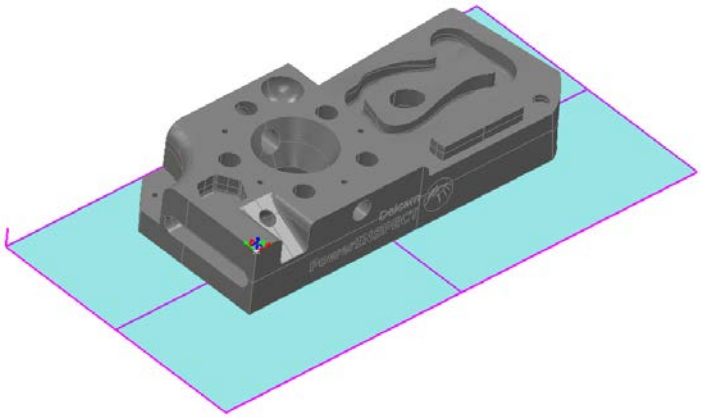
- Set the **Line** as **Line 1**, and set the **Orientated Direction** to the **X/X+ Direction**.

- Finally, set the **Point** as **Circle 1: Centre**, and the **Point Coordinate** values as **X = 56.489, Y = 22.640** (Circle 1 position) and **Z = -45** (Plane height).

- Choose **OK** to accept the form.

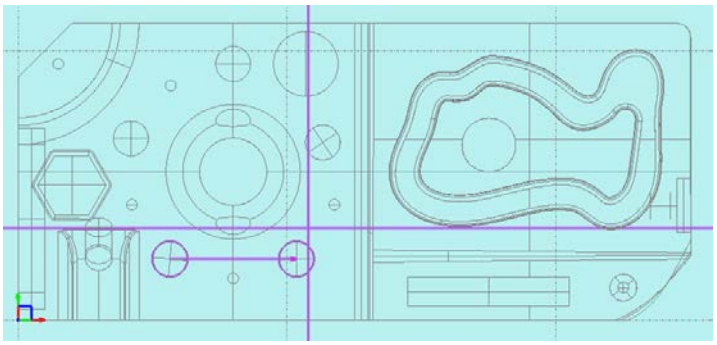
All the criteria for the **Geometric PLP Alignment** have now been satisfied, and the alignment can be seen in the **Sequence Tree**.



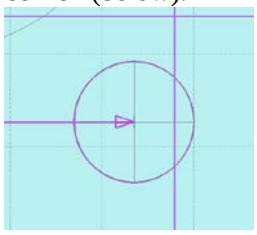


To the left it can now be seen that the **CAD View** has been updated, with the **Geometric Alignment** now positioned correctly.

To the right is a view of the model from **Above** (looking down the **Z axis**).

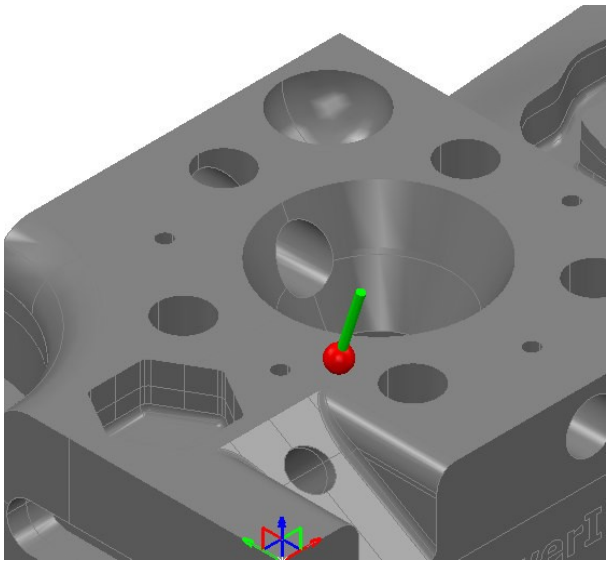


This clearly shows the alignment, which can be seen more closely in the **Zoomed View** of the bottom right hand corner (below).



To test 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 seen here, points to either an incorrect alignment, a failure in the probing of the items or an incorrect (or poor) part in relation to the CAD.



Assuming that the alignment is correct, a **Surface Inspection Group** needs to be produced, for the **Report**. The creation of these **Surface Inspection Groups** is covered later.

- **Save** the **File**  in any chosen location (e.g. **C:\Temp**).
- Name the file **GeometricPLP.pwi**

3.2 Free Form Alignment 

Although **Free Form Alignment** is considered to be the least accurate method of alignment (because it relies heavily on the skill of the user) 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 file is going to be used. Although there are a number of geometric features on this part, these will be ignored to assume there are no features. Using the methods previously described in the course for starting a new session:

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**.

In the Variables dialog box:

- **Browse** for a **Report Template** (**Excel** or **HTML**) and **Extract Variables**.
- Select **Finish**.

The new session is now ready for inspection.

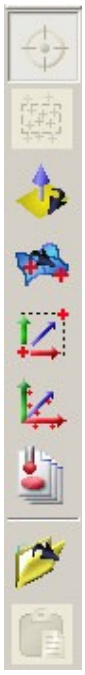
Choosing the target positions

When performing a **Free Form Alignment** the user needs to study the CAD file, and/or the physical part, carefully in order to gauge where to take the probed positions. For a **Free Form Alignment** the part needs to be effectively 'held' in position in all three axes (**X**, **Y** and **Z**). This holding should occur such that the part cannot move in a space through any translation or rotation.

Taking the **CMM Bed** as the surface to 'pin' the part down to, the user needs 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 **Seven** will be used), **3** in the **Z** direction, **2** in the **X** direction and **2** in the **Y** direction. It is important for the user 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 guide the user when aiming for the probing positions.

Once these positions have been decided it is time to create a **Free Form Alignment**.

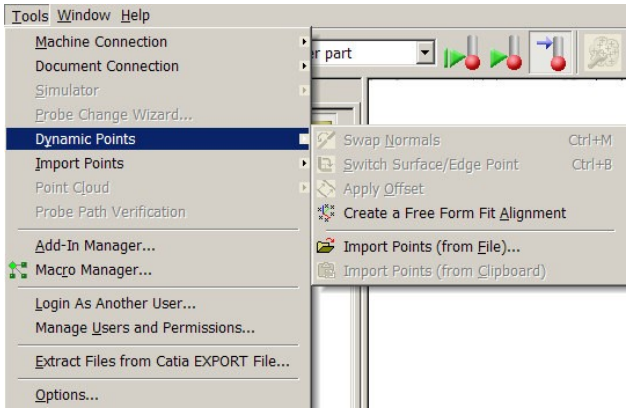
- From the **Alignments Sub-Menu** icon  , in the elemental toolbar:
- Choose the **Free Form Alignment** button  .



Once this button has been selected the **Dynamic Points Editor Toolbar** (left) appears on the right-hand-side of the screen.

It is this toolbar that allows the user to create the **Free Form Target Points**, and manipulate them in terms of their **Positions**, **Normal Directions** and **Offset Values**.

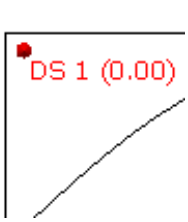
*Note: This toolbar can also be accessed from the **CAD View** toolbar using the **Dynamic Points Editor Button**  , and it's edits from the **Tools Menu** (right).*



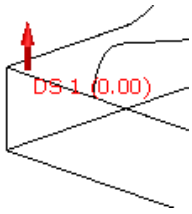
- The **Target Button**  should be selected by default.

Holding the cursor in the **CAD view**, shows it has now changed to a **Target Sight**. This allows the user to select the points on the CAD surface that are going to be aimed for in the probing process.

Points are selected by simply **double clicking** on the surface of choice. To the right it can be seen that the representation of the point can vary according to the view.



Directly Above View



Isometric View

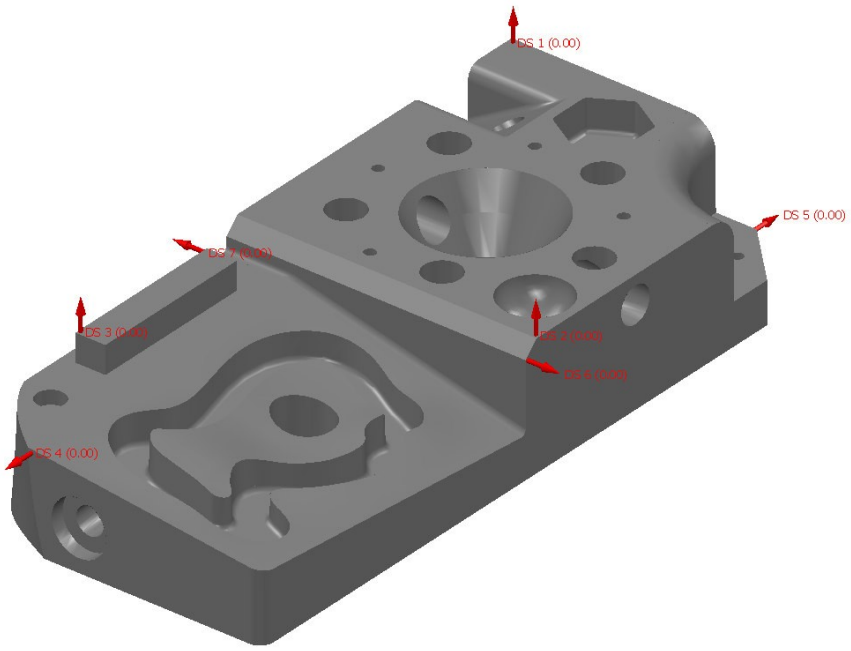
- Carefully choose a series of key points (in this case **Seven** - remembering **3** in **Z**, **2** in **X** and **2** in **Y**) on the **Physical Part** surface that can be easily aimed for. Use **Key Features** to aim for such as **Lines**, **Indents**, **Corners** and even **Projected Aiming Positions** on the part.

The points chosen on the **Physical Part** now need to be re-produced on the **CAD** file within **PowerINSPECT**.

- Replicate the points chosen on the **Physical Part** within the **CAD View**.

Do this by double clicking the **Target** cursor on the chosen surfaces to define the **7 Points** for alignment.

With the **Target Points** selected, the CAD View should look something like this:



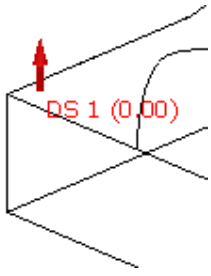
Although these points have been selected, they can still be manipulated in terms of their position and orientation, or even deleted and re-chosen.

Holding the Target Cursor over any of the chosen points, changes the cursor to a hand  , and allows the user to select and move the point around the surfaces using **Dynamic Points Selection**. This can aid the user finding probed positions that are more easily aimed for (lining up with features etc.).

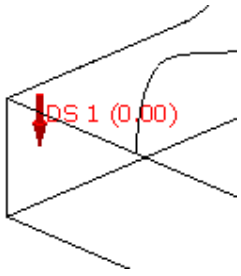
The **Dynamic Points Selection** also lets the user choose the points to be manipulated using the other **Dynamic Points Editor Toolbar** functions.

Using the **Swap Normals Button**  allows the user to reverse the direction of the points.

This can be useful if the user-defined point has been created in the wrong direction, such that the probed point is in an inaccessible position (e.g. the underside of a surface).



Normal To Surface

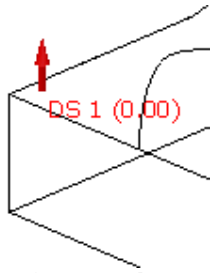


Normal Reversed

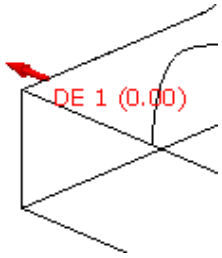
Using the **Switch Surface/Edge Point Button**  allows the user (if the point chosen is close enough to the edge) to flip the surface point onto the edge.

This would be used when edge points are required rather than surface points.

Note the change from an S (Surface) to an E (Edge) on the point label.



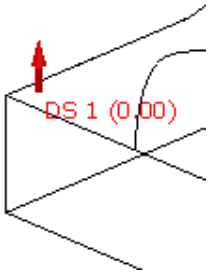
Surface Point



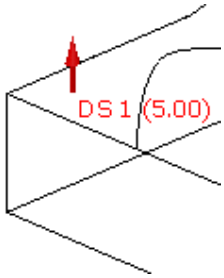
Edge Point

Using the **Apply Offset Button**  allows the user to set an offset value for the probed position.

This is especially 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 (i.e. the **Offset**)



Without Offset



With Offset (Value in Brackets)

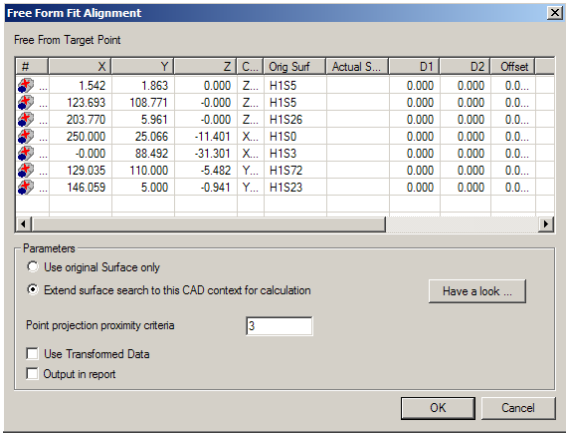
Once the **Target Points** have been selected and manipulated the **Free Form Alignment** can be created ready for probing.

- Choose the **Delay Measure Button**  from the **Main Toolbar** or **Measures Menu**.

- Select the **Create Free Form Alignment Button**  from the toolbar.

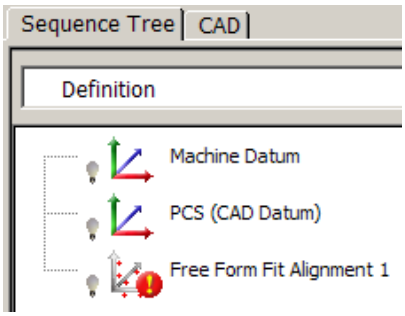
Free Form Alignment dialogue box appears indicating the **Positional Coordinates** of each chosen point, the **Surfaces** they lie on, and the **Offset** values.

Within this box the user can specify the **Point Projection Proximity Criteria**, which defines the surface area of search around the probe.

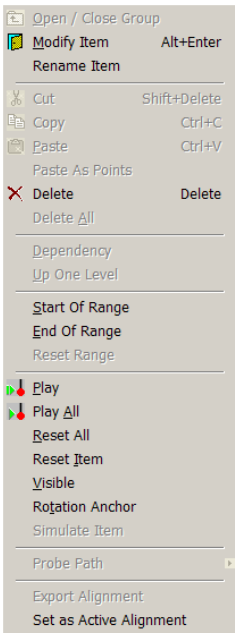


- Increase the **Projection Proximity Criteria** to **3mm** and choose **OK**.

Looking at the **Sequence Tree** a new item can be seen in the **Definition** area.



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



Note the Exclamation Mark  *next to the **Free Form Alignment 1**, which signifies that the alignment has not yet been probed or played.*

- Select the **Free Form Alignment 1**, and right click on it to bring up the **Local Menu** (see left).

- Choose the **Play** option.

*Alternatively use the **Play***  *button from the **Main Toolbar**.*

This moves the user into the **Probing Screen** allowing the user to probe the pre-selected **Target Points**.

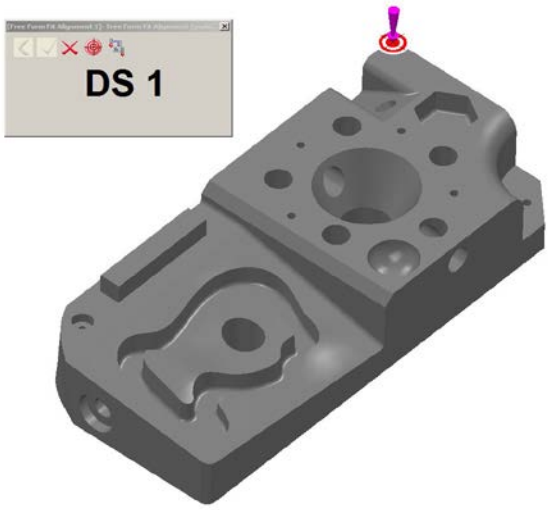
Probing The Part

Once into the **Full Screen Measure Mode** (and with the **Part** secured) the **Probing Process** can occur.

Within this screen a **Probing Box** appears, with the target number shown (e.g. **DS 1**) and a series of probing option buttons, similar to those seen in the **Geometric PLP Alignment** (**Chapter 4**). These buttons are identical, except that the **Reset All** has gone, and the **Parameters Button**  (which shows the **CAD Context** and **Proximity Criteria** for the probed points) is now shown.

The **Dynamic Point Targets**  appear for the user to aim for (see right), and this is where the skill of the user comes in, getting as close to the targets as possible.

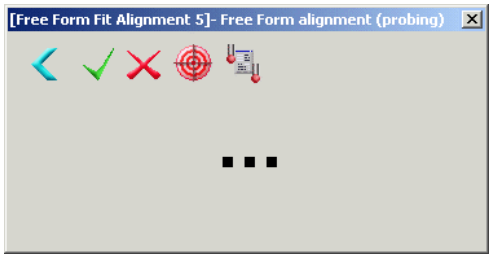
They appear in sequence order and as each one is probed the next one appears on the screen.



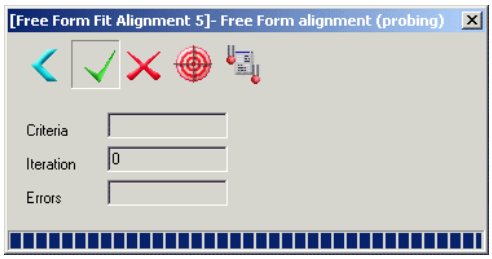
- Probe the **7 Points** to complete the alignment.

Note: Always remember to 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.

Once all of the points have been probed, the **Probing Screen** indicates that there are no more points to take (see right).



- When the points have been probed select the **Green Tick**  to **Accept** the **Alignment**.

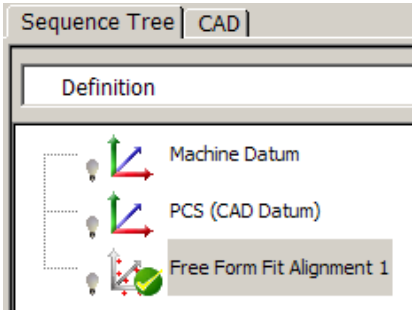


PowerINSPECT will then run through a series of iterations, to calculate the **Alignment**, with respect to the points probed by the user.

The form to the left demonstrates this, and once completed, the will automatically exit the **Full Screen Mode**. The less time this takes, the better the alignment tends to be.

The Free Form Alignment has now been created.

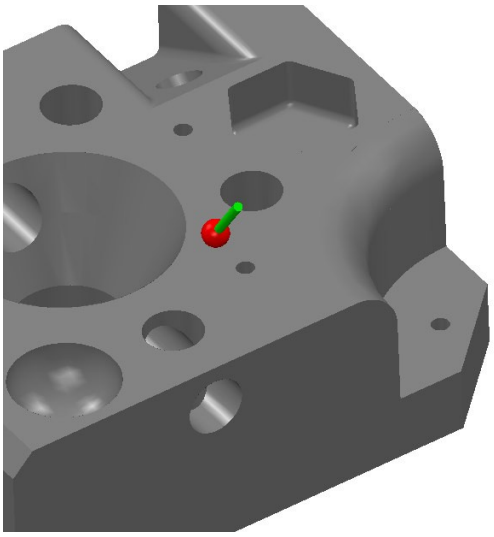
Note: The Sequence Tree has been updated  *, and the Exclamation Mark*  *has disappeared (see right).*



The **Demoblock2008** 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 (see right).

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

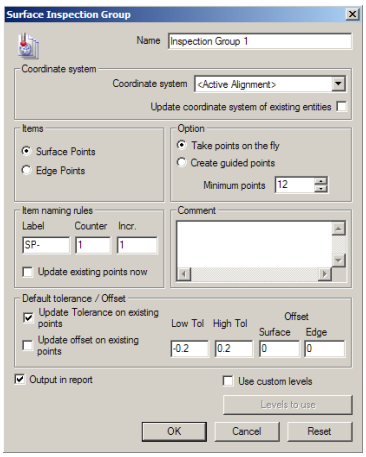
Once the part has been correctly aligned to the CAD file the next stage is to create a **Surface Inspection Group**. This first group will be used to check the accuracy of the **Free Form Alignment**, and be used to improve the alignment using the **Best Fit Analyser**. It is therefore not necessary to include this initial group in the **Report**.

Note: Surface inspection will be covered in greater detail in the next chapter

- Make sure the **Delay Measure Button**  is pressed **On**.

- Create a **Surface Inspection Group** using the  button.

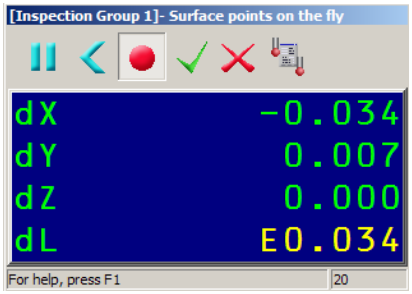
- Keep all the **Default Settings** (except *untick* the **Output in Report** option and set the **Minimum No. of Points** as **12**) in the **Surface Inspection Group** form and choose **OK**.



- Right click over the **Surface Inspection Group Icon**  that has now appeared in the **Sequence Tree** and select **Play**.

When the **Probing Screen** appears:

- Probe **12** surface points on the part, **4** in **Z**, **4** in **X** and **4** in **Y** keeping a wide spread.



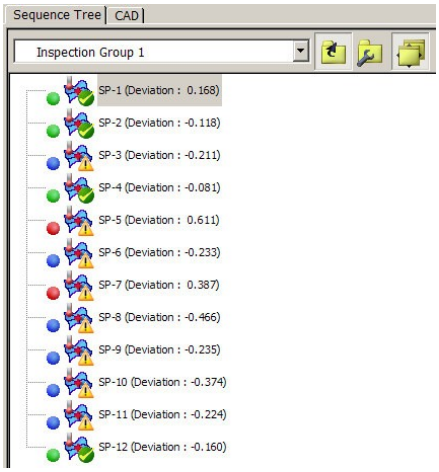
As the points are probed, the **Surface Points on the Fly** data box displays the deviation of the measured points.

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

- Once **12** points have been taken, choose **OK**  .

It can be seen that the **Sequence Tree** has now been updated, with the **Surface Inspection Group** icon  having lost the **Exclamation Mark**
Once the **Surface Inspection Group** has been created, a quick check needs to be made of the Group.

- Open the **Surface Inspection Group** to verify that **12** points have been taken (if less than **12** were taken, then the **Exclamation Mark** will remain).



- Note the **Spread** of the deviation of the points.

If the points seen in the **Surface Inspection Group** have a wide deviation, it may be necessary to re-probe the alignment. However, 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) then this points to a slight **Translation**. Similarly, opposites around an axis will indicate a **Rotation**. If this sort of pattern can be seen, then a **Best Fit** will improve the results.

Generating a Line Of Best Fit

Because of the high dependence on the skill of the user, a **Free Form Alignment** may need some fine-tuning in order to generate a better alignment. **PowerINSPECT** contains a **Best Fit Optimisation** option, which allows the user to generate a line of best fit between the probed points, and hence align the **CAD Data** and the **Physical Part** more accurately.

*Note: 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 to this it should only really be used once on each **Free Form Inspection**.*

For this example a **Best Fit Optimisation** is going to be applied to the newly created **Surface Inspection Group**.

- Move **Up One Level**  in the **Sequence Tree** so that the **Surface Inspection Group** is closed. Highlight this inspection group and choose the **Best Fit Optimisation Button**  .

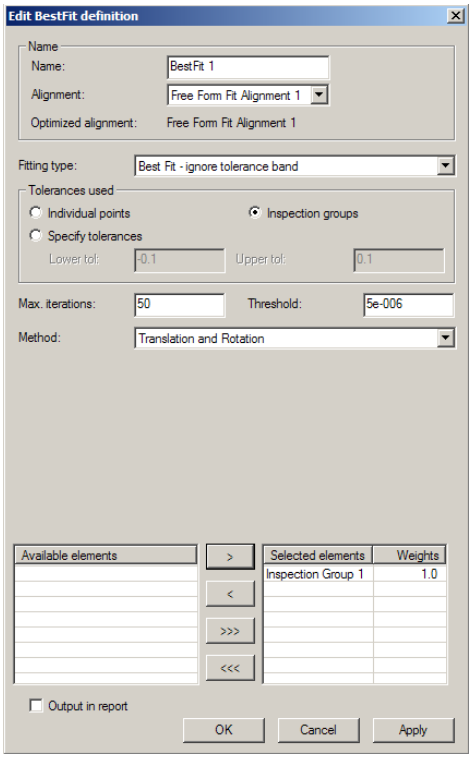
This will open the **Edit Best Fit definition** dialogue box.

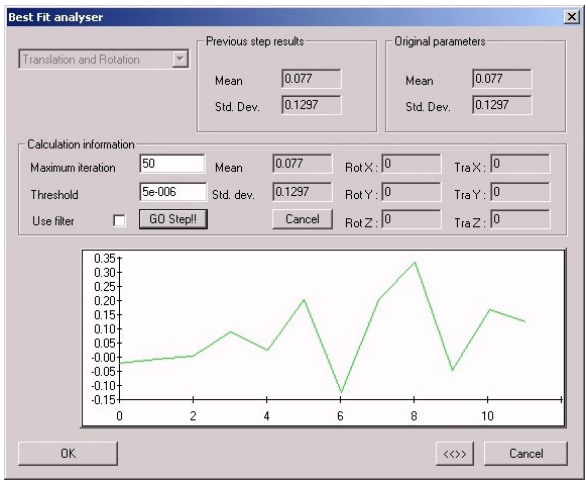
This box allows the user to **Name** their **Best Fit line**, define the **Application Method**, and choose the **Elements** they wish to apply it to.

- Name the **Best Fit BestFit 1**.
- Set the application method to **Translation and Rotation**.
- Choose the **Inspection Group 1** from the **Available Elements** area, and select it using the  **Button**.

Before choosing OK it is best to apply the optimisation to the chosen elements.

- Choose **Apply**, followed by **OK**.





Choosing **OK** progresses to the **Best Fit Analyser** (see left).

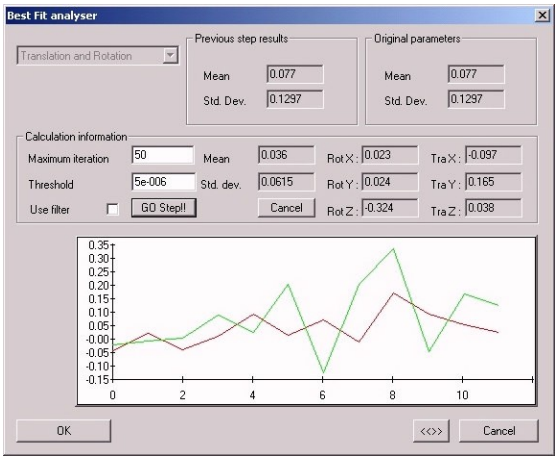
*Note: If the **Delay Measure** is on*  *, then it will be necessary to play the **Best Fit** once created. Simply right-click on the **Best Fit Icon***  *in the **Sequence Tree** and choose play to start the **Best Fit Analyser**.*

A graphical display of the current points' deviation is given (**green** line) along with the **Information** used for the **Calculation** of the **Best Fit** (see above left).

- Leave the **Default Settings** and choose the **Go Step! Button**.

The **Best Fit** will now be applied to the data.

To the right it can now be seen that the **Best Fit Optimisation** has been applied to the data and the new deviation graph (**red** line) fits closer to zero.



- Choose **OK** to accept the **Optimisation**.

The **Best Fit** has brought the results closer to zero creating a more accurate alignment.

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

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

With the alignment created, and best fitted, subsequent **Surface Inspection Groups** can now be taken. Since all items created are done so with respect to the Best Fit, further best fitting is not required. Should the results of the alignment not be accurate, it may be necessary to re-align the part though.

3.3 RPS Alignment 

Introduction to RPS Alignment

The **R**eference **P**oint **S**ystem (**RPS**) **Alignment** was 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 File**.

Generating an RPS Alignment

Again, for this example, the Headlamp CAD file is going to be used. Using the methods, previously described in the course, for starting a new session:

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**.

In the Variables dialog box:

- **Browse** for any chosen **Report Template** (**Excel** or **HTML**), and **Extract Variables**.
- Select **Finish**.

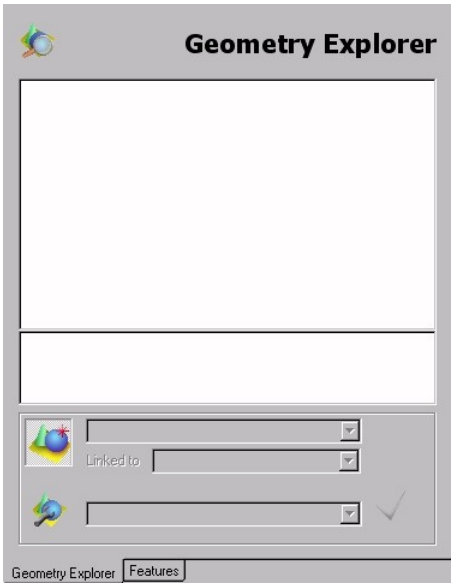
Choosing the Geometric Elements

As with the previous methods, a decision needs to be made on which **Geometric Elements** 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** will then be used to determine the **RPS Alignment**.

To create these positions, the **Geometry Explorer** will be used.

The Geometry Explorer

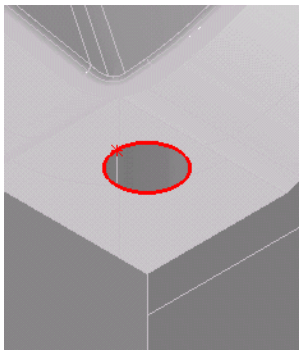


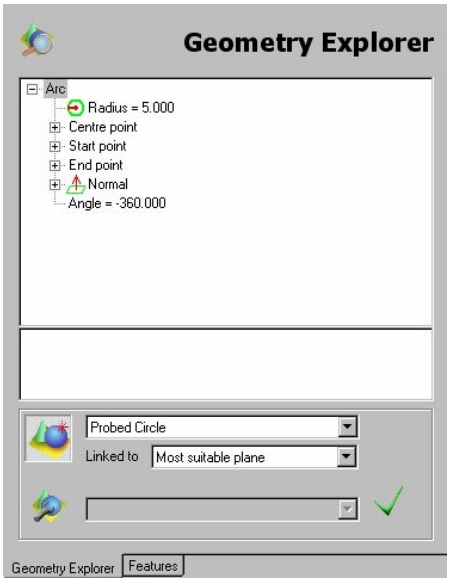
To the left, the blank **Explorer** can be seen. As each **Geometric Item** is selected its details appear in that window.
The **Geometry Explorer** can be used in two distinct ways.
The first is as a measuring device, providing **CAD Nominals** from the file, for **Geometric Alignments**. The second is as a selection tool, selecting **Geometric Items** to place in the **Sequence Tree** for probing at a later stage.

Note: The Geometric Explorer and Features Tabs can be hidden by toggling  *from the top of the Sequence Tree.*

To measure/select an item, click on the  **Button**.

Then simply hold the cursor over the required item until it's wireframe highlights (a colour change will be seen to yellow) and left click the item.
The chosen item now becomes highlighted in red (see right), with its details displayed in the **Geometry Explorer**.





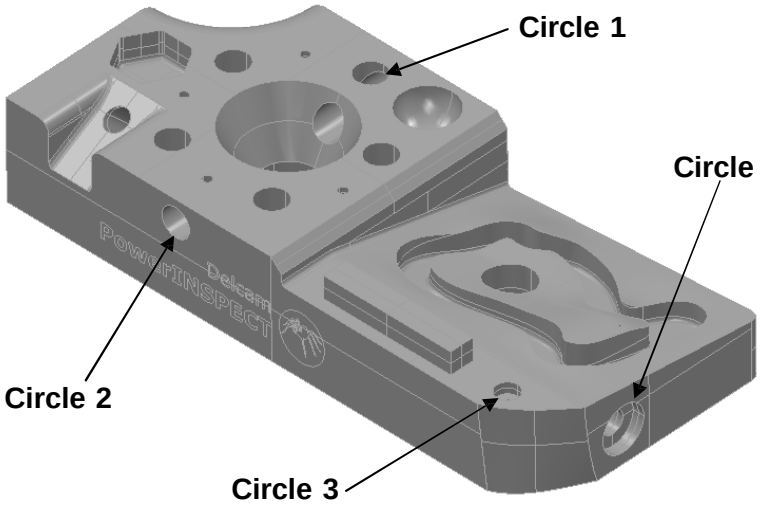
The details are displayed in the top portion of the explorer, and by moving down the levels (using the  icon), the user can view and copy specific values, like the **Arc Centre Point** for example.

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

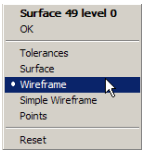
*Note: The **Geometry Explorer** selection choice can be altered, by right clicking in the graphics window. This brings up a local menu where the selection criteria can be changed, e.g. from **Wireframe** to **Points**.*

In this chapter, the **Geometry Explorer** is going to be used to select the geometric items to be probed.

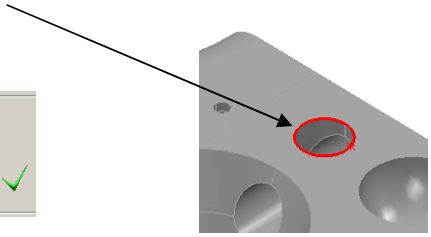
As highlighted before, the part is to be aligned using a series of probed circles, each with their own locally probed plane. The circular holes are located at each corner of the part.



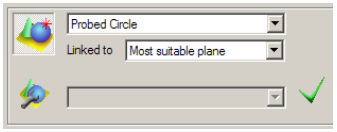
- Orientate the view so that all the circles can be seen (e.g. an **ISO 2 View**).
- Select the **Delay Measure Button.**



- Select **Wireframe Checker**
- Ensure **Wireframe** is selected from the local menu.
- Select the top wireframe circle of the hole at **circle 1** (turns red in colour).

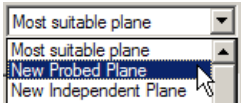


- Ensure **Linked to Most suitable plane** is selected.

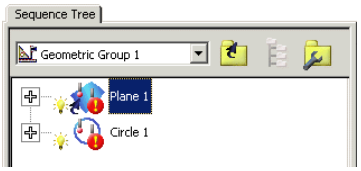


- Select to accept the feature.

*Note: If an individual plane (or correct) is not assigned to a selected feature, the option **New Probed Plane** can be selected before accepting.*

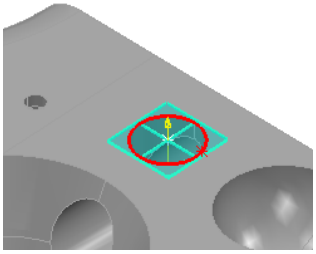


By pressing the  **Button**, **PowerINSPECT** creates a **Plane** and **Circle** to be probed. This can be seen in the second portion of the **Geometry Explorer** (see above) and in the **Sequence Tree** (see right).

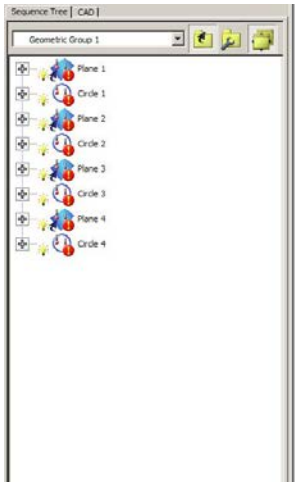


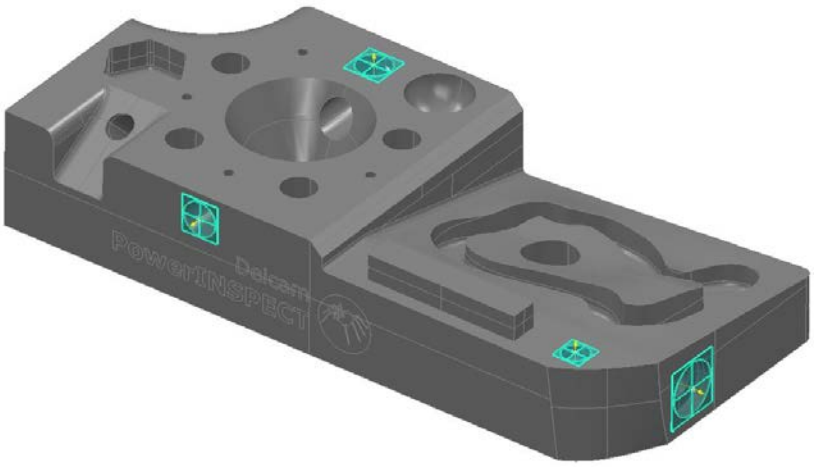
*Note the **Exclamation Marks** next to the **Geometric Items**, which again signify that they have not yet been probed or 'played'.*

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

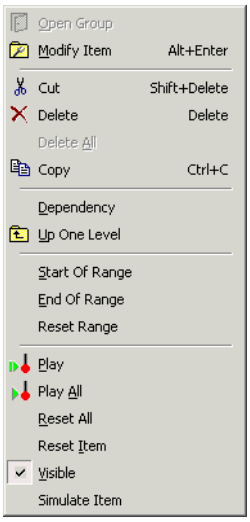


- Repeat this process for the other three **Hole Circle Wireframes**.





Once created, the **Sequence Tree** displays the eight items (four **Planes** and four **Circles**), which can also be seen in the **CAD View** (see above). With the part securely fixed to the CMM bed, these items are now ready to be probed.



- **Right Click** over ***Any*** **Geometric Item** in the **Sequence Tree** to bring up the **Local Menu** (right).

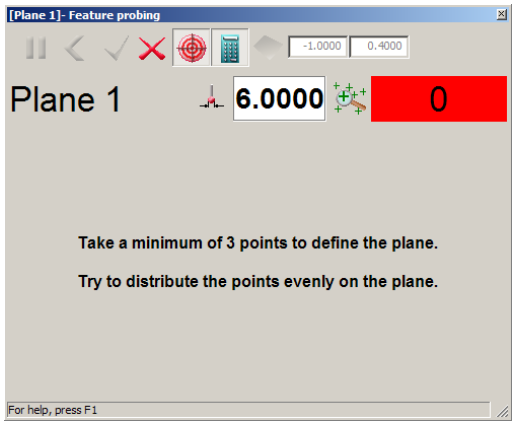
- Choose the **Play All** option.

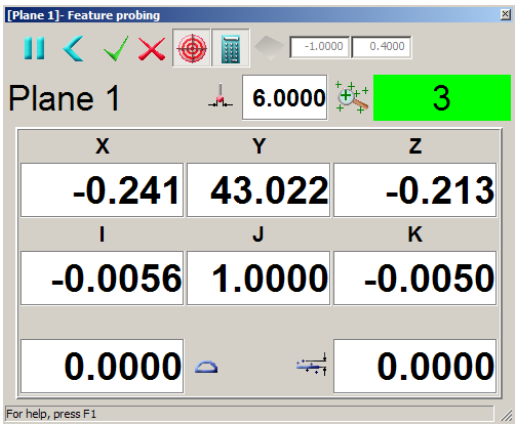
Alternatively

- Choose **Play All** from the **Measures Menu**, or the **Play All Button**  from the **Main Toolbar**.

As with the **Geometric PLP Alignment**, the first **Feature Probing** box appears asking the user to take a minimum number of points to define that feature.

- Probe three points around the **Local Plane** shown on the screen.



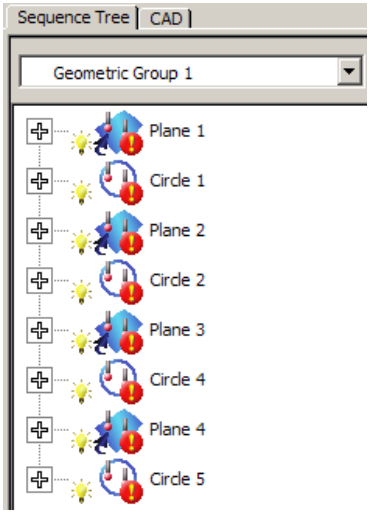


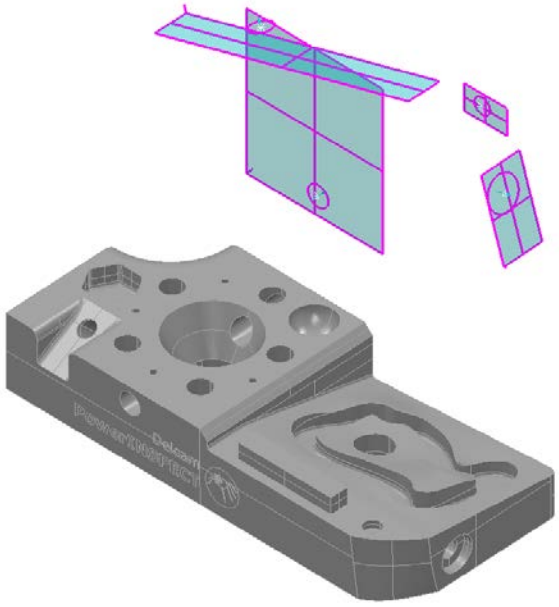
- When the points have been probed select the **Green Tick**  to **Accept**.

The process will then proceed to the next **Geometric Item**.

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

As with the **Geometric PLP Alignment**, once all the probeable items have been probed the **Sequence Tree** is updated (right), with all the items having lost their **Exclamation Marks**.





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

An **RPS Alignment** will now be used to align the **CAD Data** to the **Physical Part**.

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

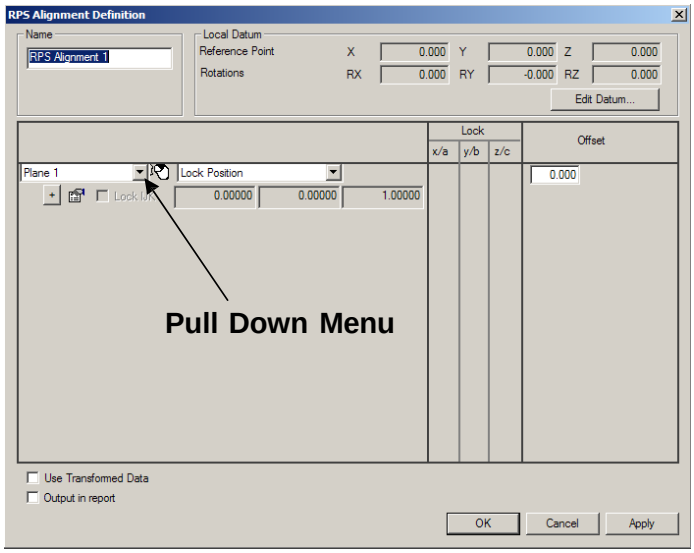
Generating an RPS Alignment

- Move up the levels  until **Geometric Group 1** is **Closed**.
- From the **Alignments Sub-Menu** icon  , in the **Element Toolbar**:
- Choose the **RPS Alignment** button  .

The **RPS Alignment** dialogue box appears and prompts the user to define the items for the alignment.

The items used for alignment are selected via the **Pull Down Menu**.

Once an item is chosen other items can be added using the + icon.

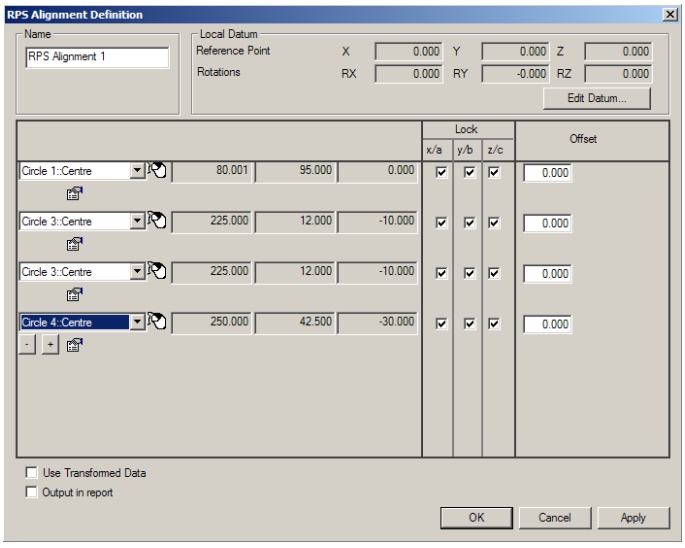


- Leaving the **Name** and **Local Datum** as **Default**, choose the **Circle 1:: Centre** to define the first position from the **Pull Down Menu**.

- Select the + **Icon**, to create a new **Pull Down Menu**.

- From this new **Pull Down Menu** choose the **Circle 2::Centre** to define the second position, and again select the + **Icon**.

- Repeat the process for the **Circle 3** and **4 Centres**.

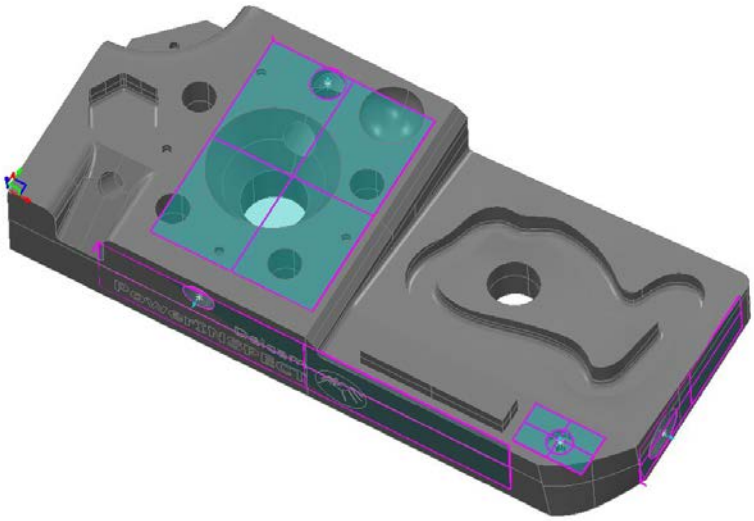


The form should look similar to that on the right.

*Note: The **Tick Boxes** allow the user to lock and unlock axes relative to particular items – thus the user can effectively create a 3-2-1 alignment, by ticking and unticking the boxes.*

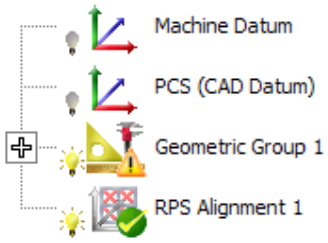
- Choose **Apply** and **OK**.

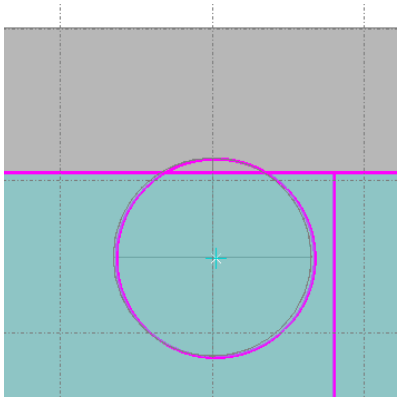
PowerINSPECT aligns the probed positions to the CAD points given, satisfying **X**, **Y** and **Z** coordinates.



To the right it can now be seen that the **CAD View** has been 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 **Sequence Tree.**





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

To the left can be seen a view looking down the **Z-axis** 

, zoomed into one corner. The alignment can clearly be seen here.

As before, 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.

3.4 Best Fit From Points Alignment 

Introduction to Best Fit from Points Alignment

This technique 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 elements 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 elements.
Before you use this alignment strategy you must have created geometric elements that provide at least three points. Points include centres of geometric elements, such as the centre of a circle or sphere. You must also know their position in the CAD data: if you don't have this information, you can extract the coordinates using the **Geometry Explorer** tab. This method will be shown below.

*NOTE: A **Best Fit** alignment can produce the same results as a **Three Spheres** alignment if three spheres are used to define the three points.*

Generating a Best Fit Alignment using Geometry Explorer

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**.

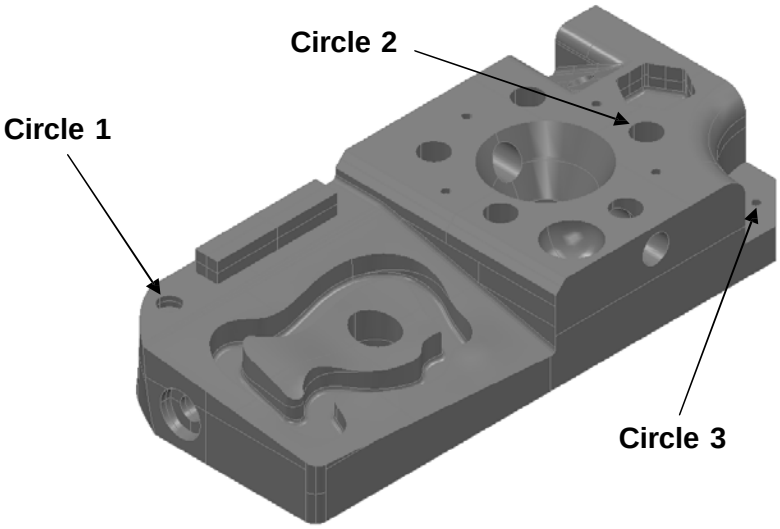
In the Variables dialog box:

- **Browse** for any chosen **Report Template** (**Excel** or **HTML**), and **Extract Variables**.
- Select **Finish**.

Choosing the Geometric Elements

As with the previous methods, a decision needs to be made on which **Geometric Elements** 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** will then be used to determine the **Best Fit Points.**

Similar to the previous RPS method, the **Geometry Explorer** will be used in conjunction with the **Wireframe Checker.** 

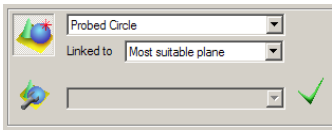


- Orientate the view so that all the circles can be seen (e.g. an **ISO 3 View**).

- Select the **Delay Measure Button.**  .

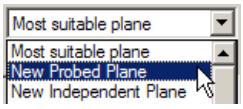
- Select **Wireframe Checker** 
- Ensure **Wireframe** is selected from the local menu.
- Select the top wireframe circle of the hole at **circle 1** (turns red in colour).

- Ensure **Linked to Most suitable plane** is selected.

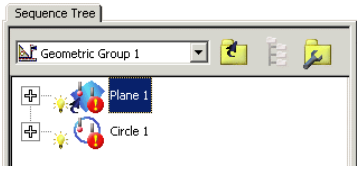


- Select  to accept the feature.

*Note: If an individual plane (or correct) is not assigned to a selected feature, the option **New Probed Plane** can be selected before accepting.*

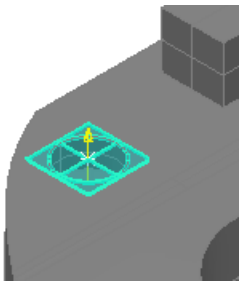


By pressing the  **Button**, **PowerINSPECT** creates a **Plane** and **Circle** to be probed. This can be seen in the second portion of the **Geometry Explorer** (see above) and in the **Sequence Tree** (see right).

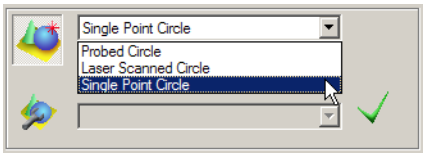


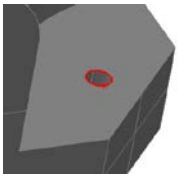
*Note the **Exclamation Marks** next to the **Geometric Items**, which again signify that they have not yet been probed or 'played'.*

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

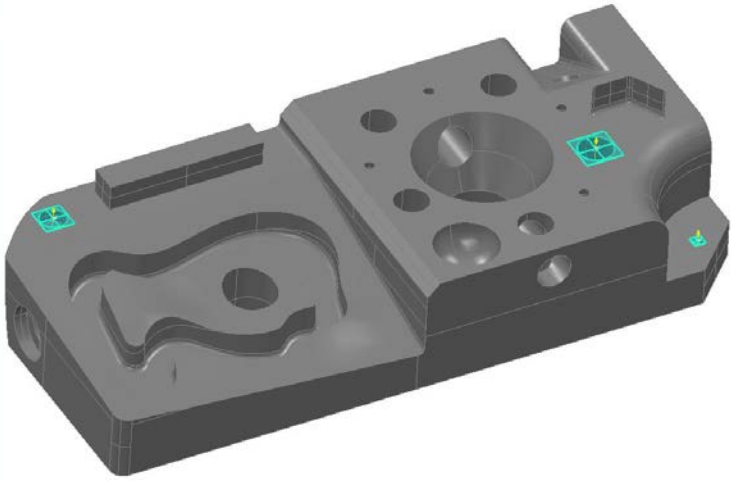


- Repeat this process for **Circle 2**.
- Select the **Circle 3** Wireframe. From the Geometry Explorer Tab change the Circle option to **Single Point Circle** before accepting the Feature.

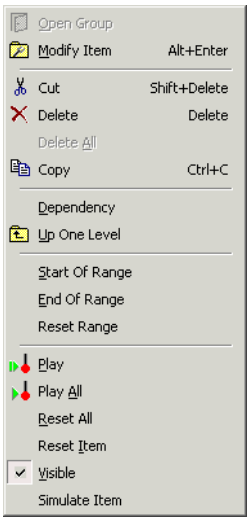




Once created, the **Sequence Tree** displays the six items (three **Planes** two **Circles** and a **Point**), which can also be seen in the **CAD View** (see above). With the part securely fixed to the CMM bed, these items are now ready to be probed.



- **Right Click** over ***Any*** **Geometric Item** in the **Sequence Tree** to bring up the **Local Menu** (right).



- Choose the **Play All** option.

Alternatively

- Choose **Play All** from the **Measures Menu**, or the **Play All Button**  from the **Main Toolbar**.

- Probe all features as directed.

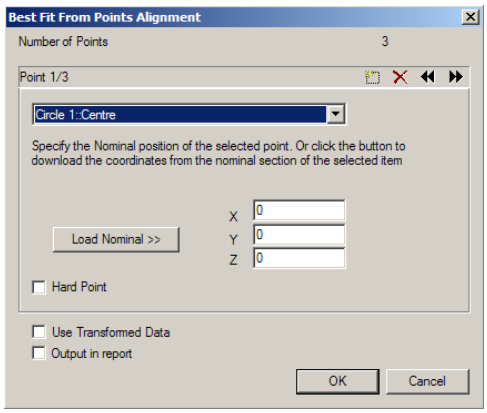
Generating a Best Fit From Points Alignment

- Move up the levels  until **Geometric Group 1** is **Closed**.

- From the **Alignments Sub-Menu** icon  , in the **Element Toolbar**:

- Choose the **Best Fit From Points Alignment** button  .

Point 1/3 is displayed.

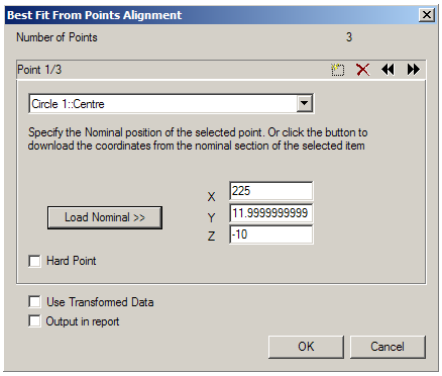


- Ensure **Circle 1: Centre** is displayed otherwise select it from the drop down list.

X, **Y** and **Z** Coordinates can be typed in for **Circle 1: Centre**. Alternatively, as the nominal values have been extracted using Geometry Explorer, these coordinates can simply be loaded in.

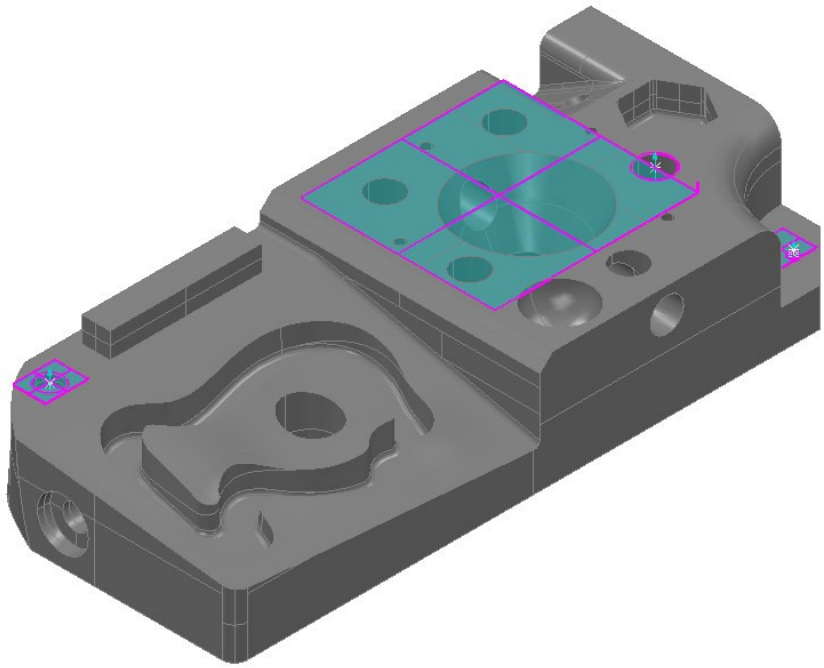
- Select Load Nominal >> to load in the nominal values for the circle centre

Note: You can make the first point a Hard Point by clicking on the 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.



- Select ⏩ to move to display **Point 2/3**.
- Ensure **Circle 2:Centre** is shown before selecting Load Nominal >>
- Select ⏩ to move to display **Point 3/3**.
- Ensure **Point 1:Centre** is shown before selecting Load Nominal >>
- Select OK to create the Best Fit Alignment.

A minimum of three points is required for the alignment. If required, further points can be added by selecting ***New*** .



4. Inspections

Introduction to Inspection

Following a suitable Alignment as described in the previous chapter, the part can now be inspected against the CAD data. This will highlight the accuracy and error (deviation outside defined tolerance) of the part, assuming the CAD data is correct. This chapter highlights the main inspection techniques available in PowerINSPECT.

The Tolerance values, through which the **Surface Inspection Groups** operate, can be changed globally (though the use of the **Start-up Wizard**, or **Measures>Parameters** menu option), or individually in each **Surface Inspection Group**.

4.1 Surface Inspection Group – Points on the fly.

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**.

In the Variables dialog box:

- **Browse** for a **HTML Report Template** and **Extract Variables**.
- Select **Finish**.
- Create an **Alignment** (if necessary), using one of the previously learnt methods.
- Turn off the **Light Bulb** for the **Geometric Group 1**  .

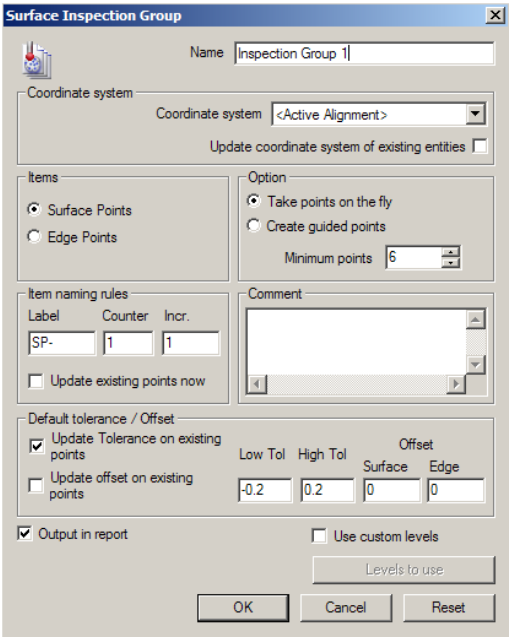
This removes the Geometric Group 1 from the screen, making it a little less cluttered when taking the **Surface Inspection**. Alternatively the **Toggle Previous Measure Display Button**  could be used, from the **Full Screen Options Toolbar**, in full screen mode later.

- Create a **Surface Inspection Group**, by clicking on the **Surface Inspection Group** Button  on the **Element Toolbar**.

The **Surface Inspection Group** form opens up.

Within this form the user can set the **Name** of the group and can decide on whether to take **Surface Points**, or **Edge Points**, **Points on the Fly** or **Guided Points (see later)**. They can choose the **Minimum Number of Points** they wish to take, as well as setting **Counter Values, Labels** and adding **Comments**.

This form also allows the user to change the **Local Offsets** and **Tolerances** for this group, and whether or not it is output to the **Report**.



In this example, a **Surface inspection** will be specified with **points taken on the fly.**

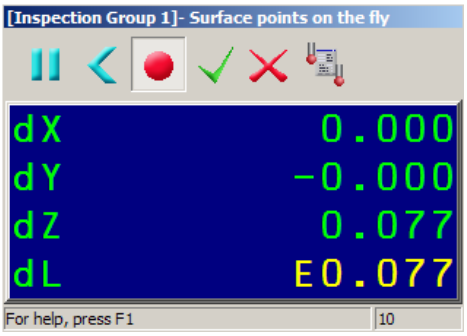
- Keep all the **Default Settings**, making sure the **Output in Report** option is ticked, and choose **OK**.

If the **Delay Measure Button**  is off, **PowerINSPECT** will automatically move to the **Probing Screen**, otherwise:

- Right click over the **Surface Inspection Group Icon**  that has now appeared in the **Sequence Tree** and select **Play**.

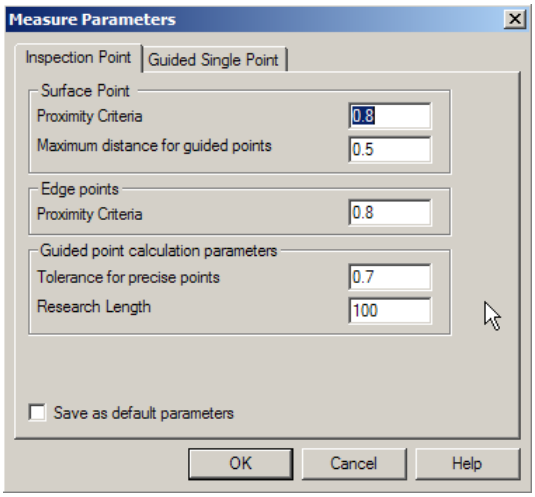
The user is now faced with new **Probing Screen**, and a minimum number of points (set in the **Surface Inspection Group** dialogue box – in this case **6**) need to be taken to satisfy the inspection.

As the points are probed the **Surface Points on the Fly** data box displays the deviation of the measured points. These are displayed in all three axes, in terms of **dX**, **dY** and **dZ** as well as a resultant deviation **dL**.

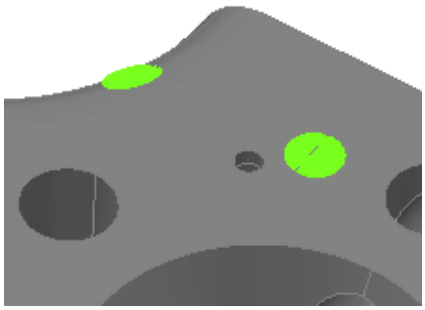


This box has similar options to that of the Feature **Probing Dialogue Box** seen in the previous chapter with addition of a **Suspend Recording Button**  , which allows the user to see the points without actually recording them as a result.

It also has a **Parameters Button**  , which displays a dialogue box with a summary of the **Measure Parameters** used for **Inspection Points** and **Guided Points** (such as proximity criteria – which may need to be increased if points are not registering).



- Probe any number of points on the **Part Surface**, making sure to satisfy at least the **Minimum Number of Points** set in the form.



- When satisfied that enough points have been taken, choose **OK.** 

It can be seen that the **Sequence Tree** has now been updated, with the **Surface Inspection Group** icon  having lost the **Exclamation Mark**. The Surface Inspection results can now also be seen in the **Report**.

- Select the Report **Tab** to view the **Surface Inspection Group Results**.

A display of all the elements chosen to be output to the report can now be seen. This report can also be exported, and viewed through a **Web Browser**.

- Choose the **Export Report Button**  , and call it **Report.mht**. Proceed to view this file with any installed **Web Browser**.

As well as report formats, the results can be viewed from within the **CAD View** and the **Sequence Tree**.

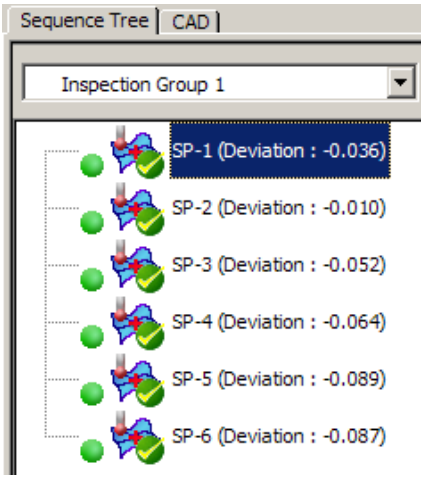
- Open the **Surface Inspection Group** to view the points that have been taken. Do this by either selecting **Inspection Group 1** in the **Sequence Tree** and choosing the **Open Group Button**  , or by choosing Open Group from the local Right-click menu.

Note: Just using the  ***Button****, will open the group to display the points in the* ***Sequence Tree****, but the statistical data will* ***NOT*** *be displayed in the* ***Print Preview****. To view this, the individual groups need to be opened.*

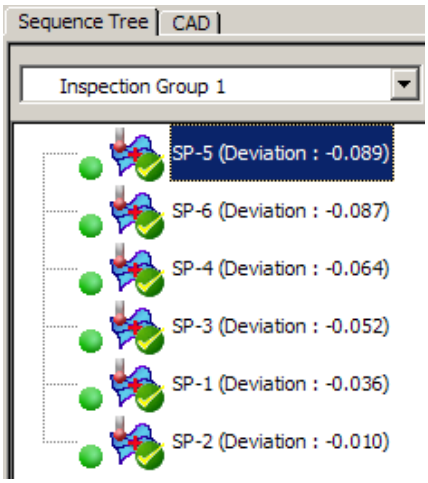
Once open the points can be individually highlighted, deleted or modified (see Chapter *Modifying Elements* later). Double clicking on each point, will open its information list, detailing positions, deviations etc.

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 choose the **Sort by Deviation** option from the local menu.



Sort By Number



Sort By Deviation

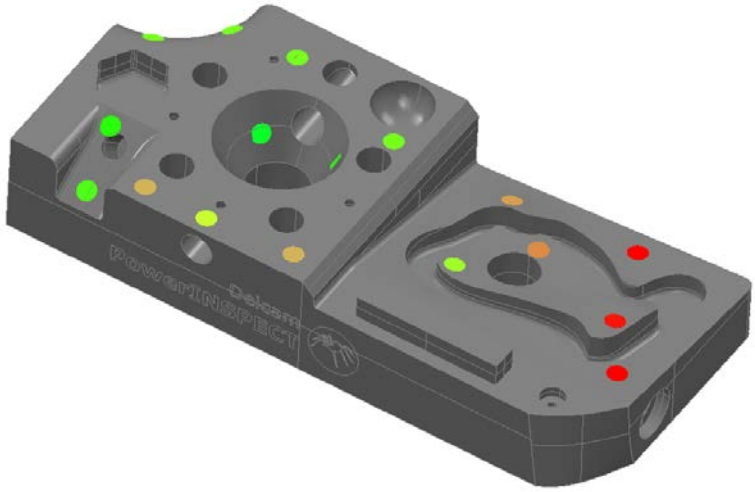
This sorts the points by the amount by which they deviate from zero, starting at the lowest and working up to the highest deviations. See above for examples of both.

Data display in CAD view

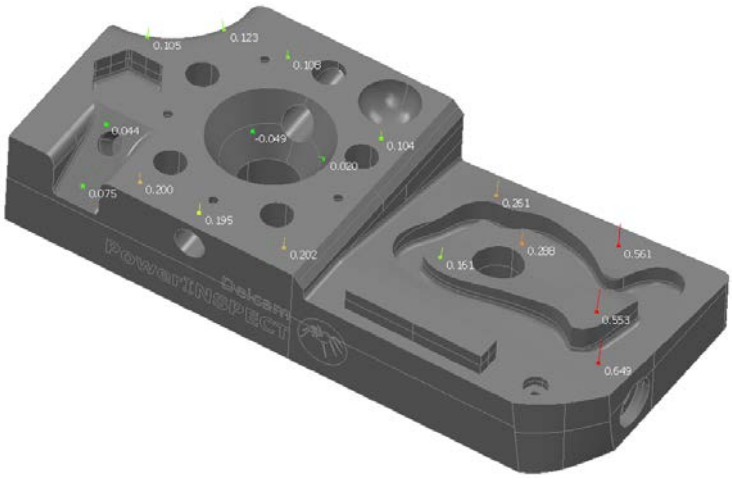
Geometric features and/or **probed points** can be displayed in all manner of formats

'Confetti' displays the points as coloured spots. **Green** represents points within tolerance, **Red** for above tolerance and **Blue** for below tolerance.

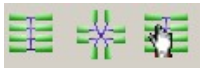
Confetti



'In Place' values display the points as 'pins' with the length of each pin representing the amount of deviation. The pin colour also corresponds with the colours as described above (confetti).



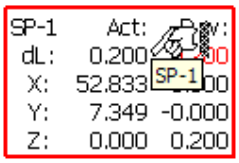
In Place Values

'Labels'  can be activated to display boxed details of the Geometric features and/or results of probed points. Labels can be displayed alongside the '**confetti'** or the '**In Place'** formats.

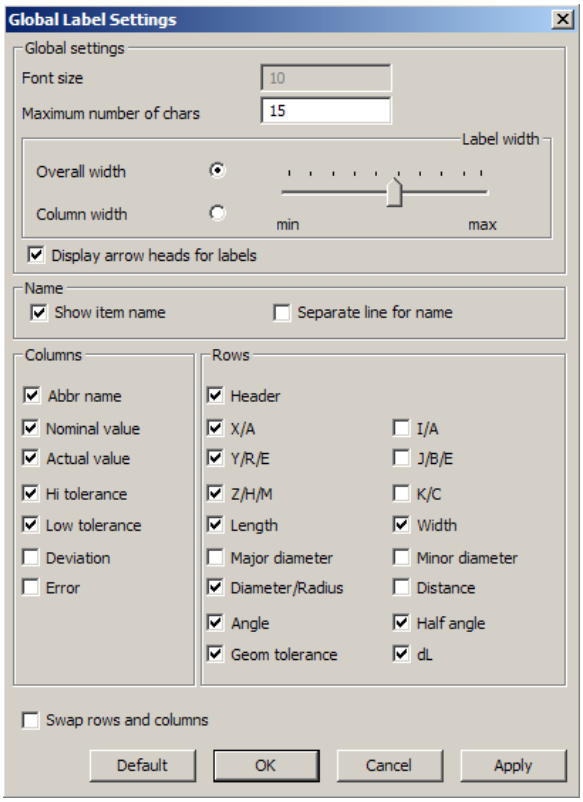
Selecting  activates and displays all the labels in a horizontal format.

Selecting  activates and displays the labels in standard format (horizontal and vertical format)

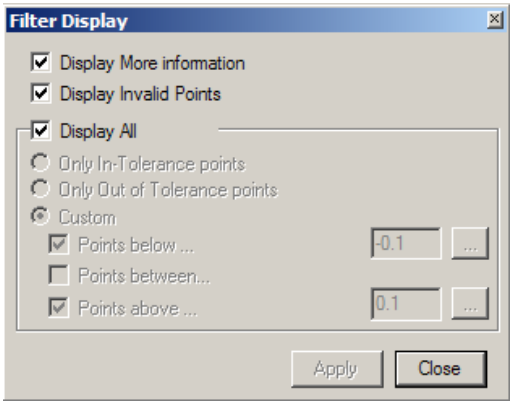
Label positioning can be customized by selecting  . Individual labels can be moved by keeping the left mouse button down on a label whilst 'dragging'.

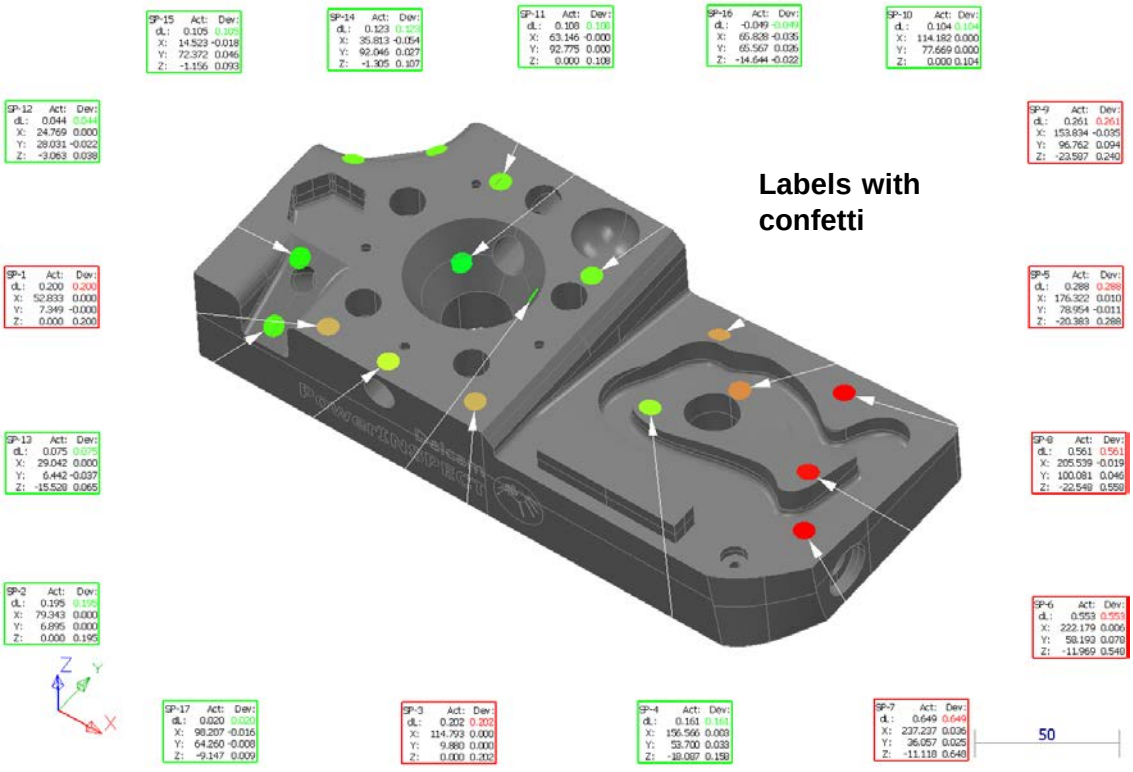


Selecting the **Global Label Settings** icon  allows the user to select the information that is displayed in the labels, including the label width.



Further filter display options are available by selecting  allowing the user to filter out the points displayed in the screen. By default, this is set to **Display all**. For example if the user only wanted to display the Out of tolerance points, they could choose the **Only Out of Tolerance points** option. The **Custom** option allows the user to set their own filter criteria.





Print Previews

With the Surface Inspection performed, the user can print the data.

- Select an **ISO 2**  view and choose the **Print Preview Button**  .

When the **Print Preview Button**  is selected, or the **Print Preview** option is chosen from the **File Menu**, then a preview of the **CAD File**, along with the **Probed Points**, is shown on the screen.

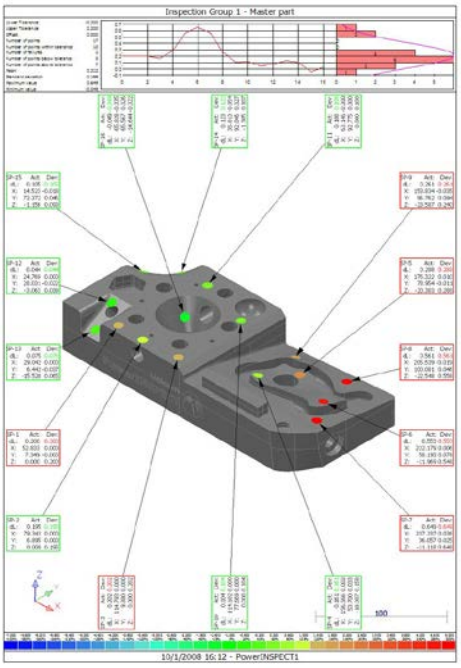
If the user were to print the page, then this would be the output.

If the **Print Preview Button** is selected

whilst a **Geometric group** is **opened**  , then a graph of deviation and coloured bar is included in the preview.

Note: Ensure 

is also selected from Tools-Options-Display Options.



With Labels

With In Place Values

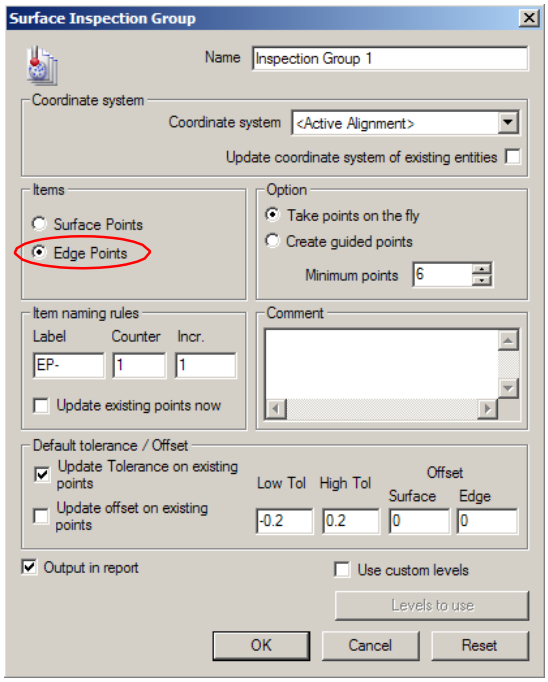
With Confetti and Colour Bar

4.2 Surface Inspection Group – Edge Points.

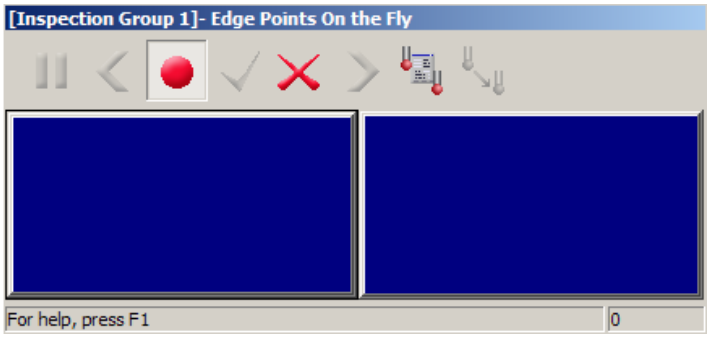
PowerINSPECT allows the inspection of Edge points.

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Create an **Alignment** (if necessary), using one of the previously learnt methods.
- Select **Surface Inspection Group**  .
- Select the option **Edge Points**.
- Keep all the **Default Settings**, making sure the **Output in Report** option is ticked, and choose **OK**.



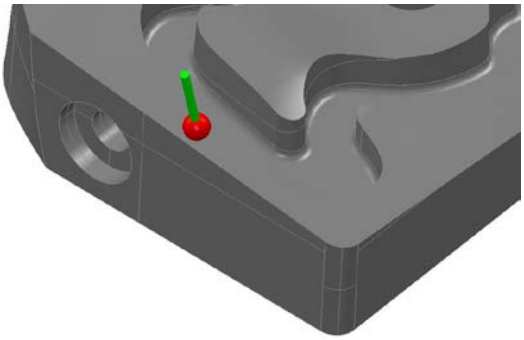
- Right click over the **Surface Inspection Group Icon**  that has now appeared in the **Sequence Tree** and select **Play**.



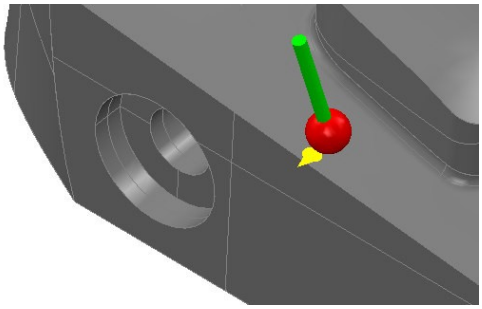
The user is now faced with new **Probing Screen**, and a minimum number of points (set in the **Surface Inspection Group** dialogue box – in this case **6**) need to be taken to satisfy the inspection.

PowerINSPECT requires two probes points to define the edge that is required for inspection.

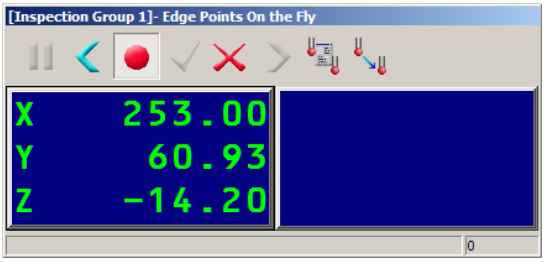
- Probe a point on the surface whose edge is required to inspect



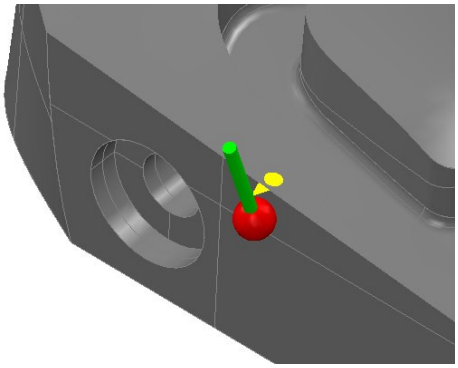
PowerINSPECT displays a yellow mark to represent the point and a yellow triangle marker to indicate where the second point must be probed.



The co-ordinates of the edge point that is required are now displayed in the left hand Probing screen.



- Probe the edge indicated by the marker to record the Edge point.

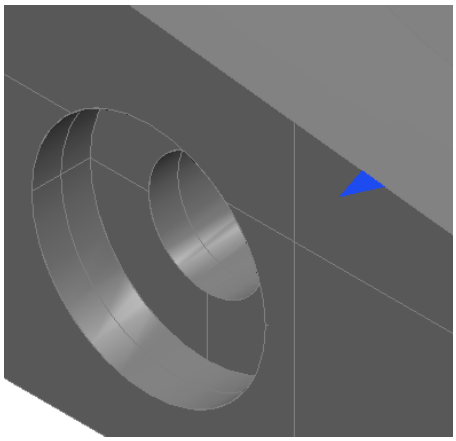


Note: The point must be probed precisely. A progress bar at the bottom of the window indicates the proximity. The proximity criteria can also be changed by selecting  *from the Probe window.*

Once the point is probed, the deviation is indicated in the right hand Probe Window.

- Repeat the process above to take further edge points.
- Once the probing is complete, select OK 

PowerINSPECT displays the edge points on the model as triangles. Display can be changed to suit (e.g. switching on labels). The colour of the triangles represents the deviation in the same way as ‘confetti’.



4.3 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.

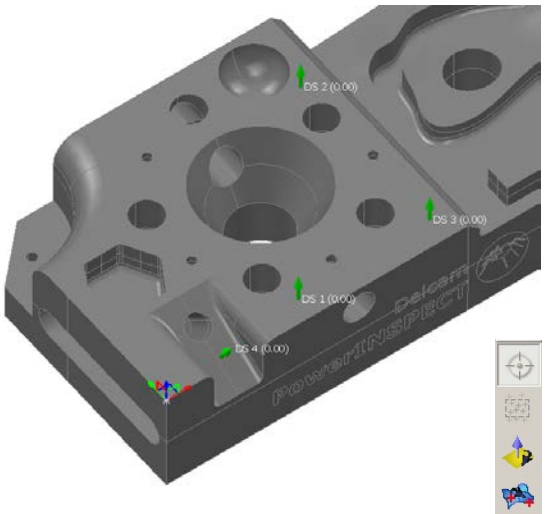
Using the Dynamic Points Editor.

*The use of the **Dynamic Points Editor Toolbar** was also described in Chapter 3.2, as part of the Free Form Alignment.*

- Create a **New Document using the Wizard** and select Measurement with a single CAD Part.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Create an **Alignment** (if necessary), using one of the previously learnt methods.
- Select the **Dynamic Points Editor** from the **Mouse Context Toolbar.**

The cursor has now changed to a **Target Sight** in the CAD view. This allows the user to select the points on the CAD surface that are going to be inspected as guided points. Points are created by simply '**double left mouse selecting'** on the surface of choice.

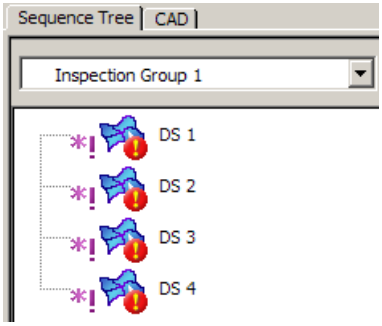
- Create a number of points on the Surface of the model. For example as shown.



Points can be manipulated using the other **Dynamic Points Editor Toolbar** functions. See chapter 3.2.

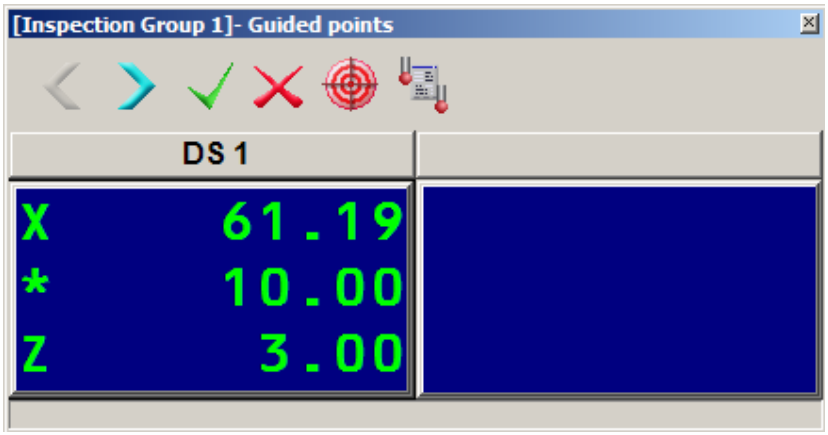
- From the **Dynamic Points Editor Toolbar**, select **Create a Guided Surface Inspection group.**

The points are placed into a Surface Inspection group in the sequence tree.



- **Select Play** button from the **Main Toolbar.**

Once into the Full Screen Measure Mode, the Probing Process can occur. The Dynamic Point Targets appear for the user to aim.



A Green coloured bar at the bottom of the window indicates the proximity.

Points will be accepted that are within the defined Proximity criteria.

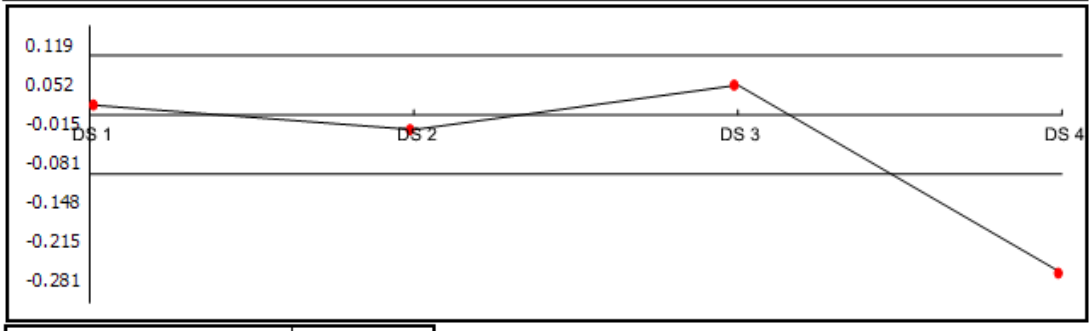
They appear in sequence order and as each one is probed the next one appears on the screen.

- **Probe** all points in sequence order.
- Check the Measurement results in the **Report.**

Inspection Group 1

Datum - *Geometric PLP Alignment 1*

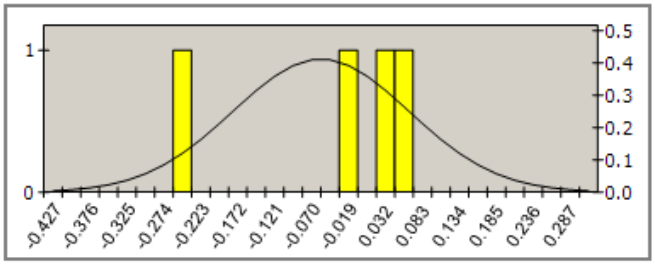
Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
DS 1	0.000	-0.100	0.100	61.073	9.801	0.000	0.000	0.000	0.015	**0.015**
DS 2	0.000	-0.100	0.100	115.964	81.009	0.000	0.000	0.000	-0.028	**-0.028**
DS 3	0.000	-0.100	0.100	118.603	11.753	0.000	0.000	0.000	0.048	**0.048**
DS 4	0.000	-0.100	0.100	31.768	11.928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

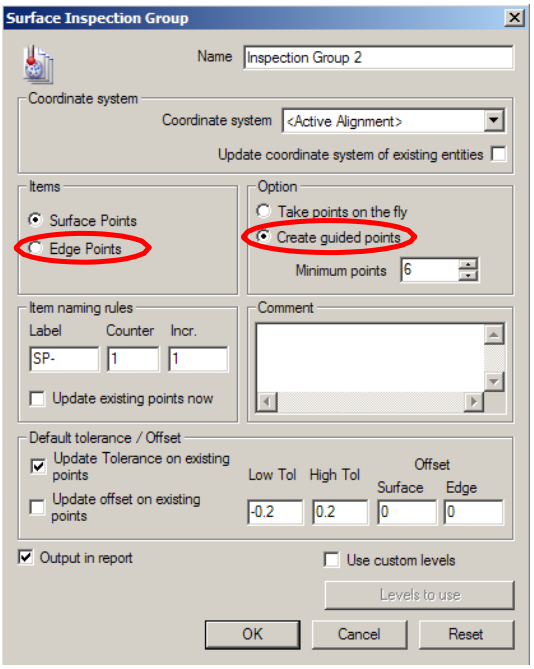
Max. Value	0.048
Min. Value	-0.263



Do not close the session

Manually Entering Points

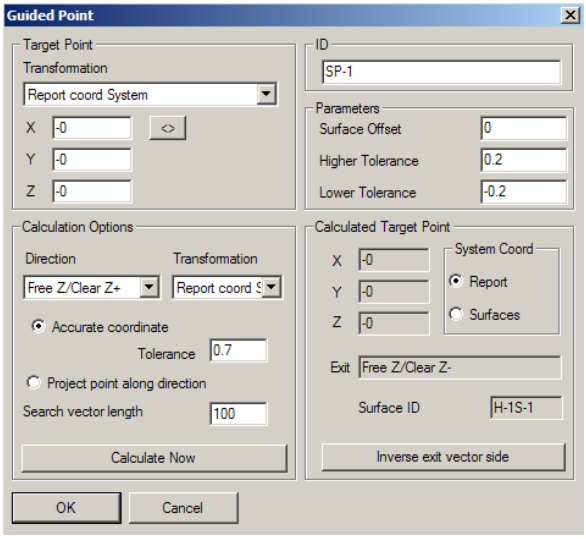
- Ensure the **Delay Measure** Button from the **Main Toolbar** is selected.
- Create a **Surface Inspection Group**, by clicking on the **Surface Inspection Group Button** on the Element Toolbar.



- Select the option **Surface Points** and **Create guided points** (as shown).
- Select **OK.**

- Open the new Surface Inspection Group.
- The Element Toolbar now displays new options.

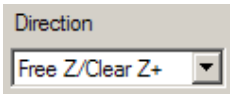
- Select the first option **Create Guided Surface Point.**



This dialog allows the user to **modify existing points** (see page 4.13) or enter **new coordinates** to create points.

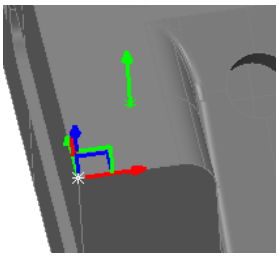
- Enter the following co-ordinates in the Target Point area X **10,** Y **15,** Z **0.**
- Ensure the option Accurate coordinate is selected.

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



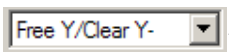
- Select **Calculate Now** then press **OK**

The point is created on the surface.



- Repeat the process above for the following coordinates.

X **55** Y **80** Z **0**

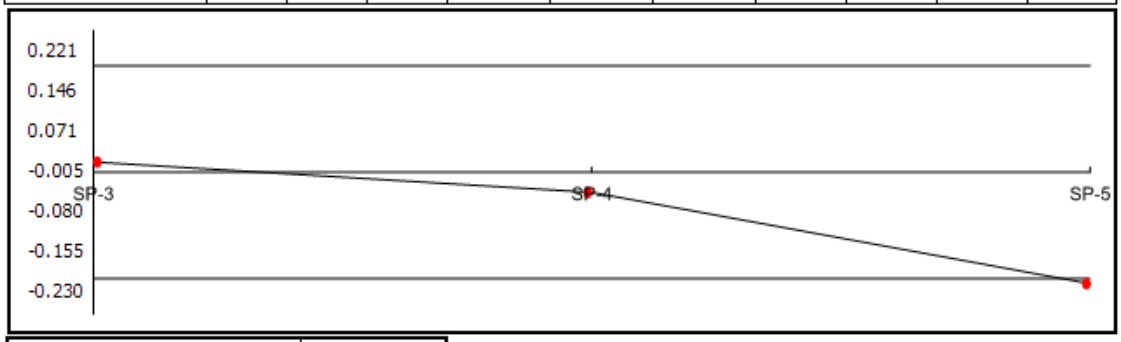
X **60** Y **0** Z **-10** *(select direction*  *)*

- Select Play  button from the Main Toolbar.
- **Probe** all points in sequence order.
- Check the Measurement results in the **Report.**

Inspection Group 2

Datum - *Geometric PLP Alignment 1*

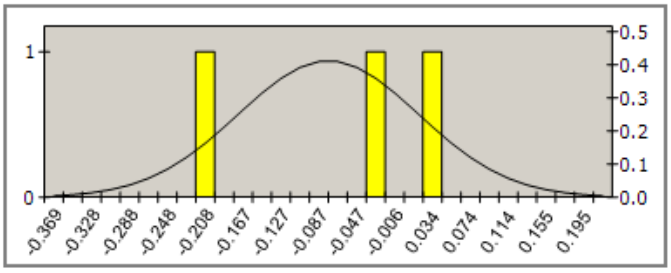
Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-3	0.000	-0.200	0.200	55.122	80.139	0.000	-0.000	0.000	0.016	0.016
SP-4	0.000	-0.200	0.200	10.228	15.022	0.000	0.000	0.000	-0.037	-0.037
SP-5	0.000	-0.200	0.200	60.311	0.000	-9.187	0.000	0.209	0.000	-0.209



Number of points	3
Out of tolerance	1
Performance	67%

Mean	-0.077
Std. Deviation	0.096

Max. Value	0.016
Min. Value	-0.209



4.4 Introduction to Section Inspections

Some users may wish to perform an inspection of a cross-section across a part. **PowerINSPECT** allows the user 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

For the purposes of this exercise, the section is going to be taken using the headlamp model as the CAD file.

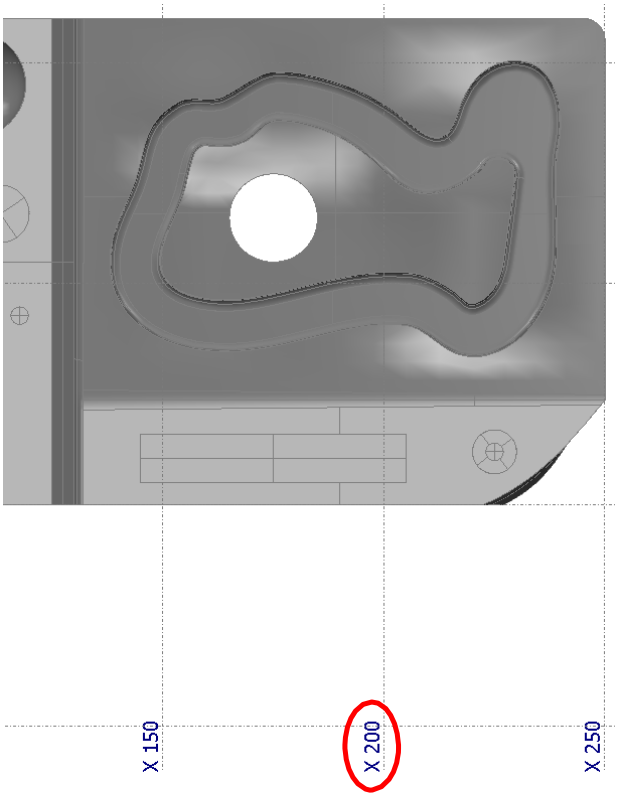
- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.

- Create an **Alignment** (if necessary), using one of the previously learnt methods.

Note: If a session is already open, or has been saved and can be opened (with the part in the same position), then 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, the user would manipulate the view, in order to make a decision on where to take the section. However, again 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**.

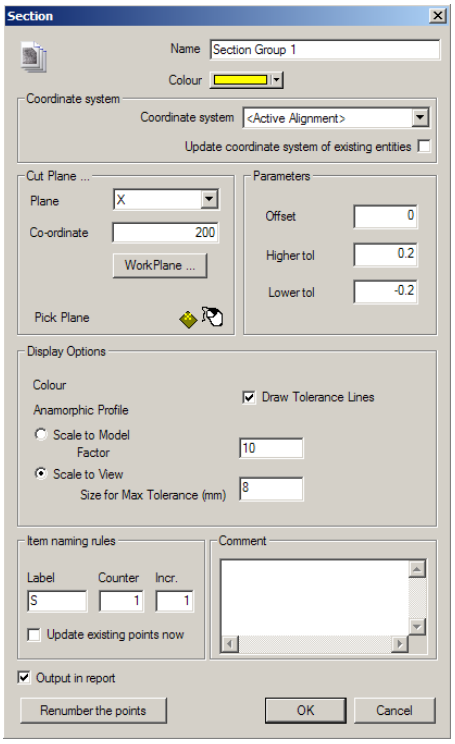


Generating a Section Inspection

In order to inspect a **Sectional** area the user needs to define the section to be probed.

- Select the **Delay Measure Button**  .
- From the **Element Tool Bar** choose the **Section Group**  **Button**.

The **Section** dialogue box appears. Within this the user can specify the **Name**, **Line Colour**, **Cut Plane**, **Parameters**, change the **Display Options** and the **Item Naming Rules** and add a **Comment**.

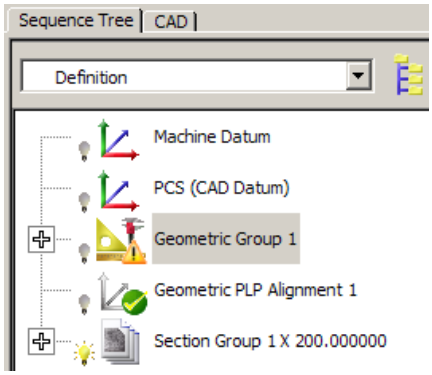


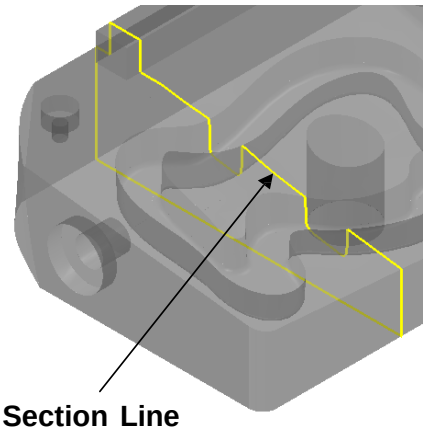
Choosing the 

Button allows the user to input a **Rotation** or **Translation** of the **Section Line**, as with the **CAD File Parameter Edits**.

- Change the **Cut Plane** to the **X-Plane**, and enter a **Coordinate Distance** of **200** (as illustrated).
- Leave all the other options as **Default**, and choose **OK**.

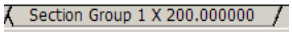
By choosing **OK**, the **CAD View** is updated with a visible **Section** and a **Section Group** has appeared in the **Sequence Tree** (right).

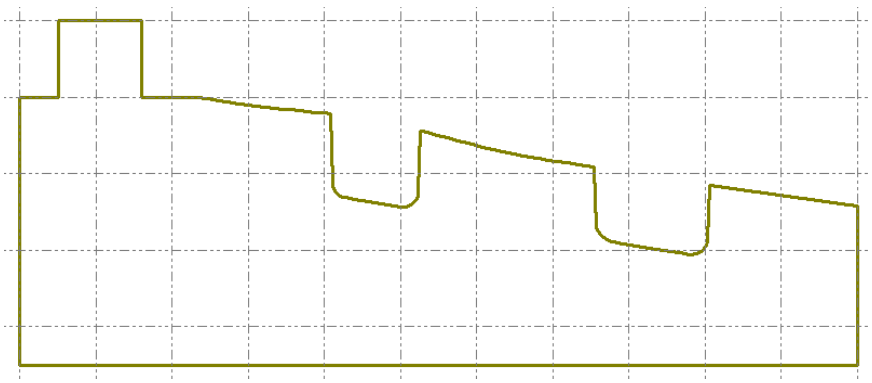




*Note the initial **Geometric PLP Alignment**.*

When a **Section Group** is created a new tab appears at the base of the **Graphics Window**. Selecting this tab opens the **Section View**.



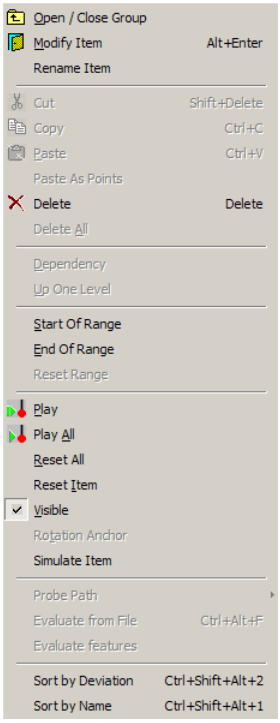


The **Section View** (above) can be re-orientated using the arrow keys on the keyboard (for **mirror** moves) and the **Page Up** (anti-clockwise) and **Page Down** (clockwise) keys for **5-degree rotations**.

Using the **Shift** key in conjunction with the **Page Up** and **Page Down** will perform **90-degree rotations**.

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

- **Orientate** the **View** so that the section may be easily taken (e.g. from above down the **Z-Axis**).
- **Right Click** over **Section Group** in the **Sequence Tree** to bring up the **Local Menu** (right).
- Choose the **Play** option.



Alternatively

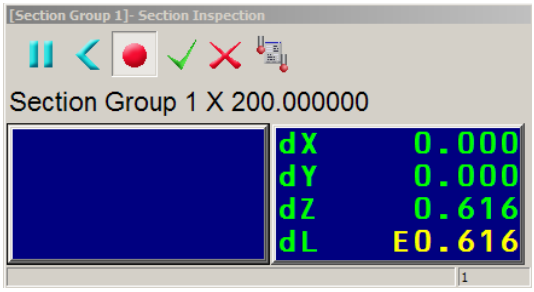
- Select the **Section Group** and choose **Play** from the **Measures Menu**.

PowerINSPECT will then move to the **Probing Screen**.

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

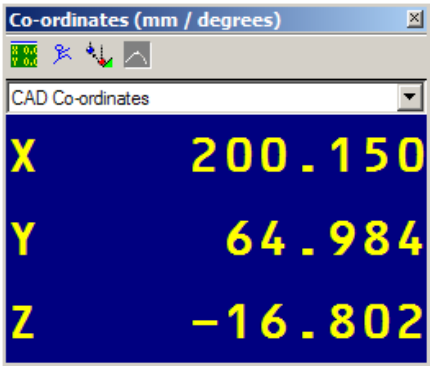
When in the **Probing Screen**, the **Section Inspection Box** appears, containing all the information about the inspection points.

Across the top are the inspection icons, which perform the same functions as highlighted in the **Free Form Alignment** section.



- Choose the **Tracking Box Icon**  .

The **Tracking Box** appears, allowing the user to see a numerical representation of the probe position. Because the **Part** has already been aligned, the tracking can be set to the **CAD Coordinates**, to monitor the **X=200** value.

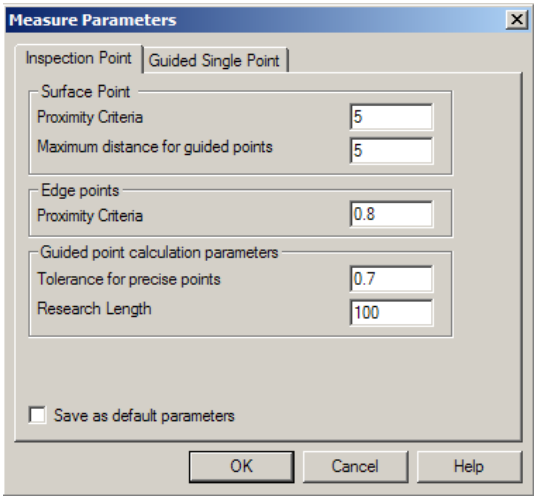


- From the **Pull Down Menu**, choose the **CAD Co-ordinates** option.

From the **Tracking Box** the probe position can now be seen 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 the user needs to specify a wider range (around **X=200**) in which to obtain probed positions.

- Choose the **Measure Parameters Icon**  .



- Select the **Inspection Point Options** tab at the top of the dialogue box.

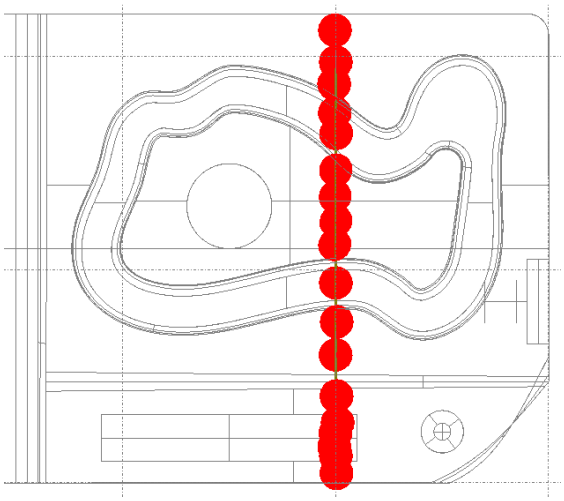
This dialogue box allows the user to increase or decrease the search parameters for inspection points for both **Surfaces** and **Edges**.

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

- Alter 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**.

Note: This is just an example of a parameter change, and it is ultimately up to the user to decide on an acceptable search parameter for personal projects.

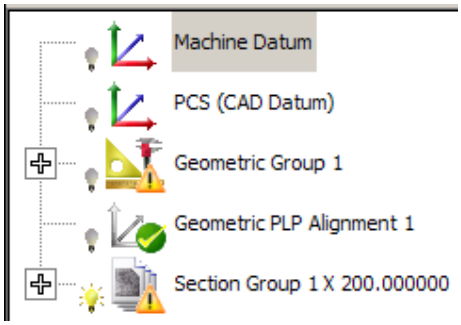


- Probe along the **X=200** line as accurately as possible.

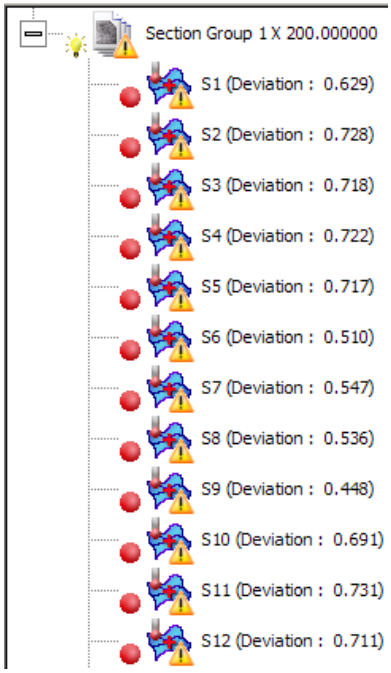
- When all the required probed points have been taken, choose the **OK**  **Button**.

The section group has now been created.

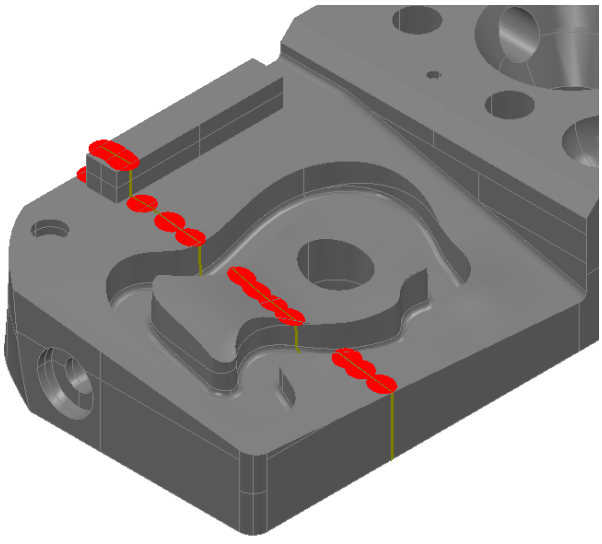
To the right it can be seen that the **Sequence Tree** has been updated, with the **Cut Plane** and **Value** now given for the **Measured Section Group**.



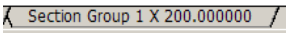
If the **Section Group** is opened (see below left) the points are displayed in the tree each with their corresponding deviation.



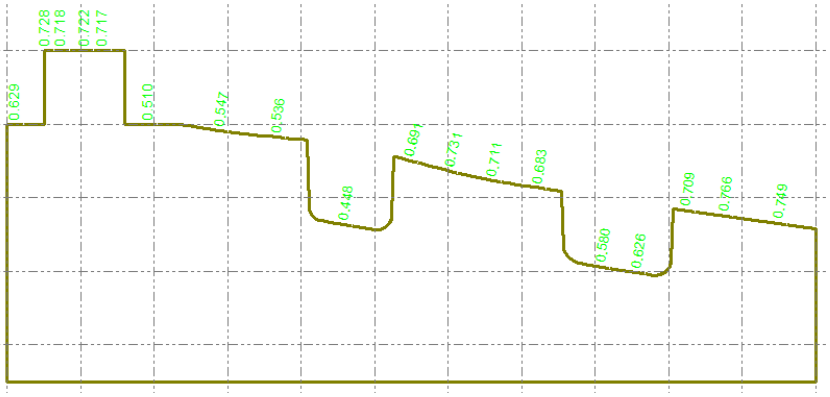
When opened, the **Section Line** can also be seen in the **CAD View** (below), with the probed confetti points along it.



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



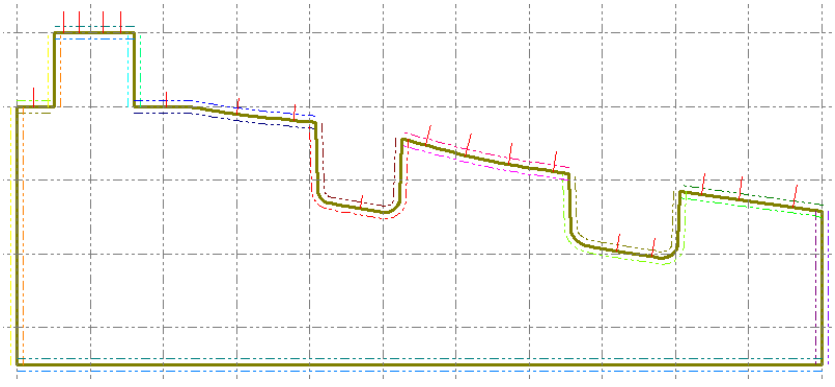
The **Section View** overleaf is shown with the **Deviation** as **Vectors**. This can be change in the **Section Group Tool Bar**.



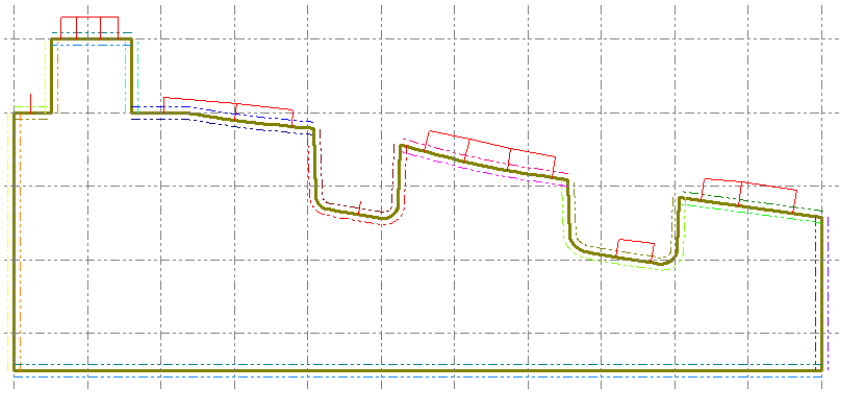
Section View with Vectors Displayed

The **Section Group Tool Bar** (right) allows the user to alter the display of the results. The results can be displayed as **Vectors**  (above), **Pins**  (below) or **Linked Pins**  (overleaf).

Along with the **Section View Toolbar**, is a **Mini View Toolbar**, which contains a series of Zoom functions such as **Resize to Fit**  , **Zoom In**  and **Zoom Out**  .



Section View with Pins Displayed

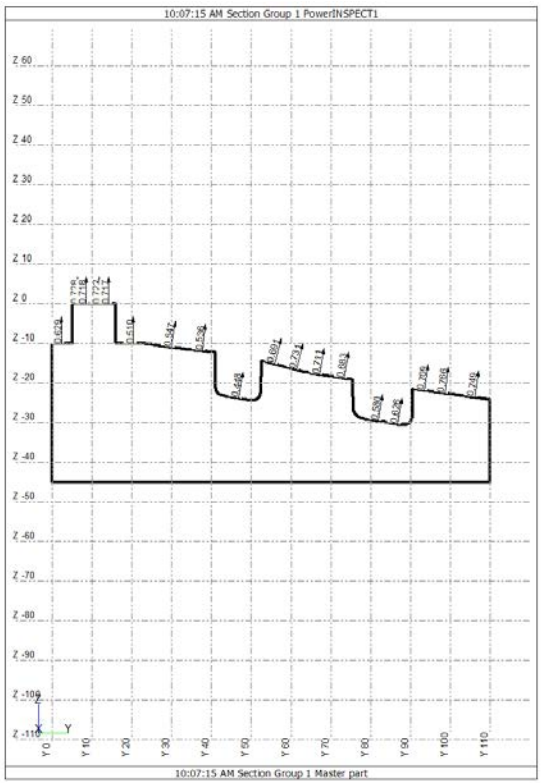


Section View with Linked Pins Displayed

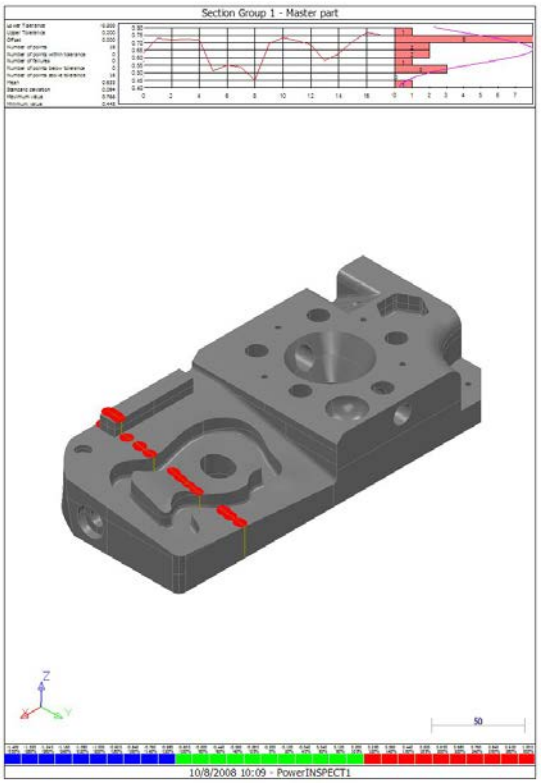
When the **Print Preview Button**  is selected, or the **Print Preview** option is chosen from the **File Menu**, then a preview of the **Section View** (or the **CAD View** of results), along with the **Probed Points**, is shown on the screen.

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

Changing the display icon will alter the view. In the example to the right the points are shown as **Vectors**.



Section View Print Preview



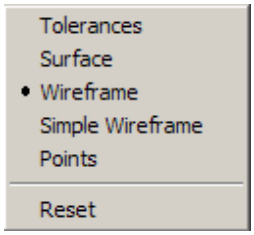
CAD View Print Preview

4.5 Geometric Feature Inspection *(using Wireframe Checker.)*

The **Wireframe Checker** button  allows you to pick geometric features directly from a CAD model in the CAD view. The details of the feature you select are automatically included 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" in an empty space in the CAD view and select from the local menu



Surface

You can use the Surface-picking mode to highlight a whole CAD surface. If the surface is part of a complex 3D-wireframe entity, PowerINSPECT extracts the entity's details.

Wireframe

You can use the Wireframe mode to highlight a complex 2D-wireframe-entity piece of wireframe, 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. You can also use this mode to select arcs (circles).

Simple Wireframe

You can use the Simple Wireframe mode to highlight 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.

Point

You can use the Point mode to pick a single point on the CAD wireframe. If you pick two points, PowerINSPECT constructs a line. If you pick three points, PowerINSPECT constructs an arc.

.

When an item is selected from the CAD model, the **Nominal data** is displayed on the **Geometry Explorer Tab.** If the item is accepted,  then it creates a new item in the **Inspection Sequence.**

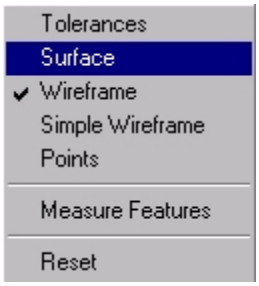
Feature Inspection Example

- Create a **New Document using the Wizard**  and select 

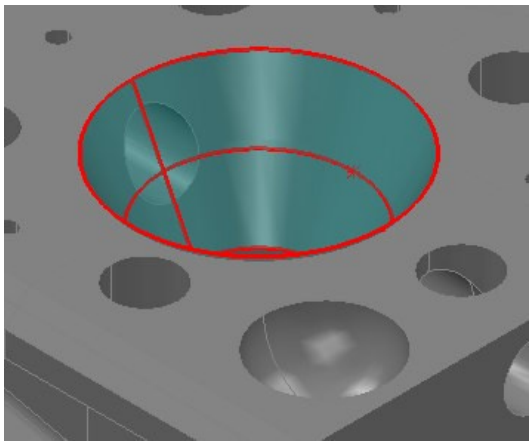
.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Create an **Alignment** (if necessary), using one of the previously learnt methods.
- Ensure the **Delay Measure** Button  from the **Main Toolbar** or **Measures Menu** is selected.
- Ensure the **Activate sequence items** Button  from the **Geometry Explorer Tab** is selected.
- Switch on the **Wireframe Checker** button 

Surface features / components will be selected from the CAD model.

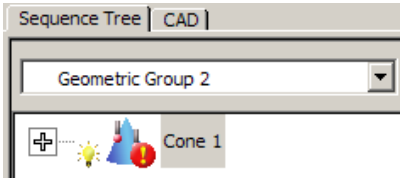
- Right-Click in an empty space in the CAD view and choose **Surface** from the local menu.



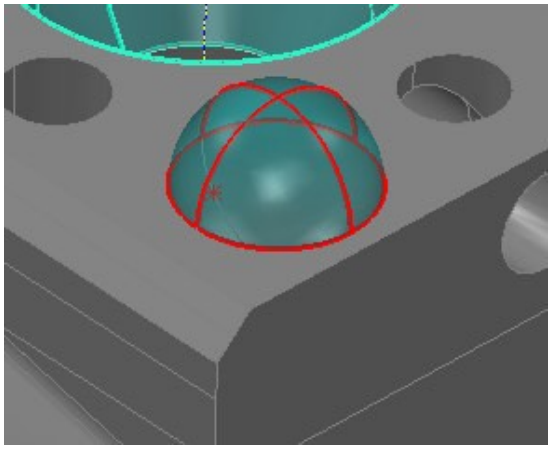
- Select the central **Cone** feature on the CAD model and then select the green tick  in the **Geometry Explorer** to accept.



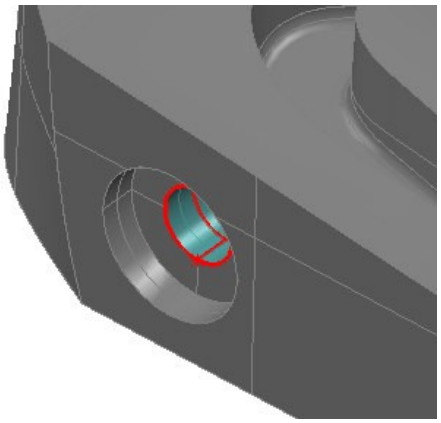
The feature has been added to the **Sequence Tree** within a new **Geometric Group**.



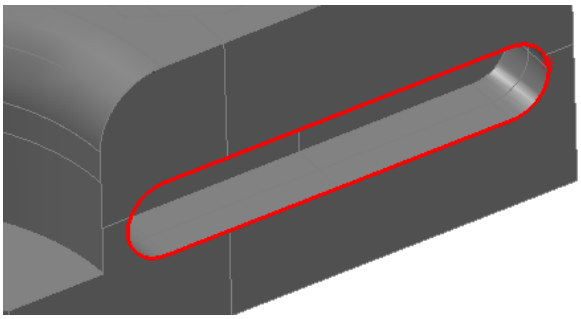
- Select the **Sphere** feature on the CAD model and then select the green tick  in the **Geometry Explorer** to accept.



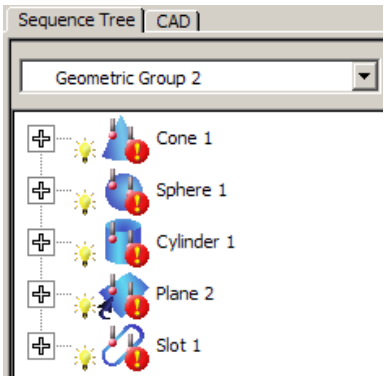
- Select the **Cylinder** feature on the CAD model and then select the green tick  in the **Geometry Explorer** to accept



- Change the picking mode back to Wireframe.
- Select the **Slot feature** on the opposite face to the cylinder. Change **linked to - New Probed Plane** before selecting  .



The sequence tree lists all items defined.



- Select **Play All** 
- Check the Measurement results in the **Report.**

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**	-

5. Measurements without CAD

Introduction to Measuring Without CAD

PowerINSPECT does not have to work with **CAD Data**. **Physical Parts** can be inspected for dimensional measurement and comparison without reference to a model. Measurements are created using **Geometric Elements**, which are either probed, or referenced to probed elements.

Preparing a Session for Measurement Inspection

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

- Select **Create a New Document**.

A new empty session has loaded

The new session is now ready for inspection.

Generating a Measurement Inspection

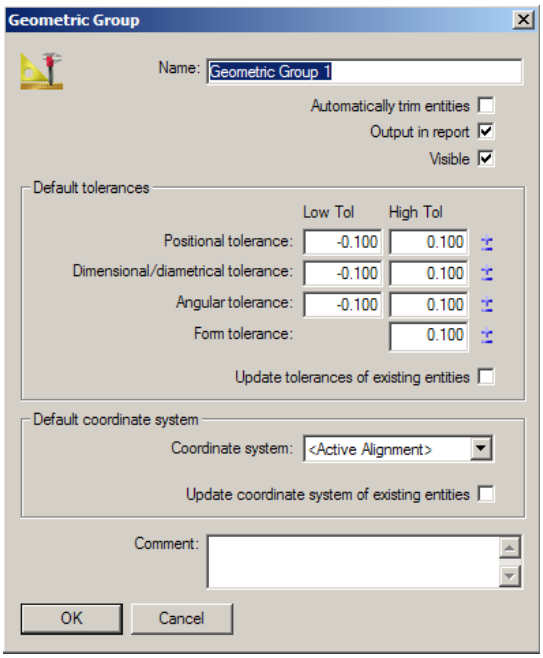
A **DemoBlock2008** (pictured right) will be used for measurement. The block should be aligned with the long facing (Delcam logo) towards the front of the CMM/Table

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 PLP generated from the elements probed.

- Select the **Delay Measure Button**  .
- From the **Elemental Toolbar**, choose the **Geometric Group**  Button.

The Geometric Group dialogue box appears and prompts the user to name the group and add a comment if required.

- Name the group **Datum Group**
- Untick Output in Report
- Leave everything else as **Default** and choose **OK**.



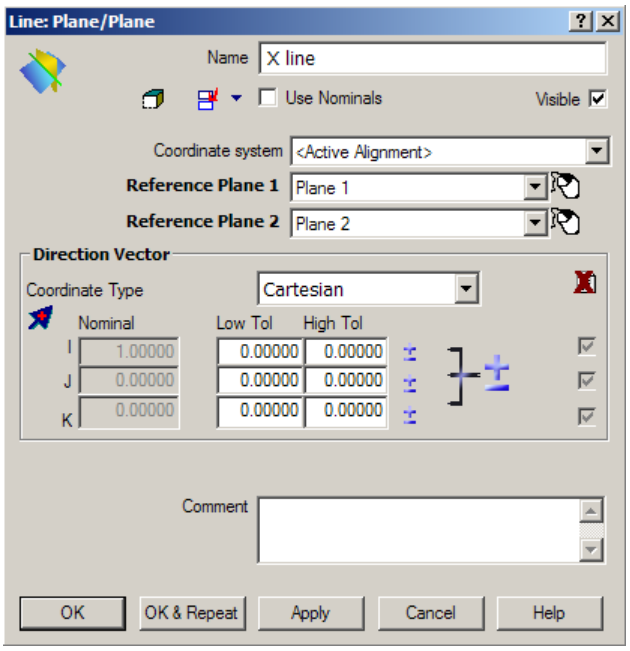
Now a series of **Geometric Items** need to be created for measurement from the **Elemental Toolbar**. For this example six items will be created. These include three **Probed Planes** defining the **X**, **Y** and **Z** planes, two **Plane Intersection Lines** for the **X** and **Y** axes and a **Line Intersection Point** for the Origin.

- Select the **Planes Button**  , and choose the **Probed Plane** 
- Use **OK & Repeat** to create three planes.

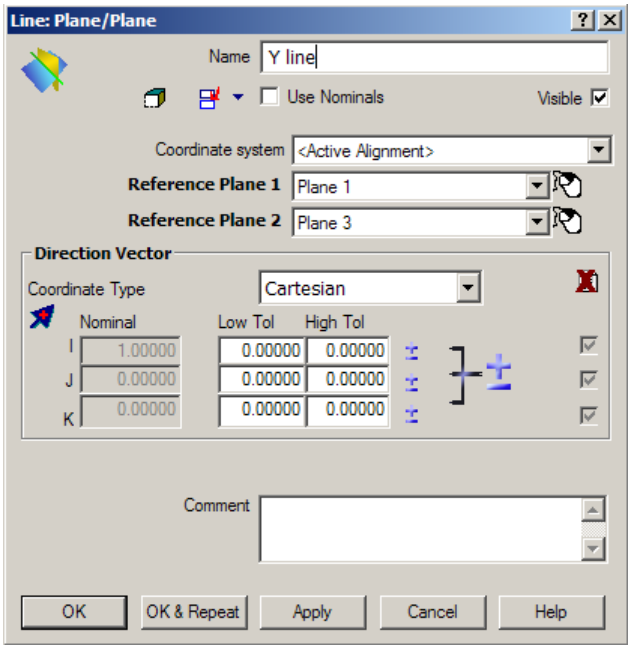
Plane 1 will be Probed from the CMM bed and will define the Z 0 plane
Plane 2 will be Probed from the left hand face of the block defining the Y 0 plane
Plane 3 will be Probed from the front face of the block defining the X 0 plane

- Create a Line called **XLine** using the **Line: Two Planes** Intersection Icon from the lines menu  filling in the form as shown below;

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 will be used to define the X axis in the PLP later.

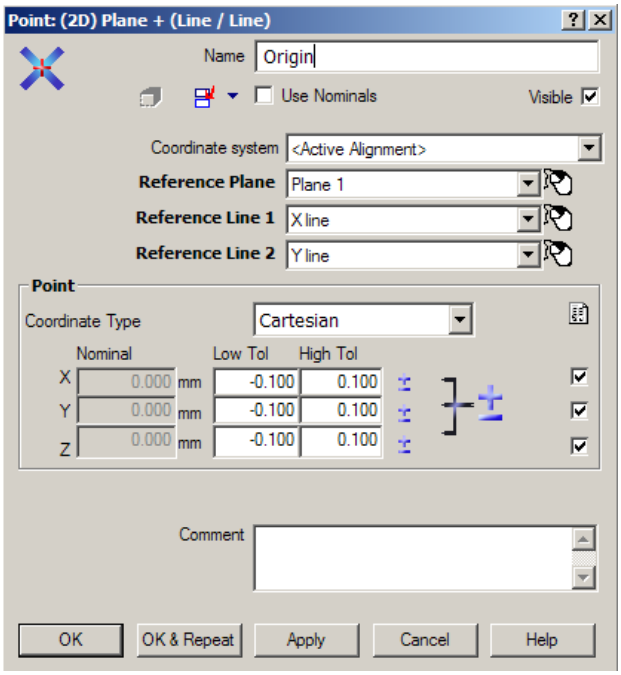


- Complete the form as shown and press **OK & Repeat**
- Repeat the process to create a second line called **Yline** defined by the intersection of Plane 1 & Plane 3



Finally a point will be created at the intersection of **Xline** & **Yline**.

- Select the **Point from Lines Intersection** Icon  from the lines menu and fill in the form as below, then press **OK**



- Complete the form as shown and press **OK.**

This point will be used as the **XYZ origin** in the **PLP.**

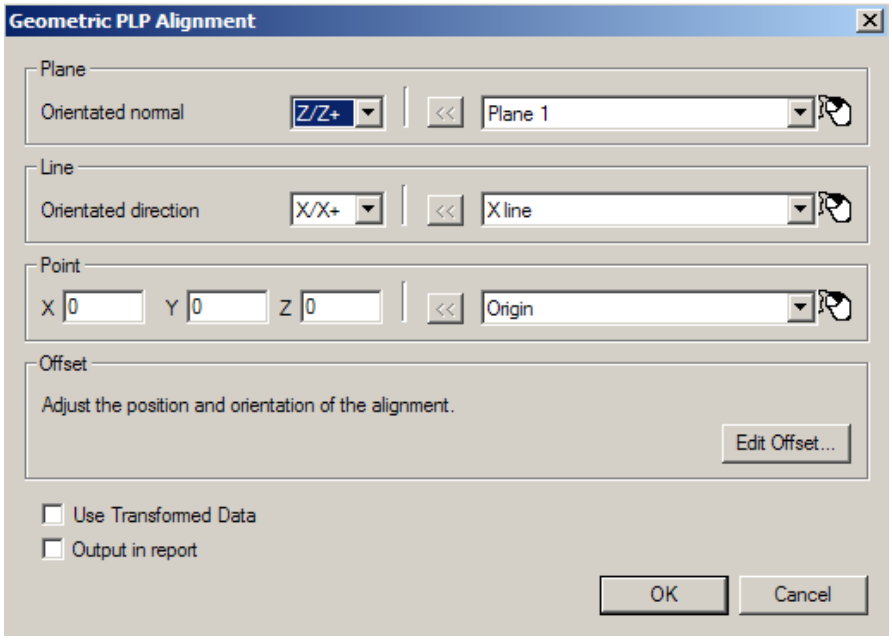
- Once the Geometric Elements have been defined select **Play All**  and probe the three planes in the order described above (CMM table, Front Face, Left Face).

The Lines and Point will be generated automatically from the Probed Planes.
With the Geometric Elements now defined and measured we can create the PLP alignment.

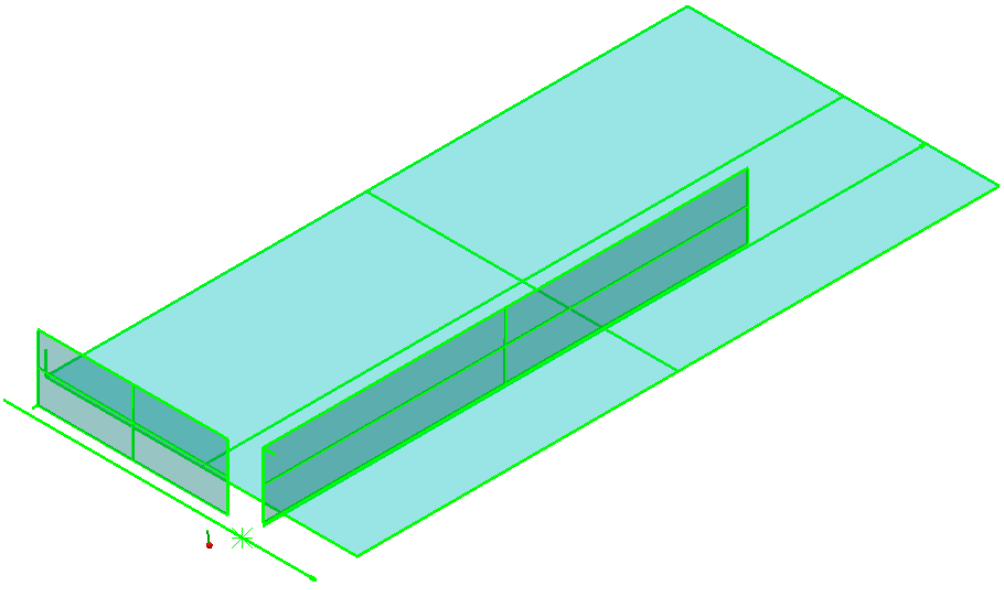
- Move up the levels  until **Geometric Group 1** is **Closed**.

- From the **Alignments Sub-Menu** icon  , in the **Element Toolbar**:

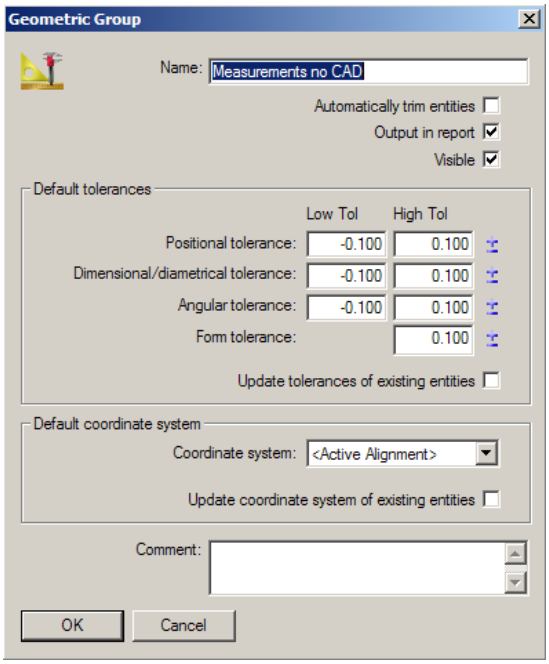
- Choose the **Geometric PLP Alignment** button  .
- Fill in the form as shown below and press **OK.**



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



- Create a new Geometric Group  filling in the form as follows;

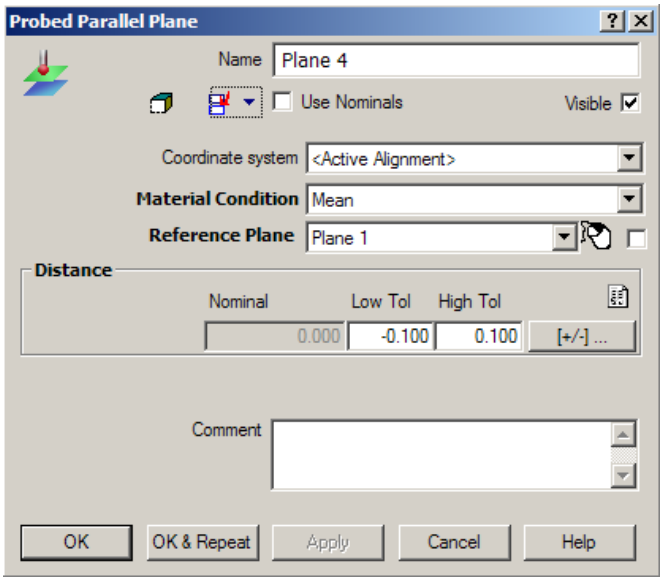


Please note that although the measurements will be output in the report there will be no CAD nominals to report against. If a drawing is available the nominals can be set by modifying each Geometrical Element to be probed.

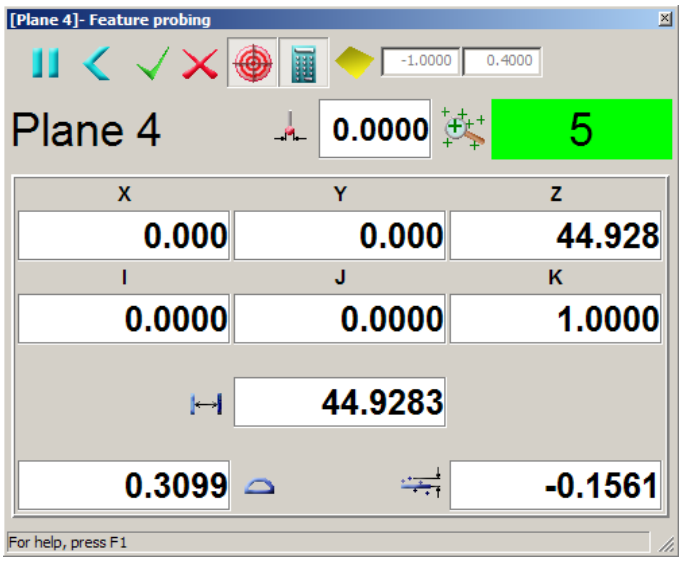
The first dimension to be measured is the height of the block. In this case a **Probed Parallel Plane** will be used. This will measure the height of the block.

- Select the **Planes Button**  again and choose the **Probed Parallel Plane**  (using **Plane 1** as the **Reference Plane**)

- Fill the form in as shown

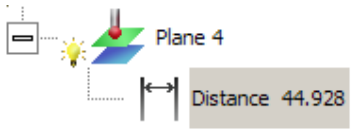


- Press **OK** and then **Play**  the item.
- Probe the horizontal top face of the block.



The Feature Probing form shows the Distance between the CMM surface and the top of the block (44.928mm), the Flatness of the top surface (0.3099mm) and the maximum deviation of points measured from the best fit plane (-0.1561mm). It should be noted that the latter two items are not output to the report.

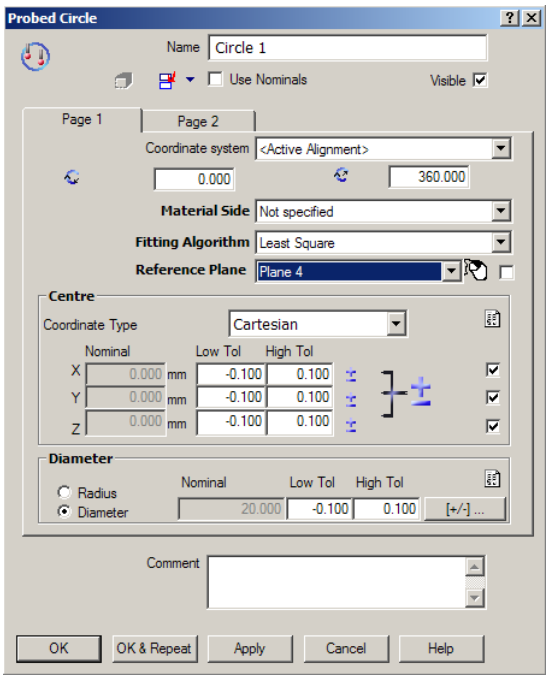
In the sequence tree it can be seen that the height of the plane has been updated



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

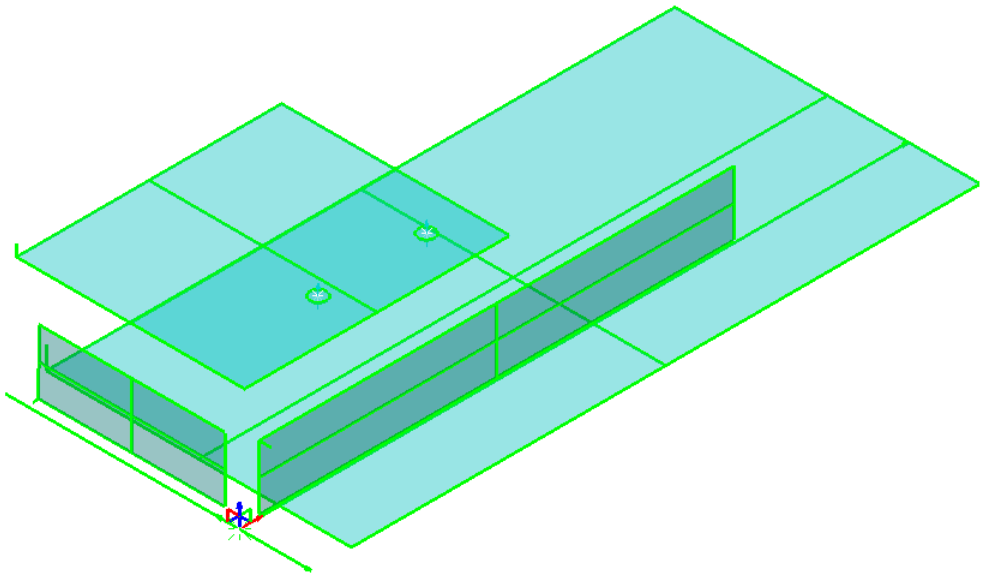
- Open  the **Measurements No CAD** group, if not already inside.

- Select the **Features Button**  , and choose the **Probed Circle**  , (referencing it to the top **Plane 4)**.

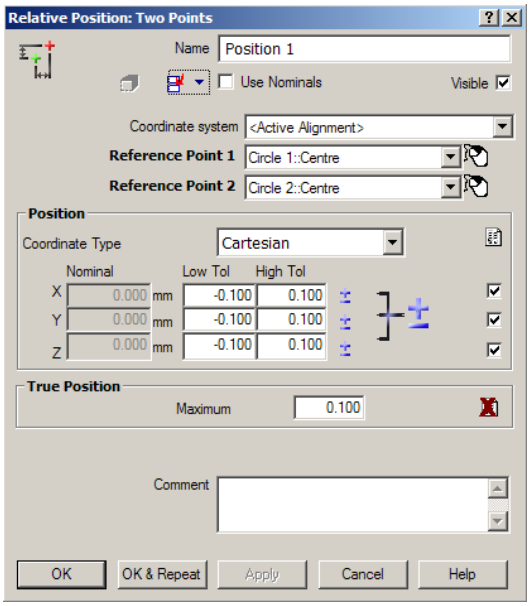


- Press **OK and Repeat** to create a second **Probed Circle**

- **Play** both items and measure the circular holes in the block using the techniques previously learned



- Select the **Dimensions Button**  , and choose the **Relative Position Button**  .
- Fill in the form as below and press **OK**

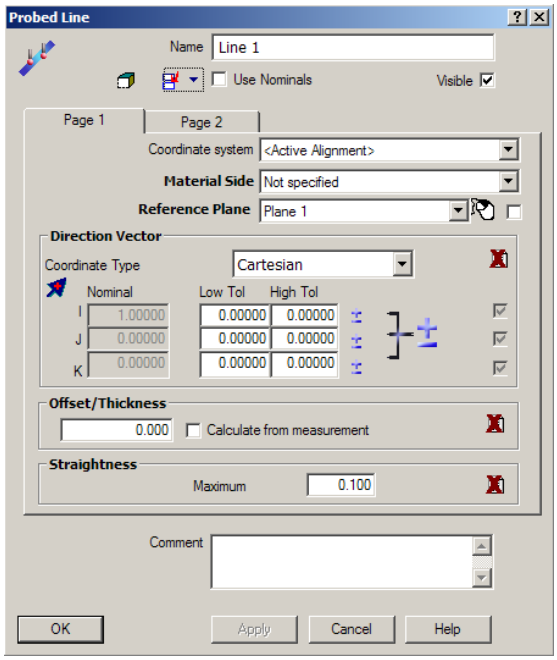


From the sequence tree the relative positions between the two circle centres can be seen referenced to the X & Y axes of the PLP.

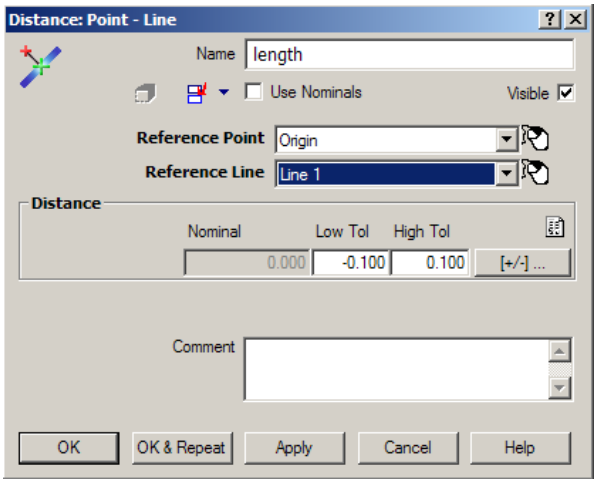
Finally we will measure the overall length of the block in the X and Y directions. The distances will be measured using the Origin Point created at the beginning of this section plus two new Probed Lines

- Select the **Probed Line**  Icon and create 2 probed lines.

Ensure Plane 1 is selected as the reference plane (Table)

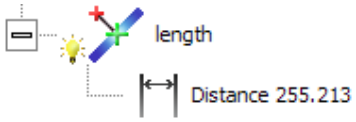


- Play the lines, measure the first on the **Right** of the block and the second on the **Back (parallel to plane 2)**.
- Now select the **Dimensions Button**  , and choose the **Line Point Distance** Icon 

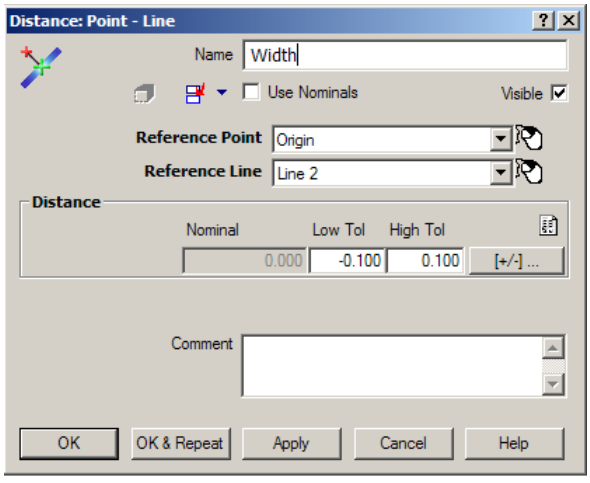


- Fill in the form as shown

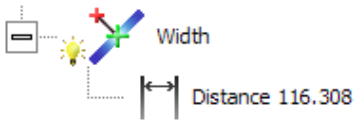
Line 1 is on the Right hand Face of the block, Origin is at the PLP datum origin. The distance between them gives the overall length of the block in X. This is shown in the updated Sequence Tree below.



- Repeat by selecting the **Dimensions Button**  , and choose the Line Point Distance Icon 
- Fill in the form as shown below;



Line 2 is on the back Face of the block, 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

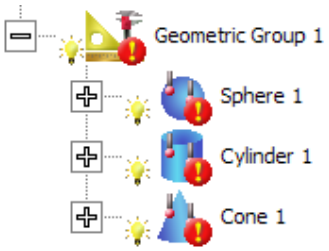


Probing 3D Features

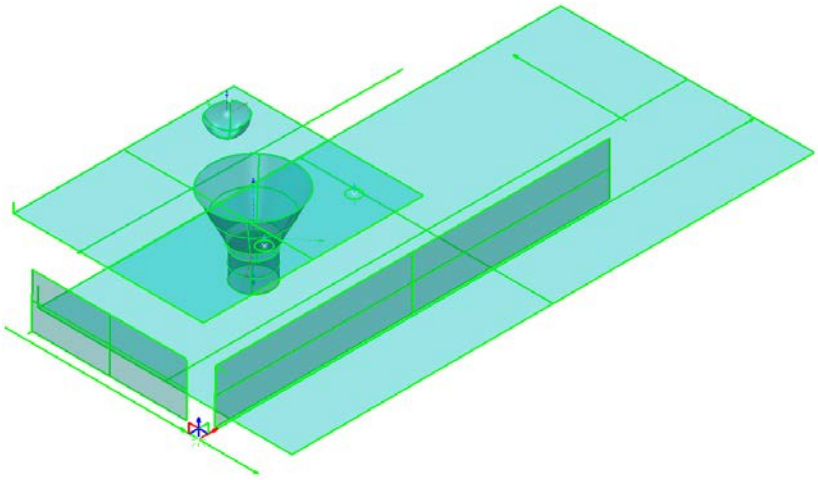
3D features will be added to the inspection.

- From the Elemental Toolbar, create a second Geometric Group  .
- From the **Features Button**  , select Probed Sphere  , Probed Cone  and Probed Cylinder  . (keeping the default settings)

The sequence tree lists the defined features.

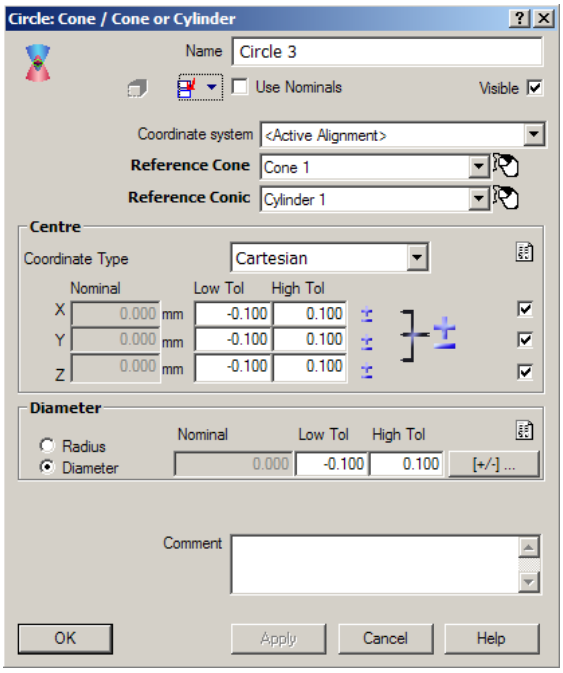


- Select **Play All**  and probe the three features in the order defined.



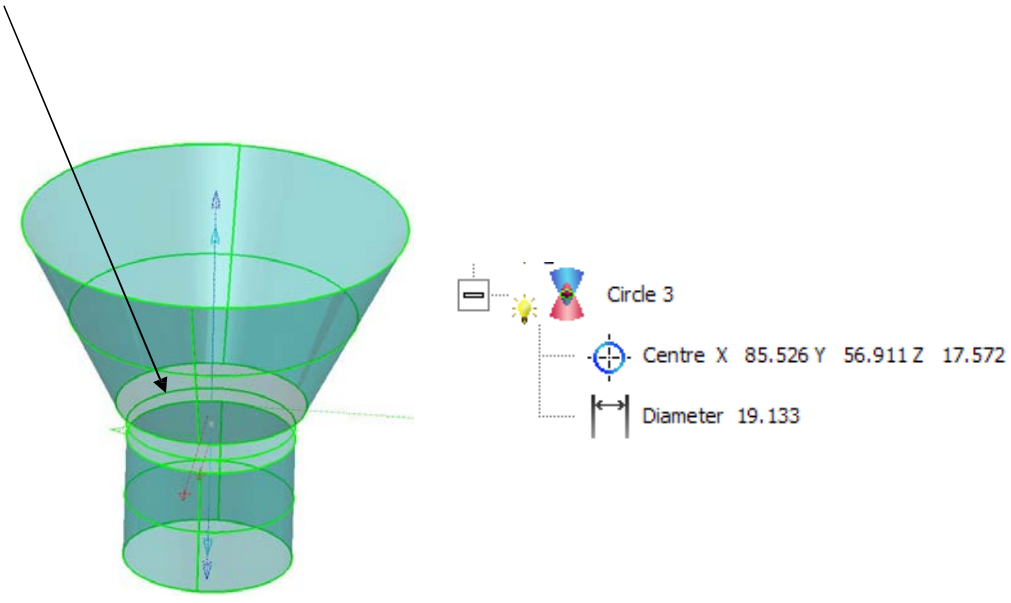
The CAD view is updated to show the new probed features

- Inside Geometric Group 1, Select **Circle: Cone Cylinder intersection**  from the **Features Button**  .



- Select the **Cone and cylinder** as reference features, and select **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 Icon in the Sequence Tree to open each Geometric Element in turn and change the nominal value.

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
Centre	X	-----	-----	-----	59.620	-----	-----
	Y	-----	-----	-----	25.391	-----	-----
	Z	-----	-----	-----	44.928	-----	-----
Diameter		-----	-----	-----	7.134	-----	-----

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

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

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

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Position	X	-----	-----	-----	47.005	-----	-----
	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	-----	-----
Centre	X	-----	-----	-----	111.905	-----	-----
	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
Centre	X	-----	-----	-----	85.526	-----	-----
	Y	-----	-----	-----	56.911	-----	-----
	Z	-----	-----	-----	17.572	-----	-----
Diameter		-----	-----	-----	19.133	-----	-----

There are many other ways in which it is possible to measure dimensions from a Physical 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 the need of a CAD model**. This can be useful for example, for reverse engineering applications. **Point Cloud** and **Points on the Fly** are two methods outlined below. The points taken will be included into the current **Inspection Sequence.**

Point Cloud Group 

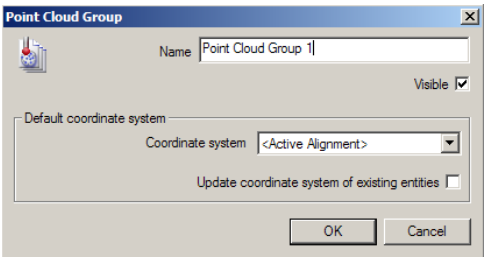
Point cloud (3D digitised data) can be obtained in two ways.

1. **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.
2. **Non-contact** – Using devices such as line or scanning point lasers.

Point cloud data will be included in this inspection sequence.

- Create **Point Cloud Group**, by clicking on the Point Cloud Group Button  on the **Element Toolbar.**

- Keep all the **Default Settings**, and choose **OK**.

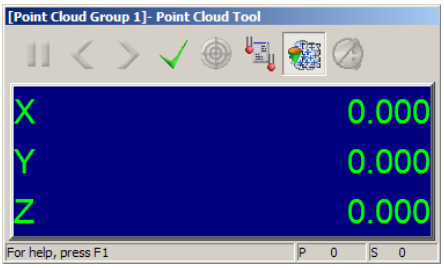


The Element Toolbar is now replaced with the Point Cloud Group Toolbar.

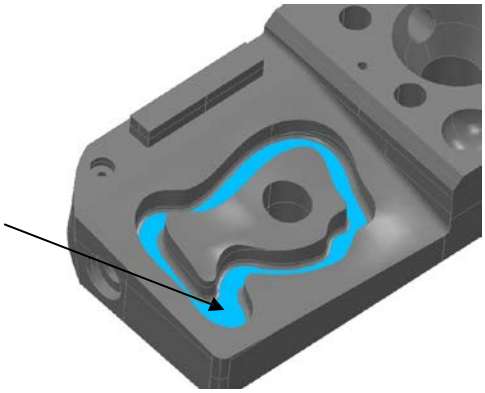
Button	Description
	**Create a point cloud by probing points on the fly**. Allows you to probe points on the part and PowerINSPECT saves them as a point cloud. You can project this point cloud to the CAD model to view the deviation between them.This button allows you to delete all the points already probed in order to re-probe them.
	**Sketch a digitised curve**. Allows you 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.
	**Import a cloud of points from a file**. Allows you 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' to find the deviation between the probed points and your CAD data.

- Select **Insert a Point Cloud**  to probe points on the fly.

The **Point Cloud Tool Window** appears ready to take points.



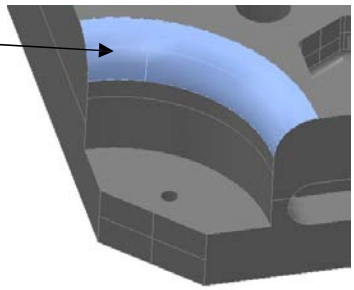
- Scan/Take points on the surface profile (as shown in blue) using the appropriate method for the probe type.



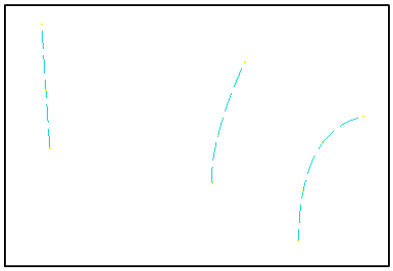
Once taken and accepted  , the points are displayed in the CAD view (using ball probe)

- Still within **Point Cloud Group 1**, select the **Curve** option. 

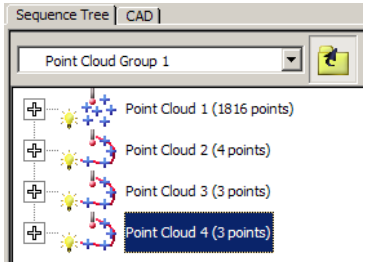
- Sketch 3 curves by taking points across the curved edge radius (as shown in blue).



Once taken and accepted,  the three curves are displayed in the CAD view.



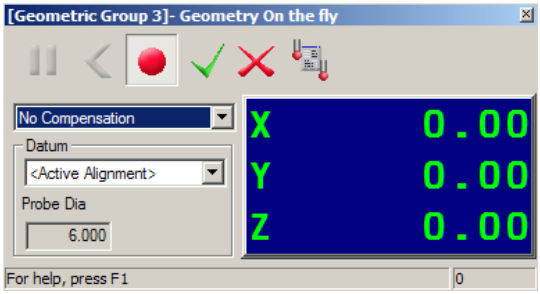
The **Point Cloud Group 1** in the sequence tree is updated.



Single Points on the fly 

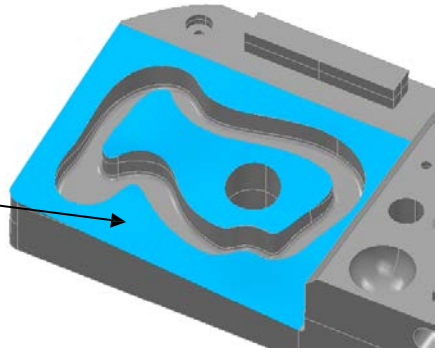
Single points can also be taken as geometric elements anywhere on the part. Single points on the fly are the same as guided single points except the nominal value does not have to be entered (to which you are guided to take a point).

- From the **Element Toolbar**, create a new **Geometric Group**  .
- Select **Single Points on the Fly.** 

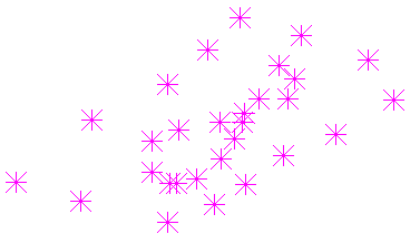


The **compensation drop down menu** can control whether probe compensation is required on the points taken.

- Probe a number the sloping surface on the block (as shown in blue).

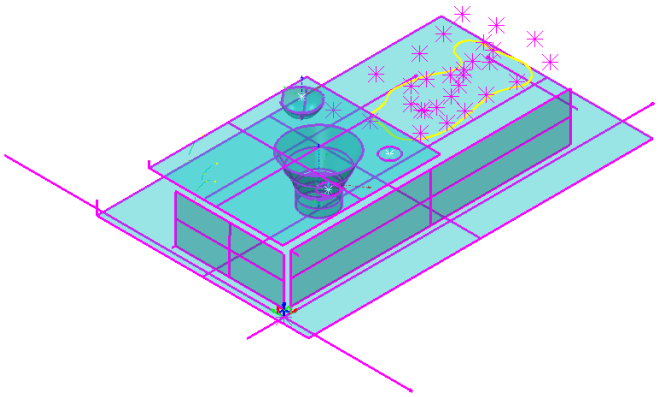


Once taken and accepted  , the points are displayed in the CAD view (using ball probe)



The CAD view shows all the Geometric features and points that have been taken.

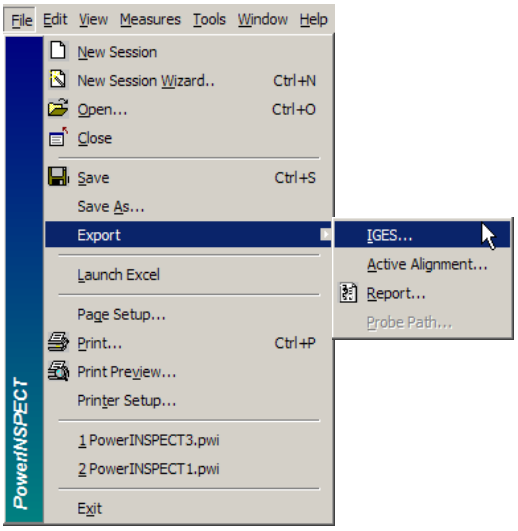
These will now be exported from PowerINSPECT.



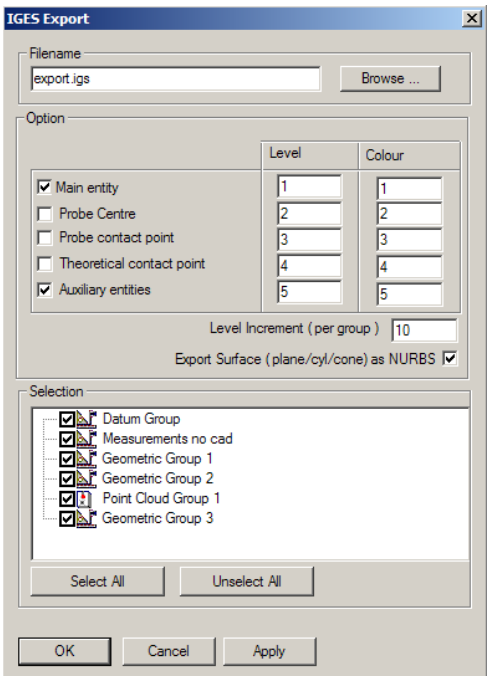
File Export - IGES

Geometric entities and surface points can be exported from PowerINSPECT to an IGES file. This therefore allows the transfer of PowerINSPECT data to CAD/CAM packages, e.g. for reverse engineering applications.

- From the **File** menu select **Export – IGES.**



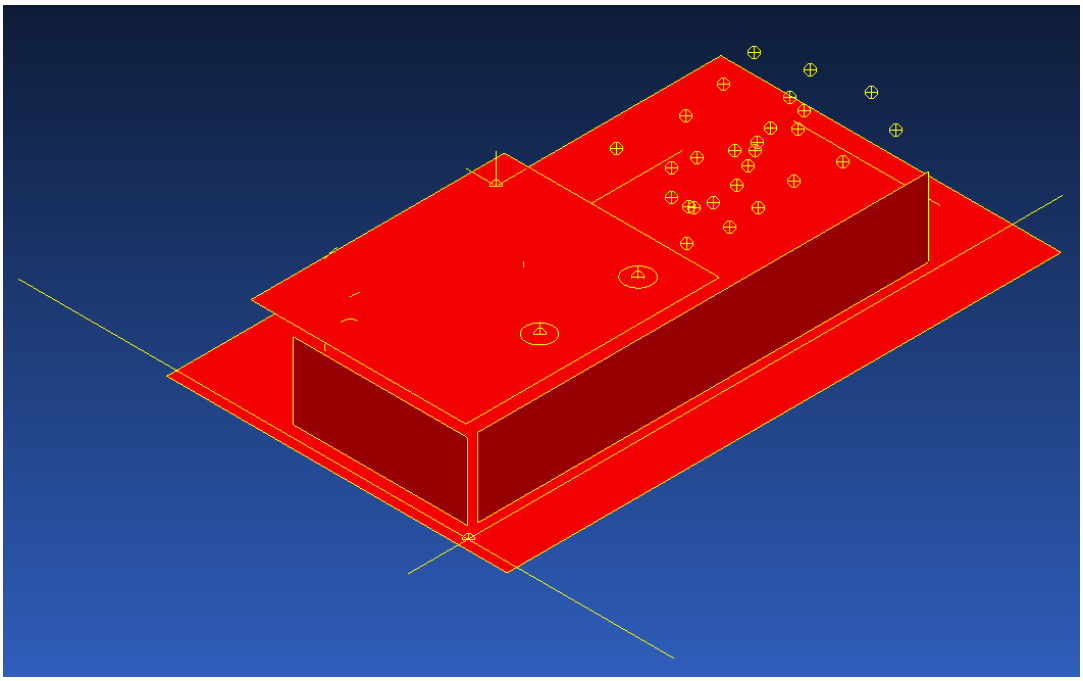
You can export the elements in a NURBS format. NURBS surfaces are the preferred format for many CAD systems, including PowerINSPECT. If you don't select the NURBS option, the 3D elements are saved in a Surface of Revolution format and planes are saved as Rule Surfaces



- Define a path to export the file and then select OK to save.

The data can be imported into a CAD system and manipulated as required.

The example below shows the data imported into **Delcam PowerSHAPE** where the probed planes and features are now represented as **NURB surfaces**.



6. Inspection program

Inspection Program Example.

The following working example creates a simple **Inspection program**. The program includes various features shown in previous chapters, including a **PLP** alignment, **Geometric Feature** measurement, **Surface**, **Dynamic points** and **Section** Inspection.

For this example, the ***DemoBlock2008.dgk*** model will be used.

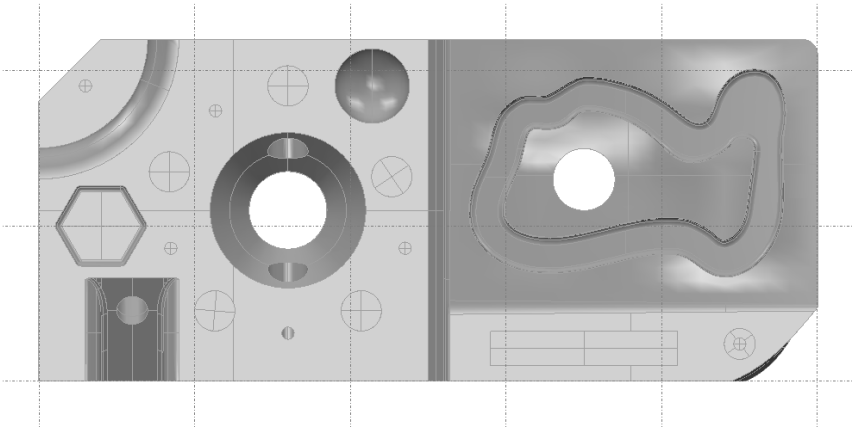
Using the methods, previously described in the course, for starting a new session:

- Create a **New Session** and **Browse** for **DemoBlock2008.dgk**. Open this file.
- Keep the **Default Settings** for **Offsets** and **Tolerances,** and choose **Next**

In the Variables dialog box:

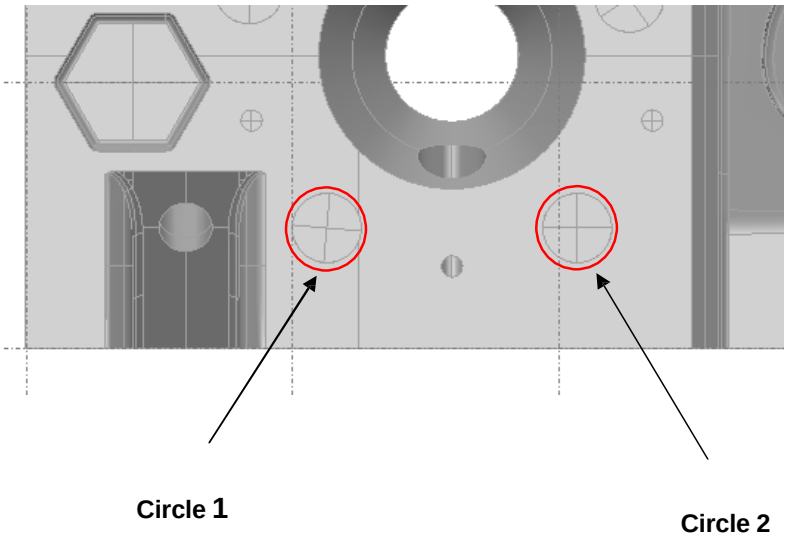
- **Browse** for any chosen **HTML Report Template** (**Excel** could be used, but for this example **HTML** is to be used), and **Extract Variables**.
- Select **Finish**.
- **Shade** the view  and then re-orientate it to a **Z** view from top. 

The new session is now ready for programming.



Geometric PLP Alignment

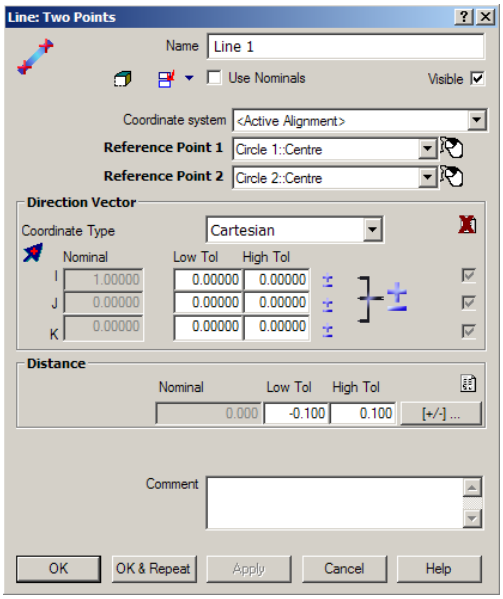
- From the Mouse context toolbar select **Wireframe Checker** 
- Select the wireframe of the hole **Circle 1** (as shown below).
- Make sure that the **linked to** option is set to **New Probed Plane.** This associates the circle with the new probed plane.
- Press the **Green tick**  button to accept the selected feature.
- Repeat the above process to select hole **Circle 2.**
- Select **Plane 1** from the **linked to** option.
- Press the **Green tick**  button to accept the selected feature.



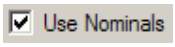
To satisfy **PLP** criteria, a **Line** is to be created between the **Centres** of the **Two Circles**, and therefore the **Line Between Two Points** option is required for the item.

- Select the **Lines Button**  , and choose the **Line: Two Points**  .

- Set **Reference Point 1** as **Circle 1: Centre**, and **Reference Point 2** as **Circle 2: Centre**.

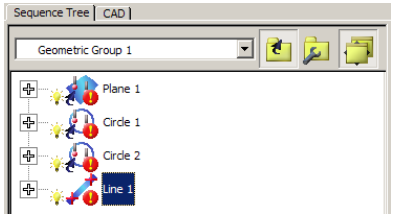


- Use the **Nominal Loading** drop-down button to load the line values from the **CAD Entity**.

- Tick  to display the nominals in the report.

- Leave the other settings as **Default**.

- Choose **OK**.



The **Line** has now been created and the **Sequence Tree** has been updated to include ***all*** the **Geometric Items** (left).

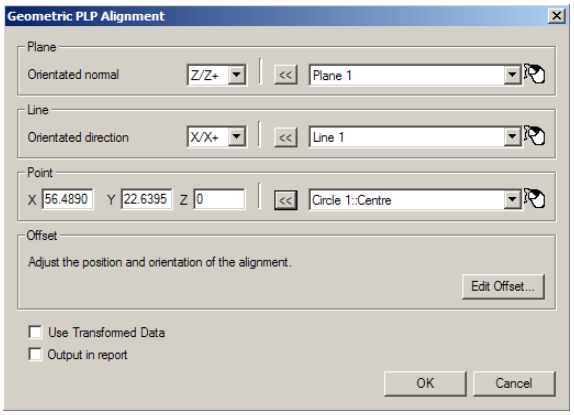
- **Close** the Geometric group by selecting **Up One Level**  .

- From the **Alignments Sub-Menu** icon  , in the **Element Toolbar**:

- Choose the **Geometric PLP Alignment** button  .

- **Select** the **Circle 1: Centre** as the point element, and use the **Load Nominals** button  to load the CAD Nominals for all three items.

- Select **OK.**



Geometric Feature Inspection

2D/3D Geometric Features will now be selected to begin the Inspection Sequence.

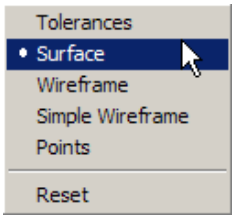
- From the **Element Toolbar** select the **Geometric Group** button  , and accept the **Defaults**.

In this group, the Slot and Cone will be measured.

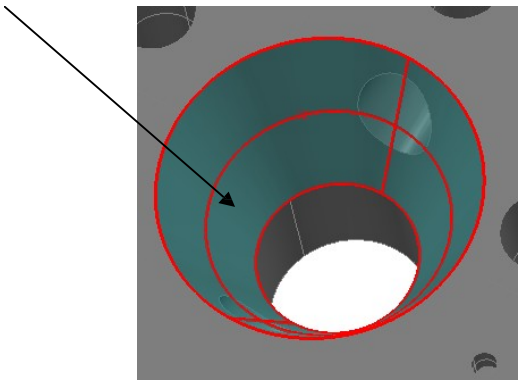
- Ensure the **Wireframe Checker** button  is switched on.

Because the cone is a surface, the Geometry Explorer needs to be set up to look for Surface Components.

- Right-Click in an empty space in the CAD view and choose **Surface** from the local menu.



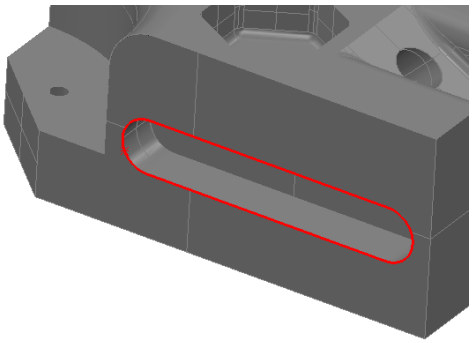
- **Select** the **Cone** in the centre of the model, and choose the green tick  to accept the feature.



Because the slot is a Wireframe item, the **Wireframe Checker** needs to be returned to the Wireframe state.

- Right-Click in an empty space in the CAD view and choose **Wireframe** from the local menu.

- Select the Rounded slot on the side of the block.



- Ensure the **linked to** option is set to **New probed plane**, before selecting the green tick. 

- **Close** the Geometric group by selecting **Up One Level**. 

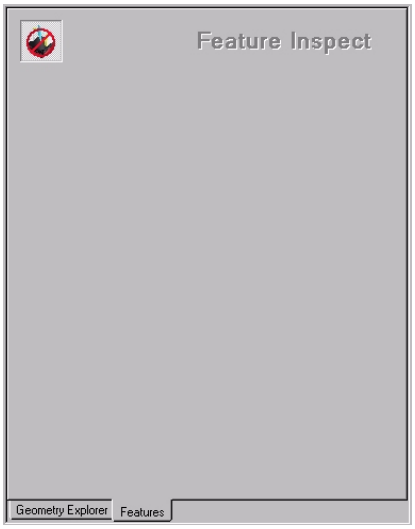
Bouncing Ball

"Bouncing ball" is an on-screen guide which aids the user as to what has already been measured and what is to be measured next. Points are displayed as spheres in different colours to guide the user during the inspection. During inspection, the following default colours will be displayed:

Blue sphere: Initial colour of unmeasured points.
Red sphere: Next point to be measured.
Grey sphere: After point has been taken
Transparent Red: Actual position of point taken/probed.

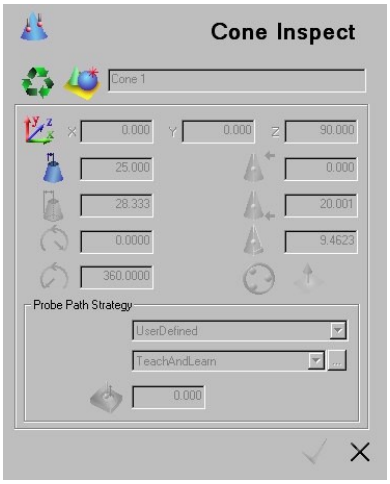
To illustrate this functionality, the **Bouncing ball** will be applied to the two Geometric Features defined above.

- Open **Geometric Group 2** in the **sequence tree** to show both geometric features.
- At the **Geometry Explorer**, select the **features tab**.

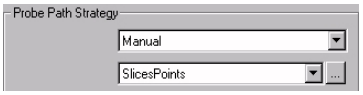


The CNC mode option is currently switched off  . In order to use **Bouncing Ball**, this option needs to be activated.

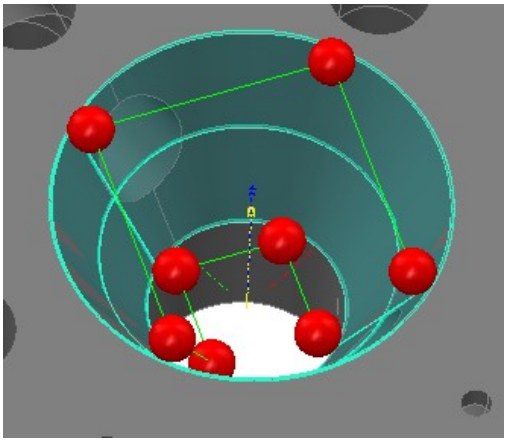
- Toggle the icon to switch CNC mode **ON**. 
- Highlight **Cone 1** in the **sequence tree,** which will update the features tab.



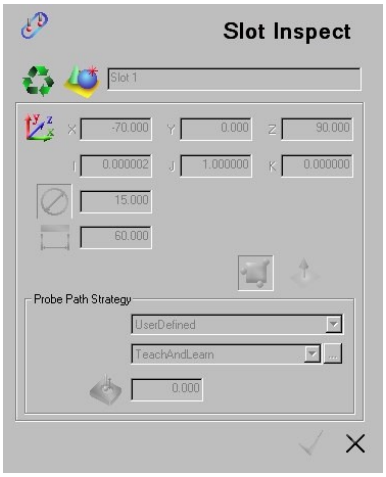
- Select the recycle button  to edit the options.
- From the **Probe path strategy** drop down list, select **Manual** and then **Sliced points**.



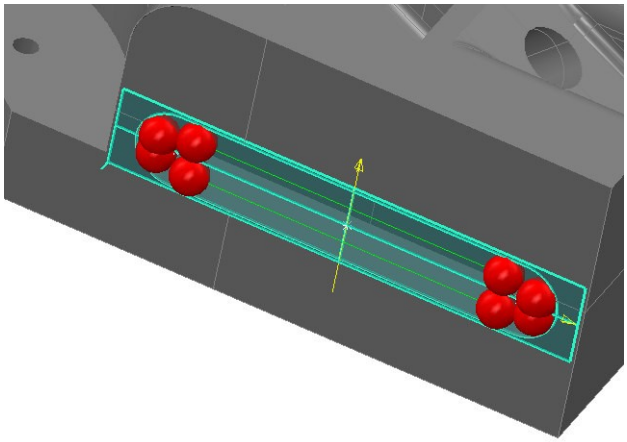
The cone now shows red spheres as the probe points and green lines/arcs as the shortest distance between the points.



- Select  to accept and save the geometric feature.
- Highlight **Slot 1** in the **sequence tree,** which will update the features tab.



- **Repeat** the above procedure for the **Slot feature** but also change **Probe externally**  to **Probe internally**  .



- Select  to accept and save the geometric feature.
- Select  to close the dialogue and then select the **Geometry explorer tab.**

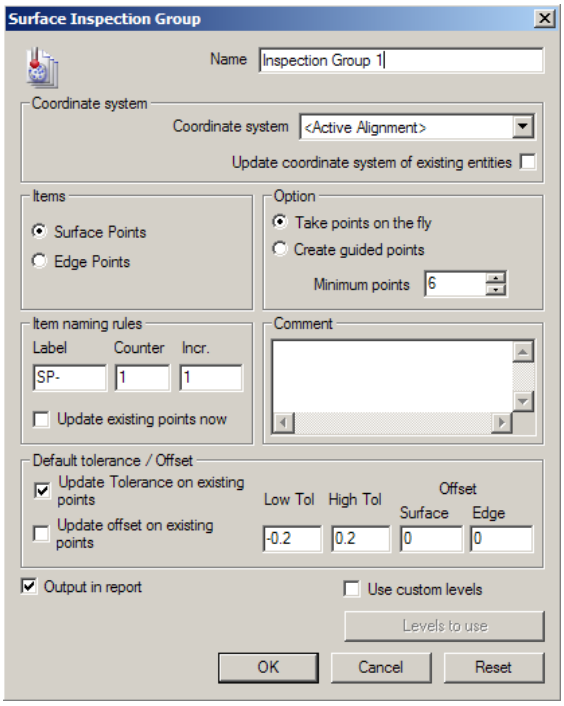
The bouncing ball feature is now applied to the two geometric features. This will be activated in the sequence tree during “play” mode.

- Switch **OFF** CNC mode 

Surface Inspection

The next stage is to perform a random Surface inspection.

- Create a **Surface Inspection Group**, by clicking on the **Surface Inspection** button  on the **Element Toolbar.**



The form shows a random (**Take points on fly**) **Surface Inspection** with a **minimum number of points** to probe as 6.

- Leave all settings as **Default.**
- Choose **OK.**

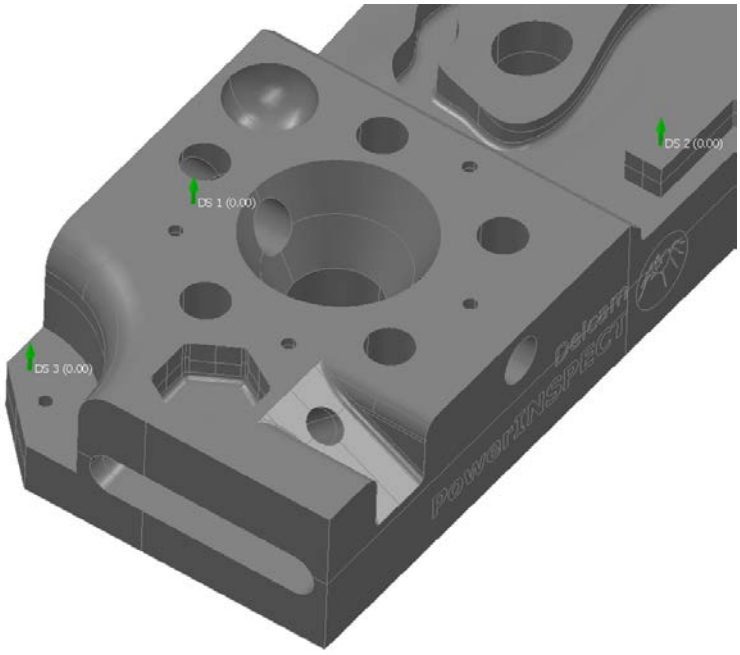
This inspection group has been added to the Sequence tree.

Guided Surface Inspection

This will create another surface inspection but instead using **Guided Points**, The **Dynamic Points Editor** will be used to specify the specific points to be probed. This was discussed in Chapter 6.

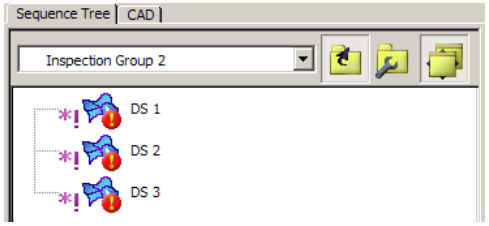
- Select the **Dynamics Points Editor**  from the **Mouse Context** toolbar.

- Double-click on the model surface approximately in the areas shown below, to specify the three **Dynamic Points.**



- Select **Create a Surface Guided Inspection Group**  from the right hand menu bar.

In the sequence tree, an Inspection Group has now been created including the three dynamic points.



- **Close** this Surface Inspection group by selecting **Up One Level**  .

- Switch off the **Dynamic Points Editor** by selecting  again.

Finally, a **Section Inspection** will be included in the program.

Section Group Inspection

The same method will be used as described in Chapter 8.

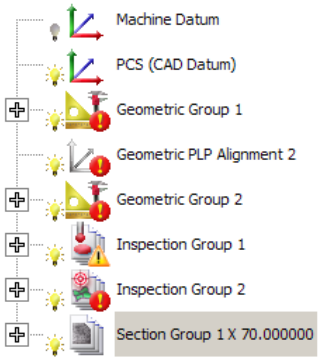
- Orientate the model to a **Z** view from top. 

- From the **Element Tool Bar** choose the **Section Group**  **Button**.

- Modify the form to create a section on **X=70**

- Leave all the other options as **Default**, and choose **OK**.

The Program is now complete with the sequence tree outlining all the items that have been created.



Two extra features (inspection aids) will be added to the inspection sequence to assist the inspectior.

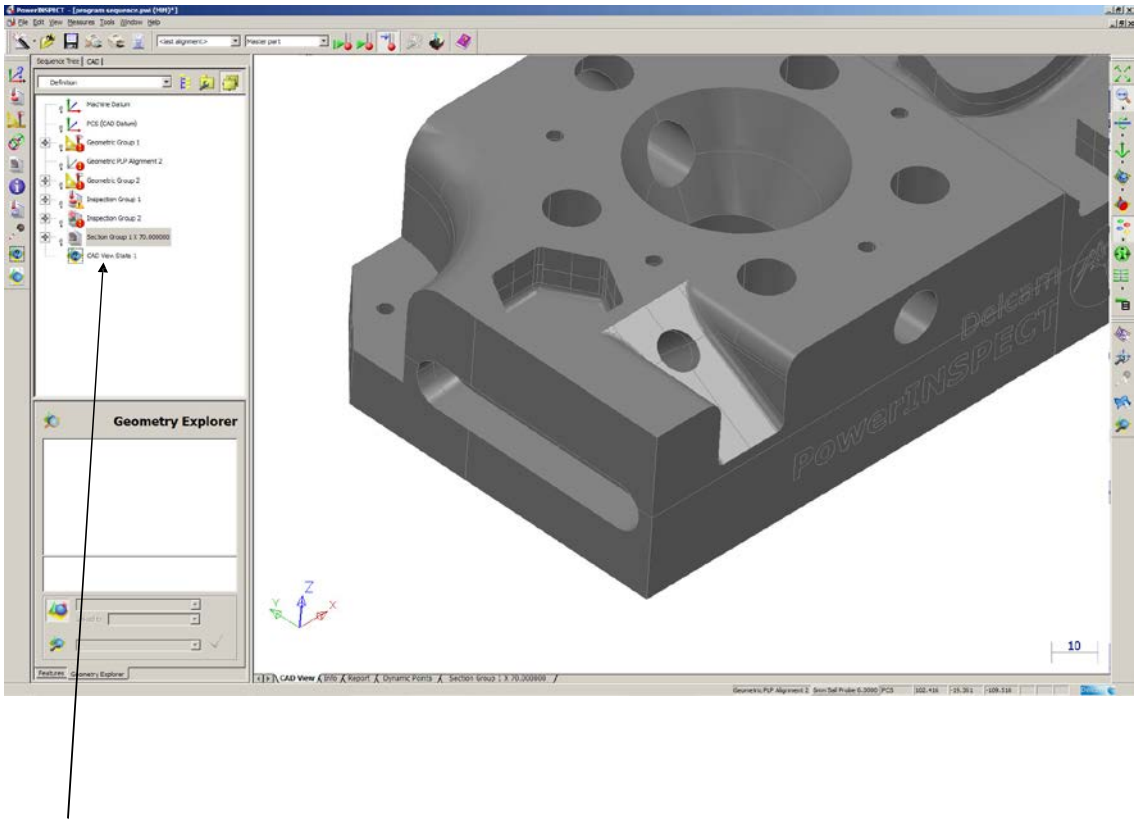
CAD View State

CAD View State  allows the user to save a current view (orientation and zoom) of a CAD view and add it to the **Inspection Sequence**.

During the inspection, the CAD view will load automatically allowing the inspector to continue probing. Not having to pause and manipulate the CAD model manually, speeds up the whole inspection process.

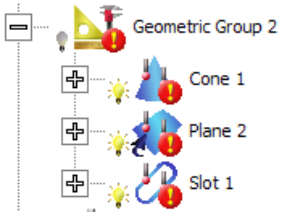
A **CAD View State** will be added to the Sequence Tree to illustrate this functionality.

- Zoom and Orientate to a suitable view around the **slot feature** and select **CAD View State**  on the **Element Toolbar**.

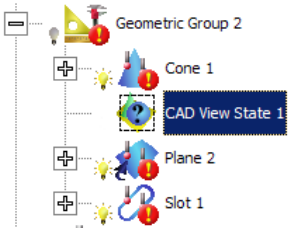


This CAD View State has been added to the bottom of the Sequence.

- Expand **Geometric Group 2** by clicking the **+** symbol adjacent to the name, to show the two geometric features.



- Select the new **CAD View State**. Keeping the left mouse button down, "drag and drop" the CAD view on top of the **Cone 1** feature.



The view is now placed just before the Slot feature, displaying the view just prior to the inspector probing the feature.

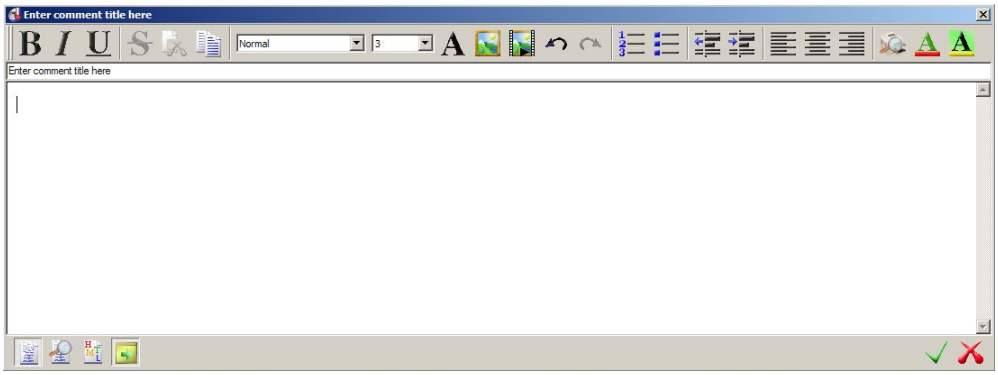
Comment

Another feature to assist the operator during inspection is by including a **Comment** box into the **Inspection Sequence.**

As with **CAD View State**, the item is placed into the sequence where the comments / information needs to be displayed.

- Select Comment  from the Element Toolbar.

This launches a dialogue box where instructions including images and videos, can be added.



- Enter the following text.

“Probe 10 points in and around the sphere feature”.

- Press the **Green tick** button to accept.

The stored comment is added to the bottom of the sequence.

- Drag and Drop the new **Comment box** on top of **Geometric Group 2**. The comment will now display just prior to the Surface Inspection

The program is now complete and ready for inspection.

- Select **Play All** from the **Main Toolbar**, or **Measures Menu.**

The screen changes to the Play Mode view indicating what needs to be measured and the minimum number of points required.

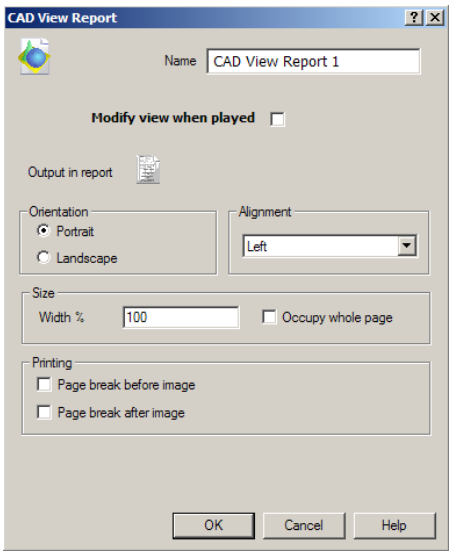
- Measure all the requested points.

- Check the Measurement results in the **Report.**

CAD view report

In addition to **CAD view state**, PowerInspect also allows a CAD images to be included in the final report. This feature is called **CAD view report**  and is also accessed from the **Element Toolbar**.

- Return to the CAD view. 
- Apply **labels**  as preferred.
- Orientate the model to a suitable view.
- Select **CAD view report**  from the **Element Toolbar.**



The form allows the user to change the Orientation, Alignment and Size of the image in the report.

- Select **OK.**
- View the final results and image in the **Report.**
- **Export**  the report.
- **Save** the **File**  in any chosen location (e.g. **C:\Temp**).
- Name the file **Inspection program.pwi**

7. Modifying Elements

Introduction to Modifying Elements

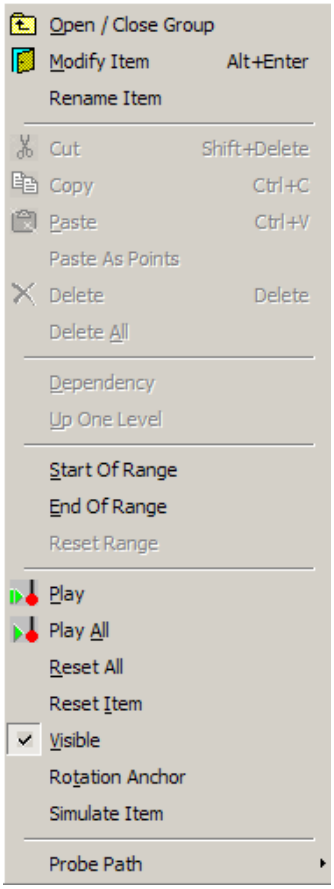
Many elements created within **PowerINSPECT** can be modified one way or another. Modifications can be made to elemental items, probed points and even **PowerINSPECT** itself (display options etc.).

This chapter briefly goes through the deletion and reordering of points, and some display modifications.

Accessing Modification

Editing or modifying items can be accessed in three ways.

The first is to choose (with the item selected) **Modify** from the **Local Right-Click Menu** (see right).



The second simply involves (again with the item selected) clicking on the **Modify Button**  in the **Sequence Tree Toolbar** (see below – and located at the top of the Sequence Tree), and the third is to use the short-cut key combination of **Alt+Enter**.

For pre-created **Geometric Items**, **Modification** brings up the dialogue box that specified the item's parameters. This allows the user to change the constraints that set the item.

For example in the **Geometric PLP Alignment** where a **Line** was specified using the **Centres** of **Two Circles**, it may be altered to pass through different elements.

In the same way, the **Alignment** itself could be modified, by bringing up the alignment box that created it to change the **Plane**, **Line** and **Point** that defined it.

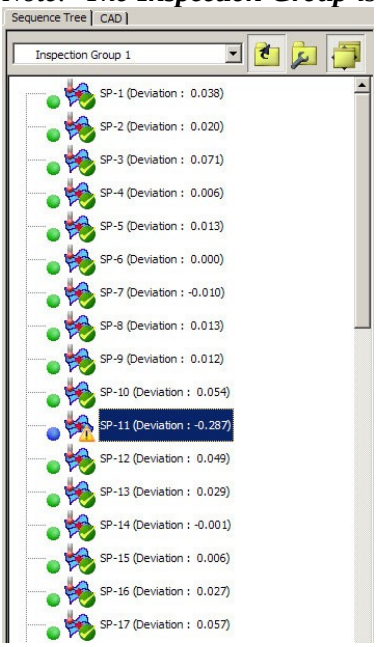
Most modifications work on similar principles, but these might affect the results. The next example runs though the deletion of an unwanted result, and demonstrates how to rectify the point numbers in order to maintain a continuous set of results.

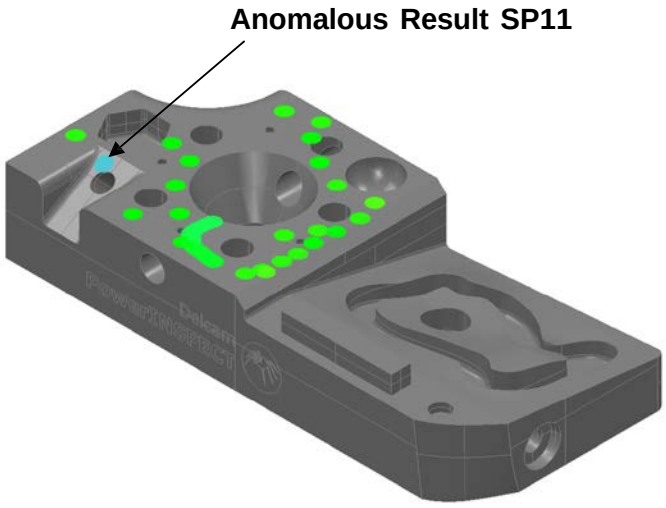
Anomalous Result Example

It is possible that when probing a point, an error can be made (e.g. probing too hard, taking too many points, points in the wrong place, false triggering etc.) that creates an unwanted or anomalous result. These results can affect, for example, a line of best fit or an inspection report, so it is often best practice to delete them.

In this example the **DemoBlock2008** part was probed and during its **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, hence this should be removed from the group.

*Note: The **Inspection Group** is **Open** (below left) so as to see the points.*



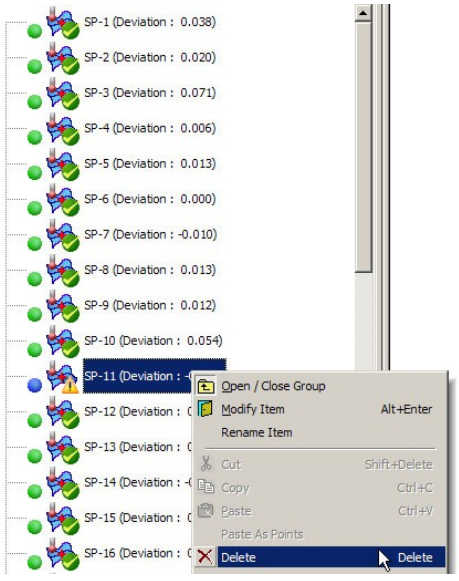


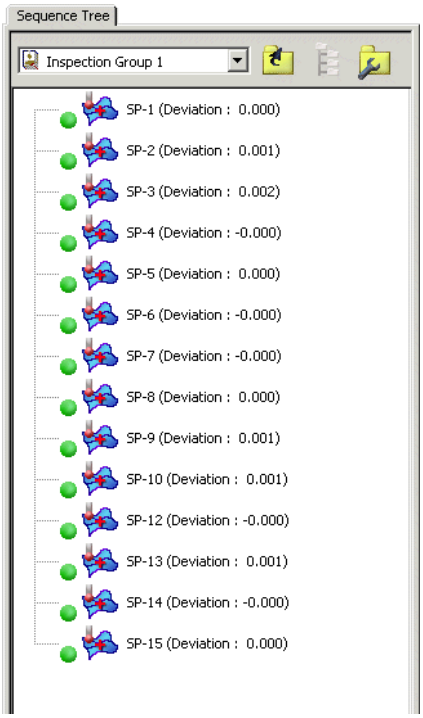
Assuming an anomalous result exists:

- Open the affected **Group**  .

- Right-click on the **Anomalous** result (in this case **SP11**) and choose the delete option from the local menu (see right).

The point will then be removed from the screen, and the **Inspection Group**.





However this leaves a gap in the point numbers, which need to be reordered (left).

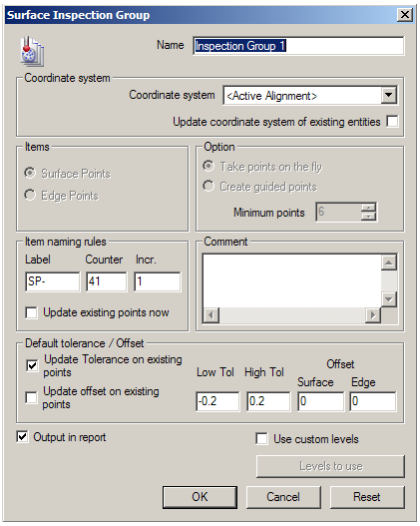
- Move Up One Level  in the **Sequence Tree**.

The numbering of the **Surface Inspection Group** can now be modified.

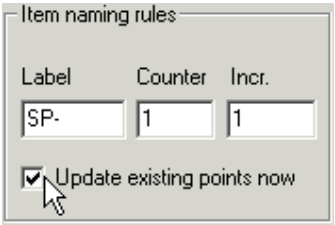
- Highlight **Inspection Group 1** and choose the **Modify Button**  .

Choosing modify brings up the **Surface Inspection Group** dialogue box and allows the user 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**:

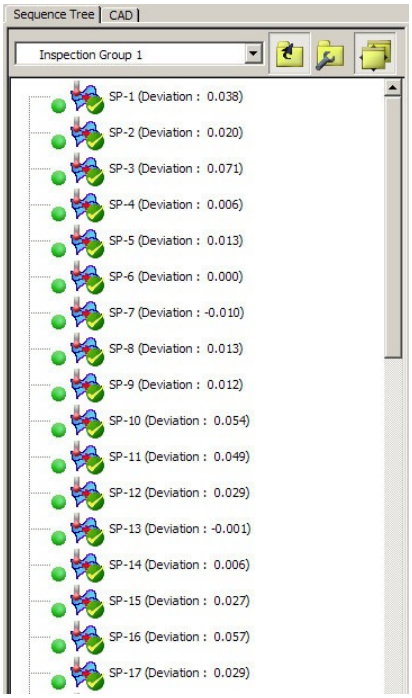


- Enter a **Counter Value** of **1**.
- Enter an **Incr.** (**Increment**) **Value** of **1**.
- Tick the box marked Update existing points now.
- Choose **OK**.



The **Points** will then be updated in the **Sequence Tree** (see right).

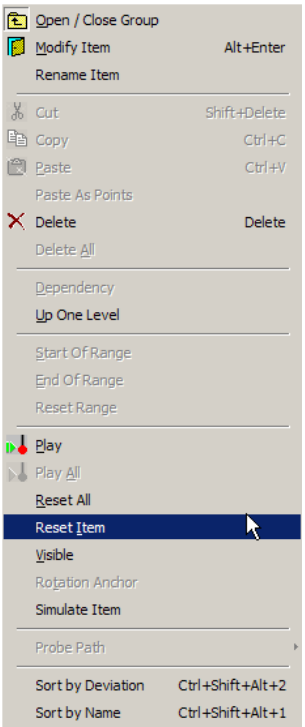
Note that point numbers **SP11** and **SP12** have been updated to become **SP10** and **SP11** respectively.



Replaying Probed Items.

Individual Features and Points can be replayed. For example, if a confirmation is required.

- Right-click on **SP10** and choose the **Reset Item** option from the local menu (see right).



- The point will be reset and indicated in the sequence tree as 

*Note: Selecting **Reset All** will reset **all** the measures in the Group.*

- Select the point and then **Play Element** 

The point will indicated as a dynamic point in the CAD view 

- Probe the point.

The sequence Tree and report will be updated with the new measure.

8. Multiple Alignments

Introduction to Multiple Alignments

PowerINSPECT allows the ability to use **Multiple Alignments** in a single PowerINSPECT session.

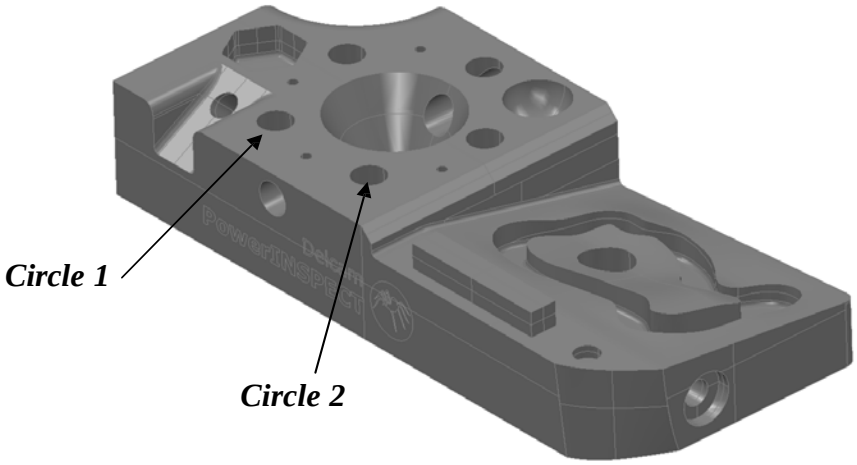
Multiple Alignments can be used in a variety of situations.

- To define a local coordinate system or datum to help in the measurement of features, especially on large and flexible components.
- Provides flexibility to inspect large and flexible components in assembled condition more accurately.
- PowerINSPECT allows you to choose an alignment (that has been created) for each item in the inspection sequence
- If the original location of the component is changed, it now can be realigned without losing the previously measured results.
- Customer requirements can dictate that multiple alignments are used.

Multiple Alignment Example

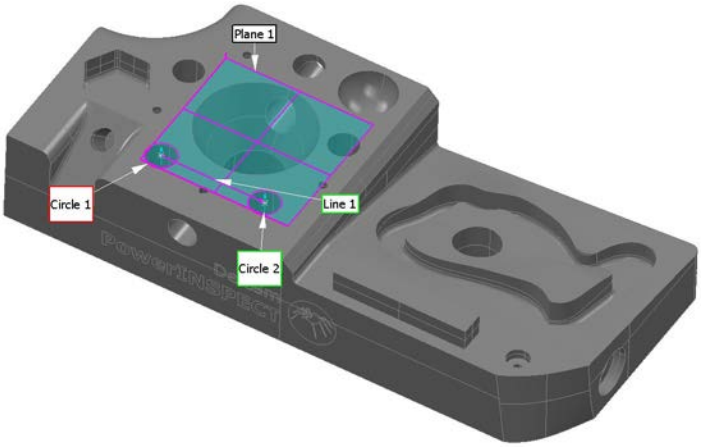
The following Inspection session outlines how alignments can be used in combination. For this example, the DemoBlock2008 file is going to be used

- As described in **Chapter 3.1**, **Create a PLP alignment** using the two holes as before.



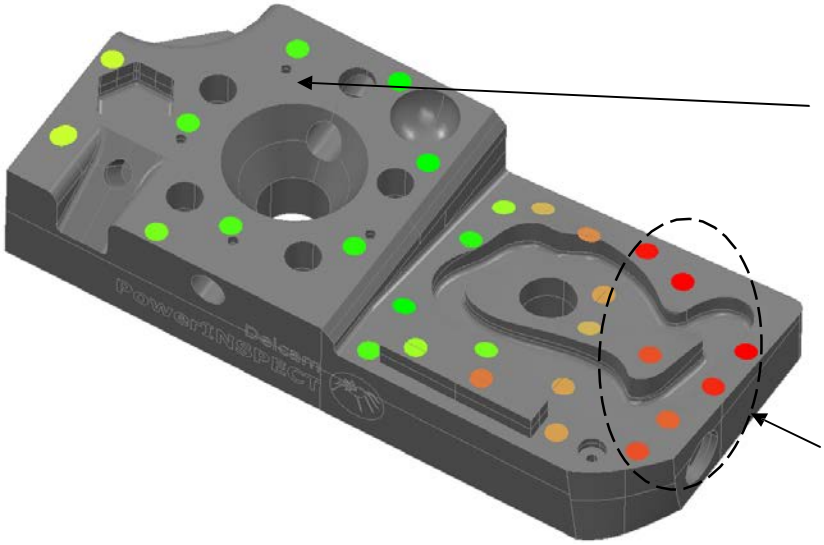
The **Two Circles** chosen are indicated (left), and the **Plane** will be the top face. The **Line** will be between the **Centres** of **Both Circles**, and the **Point** will be taken from the **Centre** of **Circle 1**.
Both circles and line will therefore be projected onto the base (plane)

The Alignment is shown once created.



A **Surface Inspection** will now be created, as described in **chapter 4.1**.

- Create a Surface Inspection Group, by clicking on the Surface Inspection Group Button  on the Element Toolbar.
- Take **Surface Points on the Fly** across the model as shown below.



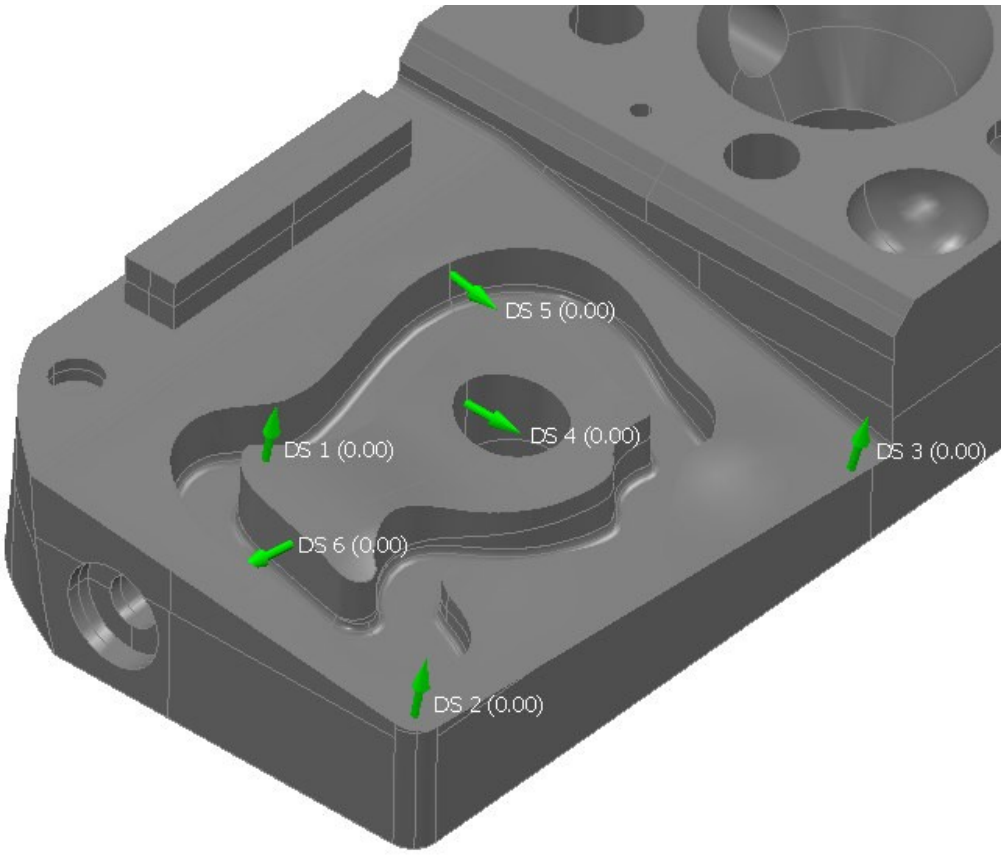
The results show that the surface used for the PLP alignment itself (plane 1) is within tolerance.

However, the 3D surface on the right hand side of the block is correct in the centre but out of tolerance towards the edge of the block.

A second alignment will now be created to check the 3D surface to itself (rather than the first alignment (PLP) which was on a different surface).

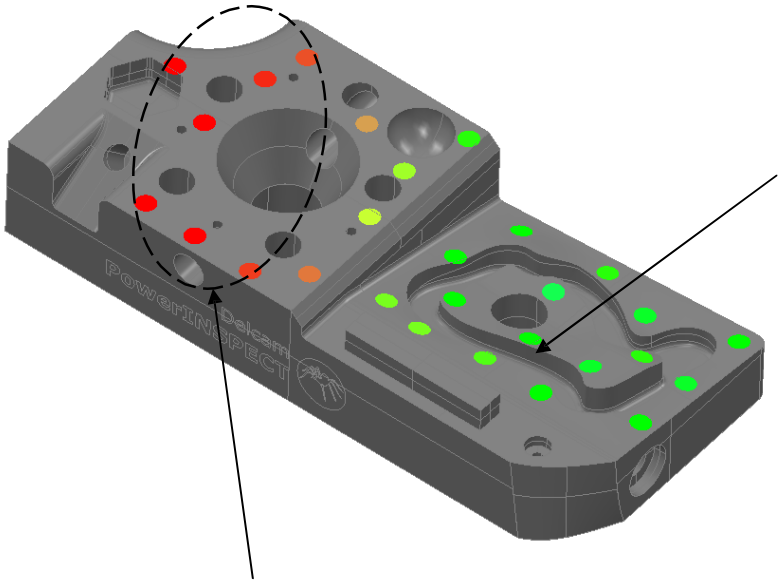
However, due to the free form nature of this surface, the most appropriate alignment would be the **Free Form** method.

- As described in **Chapter 3.2**, **create a Free Form alignment** using dynamic points positioned on the surface as shown below.



- Create a new **Surface Inspection Group**, by clicking on the Surface Inspection Group Button  on the Element Toolbar.

- As before, take **Surface Points on the Fly** across the model as shown below

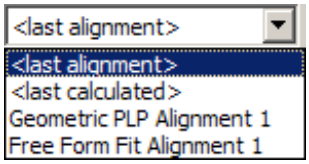


The results show that the 3D surface used for the Free Form alignment itself is now within tolerance.

But the original PLP surface (Plane 1) on the left hand side of the block is correct in the centre but now out of tolerance towards the edge of the block.

The two sets of results indicate a strong possibility of some deformation around the centre of the component.

The active alignment and associated Geometric groups can be changed by selecting it from the main menu.



Transforming Data

Multiple Alignments can also be utilised to measure components that are too large for the CMM limits. A new alignment and measurements can be taken after the component is moved to its new position. The original data (e.g. Alignment and Surface points) can then be transformed to the new alignment position.

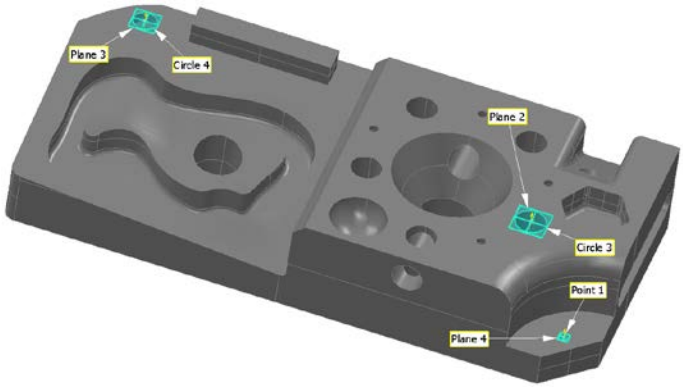
If a portable arm is used, the device can be moved to a new location. A new alignment and measurements can then be taken. However the **Repositioning Wizard** in PowerINSPECT, also allows simple device Repositioning. This technique will be shown later.

To illustrate **Data transformation**, a new third alignment will be added to this Inspection session.

As the component is not large enough, it will be moved to a new location to simulate the effect.

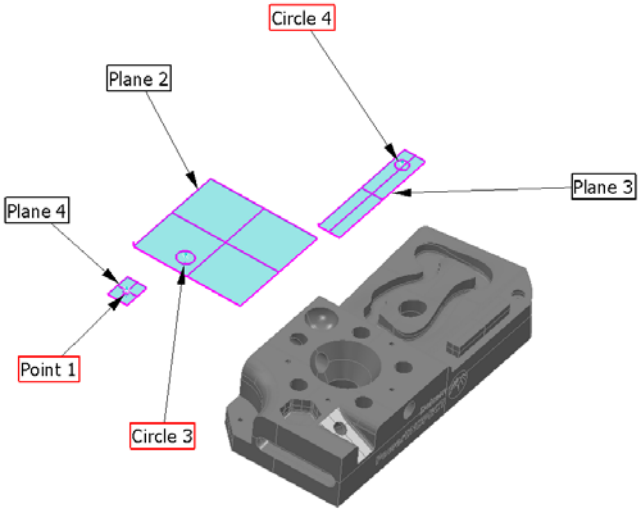
- **Move** the component to a **new location** (but still within range of the device).

- As described in **Chapter 3.4**, a **Best Fit From Points alignment** will be created as the third alignment using the features shown below.



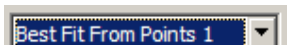
- Select the features and then Play 

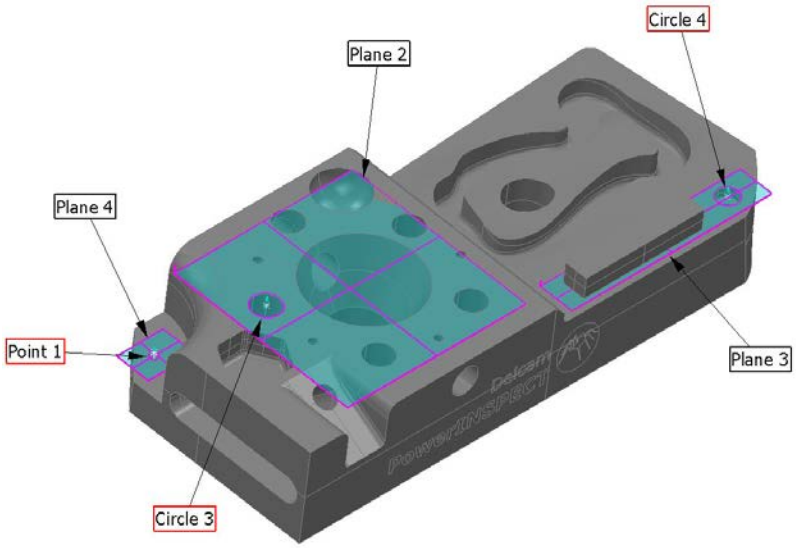
The features are displayed prior to alignment.



- Create a **Best Fit from Points alignment** using **Point 1** and **Circle 3, 4 centre** points.

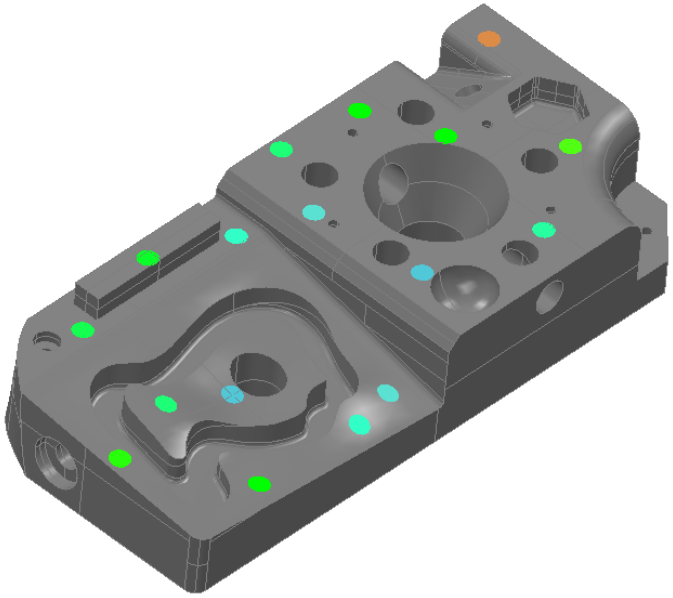
- Ensure the new Alignment is selected as the **active** alignment from the main toolbar.



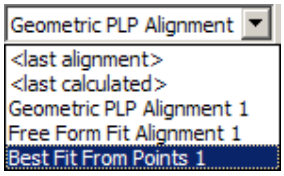


- Create a new **Surface Inspection Group**, by clicking on the Surface Inspection Group Button  on the Element Toolbar.

- As before, take **Surface Points on the Fly** across the model as shown below

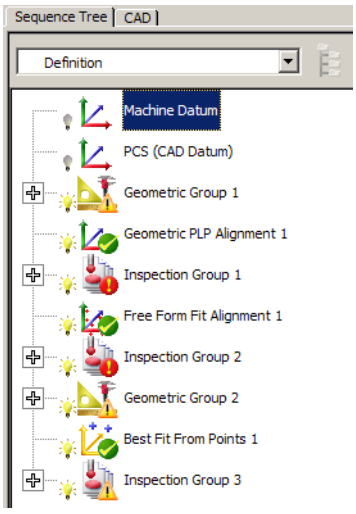


Selecting the other alignments display them in the previous position.

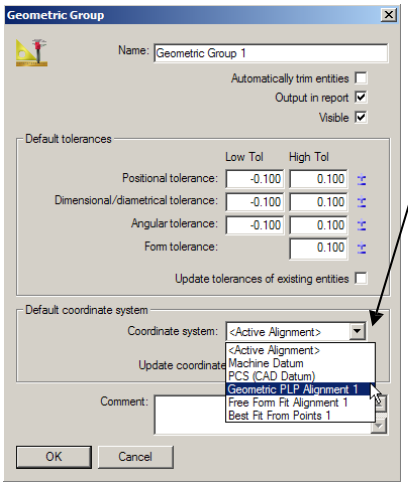


These previous two alignments will now be **transformed** to the **current active alignment**. This will then provide us the ability to cross reference the measured geometry and surface inspections in all alignments. All alignments will also sit on the CAD once transformed.

The sequence tree should be as follows.

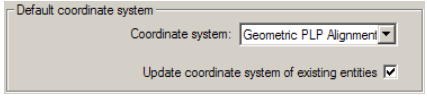


- Select **Geometric Group 1** and then **modify**  (or right mouse button select).



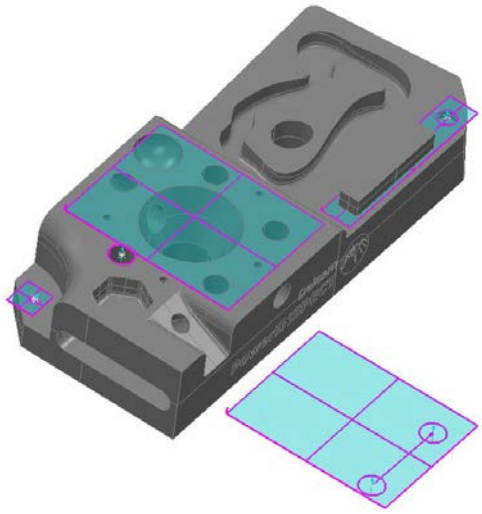
- Select **Geometric PLP Alignment 1** from the Default coordinate system.

- Ensure the option **Update coordinate system of existing entities** is ticked, before selecting **OK.**

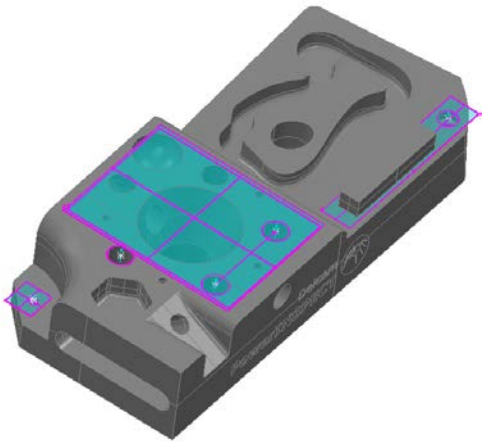


This transforms the coordinate system of the **Geometric Group 1** and its contents (PLP) to the current active alignment (Best Fit from Points).

BEFORE PLP TRANSFORMATION



AFTER PLP TRANSFORMATION



- Select the **Report** tab

Geometric Group 1

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

		Hi-Tol	Lo-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	13.017	**0.015**	-

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

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Centre	X	0.100	-0.100	103.511	103.453	**-0.058**	-
	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	13.008	**0.006**	-

Line 1 (Datum - *Geometric PLP Alignment 1::*)

	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Distance	0.100	-0.100	47.023	46.965	**-0.058**	-

The **Geometric Group 1** features now refer to the transformed Alignment (PLP)

Inspection Group 1

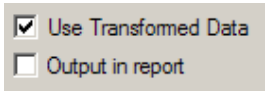
Datum - *Best Fit From Points 1*

Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-35	0.000	-0.200	0.200	50.930	6.852	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-36	0.000	-0.200	0.200	48.331	45.747	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-37	0.000	-0.200	0.200	46.784	80.845	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-38	0.000	-0.200	0.200	68.911	97.442	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-39	0.000	-0.200	0.200	93.195	102.763	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-40	0.000	-0.200	0.200	104.408	76.277	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-41	0.000	-0.200	0.200	121.122	85.119	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-42	0.000	-0.200	0.200	118.962	38.640	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-43	0.000	-0.200	0.200	89.240	10.708	0.000	xxxxx	xxxxx	xxxxx	xxxxx
SP-44	0.000	-0.200	0.200	9.229	31.547	0.000	xxxxx	xxxxx	xxxxx	xxxxx

However the alignment itself and surface inspection data also require transformation.

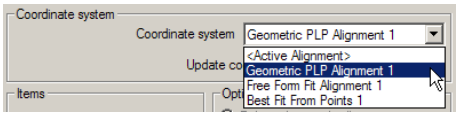
- Select **Geometric PLP Alignment 1** and then **modify**  (or right mouse button select).

- Tick the option Use Transformed Data, before selecting **OK.**

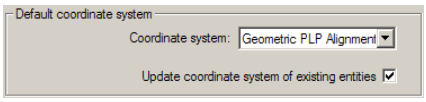


- Select **Inspection Group 1** and then **modify**  (or right mouse button select).

- Select **Geometric PLP Alignment 1** from the Default coordinate system.



- Ensure the option **Update coordinate system of existing entities** is ticked, before selecting **OK**.



The report is now updated correctly.

Inspection Group 1

Datum - *Geometric PLP Alignment 1*

Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-35	0.000	-0.200	0.200	50.930	6.852	0.000	0.000	0.000	0.003	0.003
SP-36	0.000	-0.200	0.200	48.331	45.747	0.000	-0.000	0.000	-0.030	-0.030
SP-37	0.000	-0.200	0.200	46.784	80.845	0.000	0.000	0.000	0.016	0.016
SP-38	0.000	-0.200	0.200	68.911	97.442	0.000	0.000	-0.000	-0.026	-0.026
SP-39	0.000	-0.200	0.200	93.195	102.763	0.000	-0.000	-0.000	-0.035	-0.035
SP-40	0.000	-0.200	0.200	104.408	76.277	0.000	0.000	0.000	-0.039	-0.039
SP-41	0.000	-0.200	0.200	121.122	85.119	0.000	-0.000	0.000	-0.016	-0.016
SP-42	0.000	-0.200	0.200	118.962	38.640	0.000	-0.000	0.000	-0.027	-0.027
SP-43	0.000	-0.200	0.200	89.240	10.708	0.000	-0.000	0.000	-0.013	-0.013
SP-44	0.000	-0.200	0.200	9.229	31.547	0.000	-0.000	-0.000	0.099	0.099

- Repeat the above process for all the items in the sequence tree (Including Best Fit) ensuring all data used to create an alignment is transformed into that alignment.

Sample sections taken from final report.

Inspection Group 1

Datum - *Geometric PLP Alignment 1*

Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-35	0.000	-0.200	0.200	50.930	6.852	0.000	0.000	0.000	0.003	0.003
SP-36	0.000	-0.200	0.200	48.331	45.747	0.000	-0.000	0.000	-0.030	-0.030
SP-37	0.000	-0.200	0.200	46.784	80.845	0.000	-0.000	0.000	0.016	0.016
SP-38	0.000	-0.200	0.200	68.911	97.442	0.000	0.000	-0.000	-0.026	-0.026
SP-39	0.000	-0.200	0.200	93.195	102.763	0.000	-0.000	-0.000	-0.035	-0.035
SP-40	0.000	-0.200	0.200	104.408	76.277	0.000	-0.000	0.000	-0.039	-0.039
SP-41	0.000	-0.200	0.200	121.122	85.119	0.000	-0.000	0.000	-0.016	-0.016
SP-42	0.000	-0.200	0.200	118.962	38.640	0.000	-0.000	0.000	-0.027	-0.027
SP-43	0.000	-0.200	0.200	89.240	10.708	0.000	-0.000	0.000	-0.013	-0.013
SP-44	0.000	-0.200	0.200	9.229	31.547	0.000	-0.000	-0.000	0.099	0.099

Inspection Group 2

Datum - *Free Form Fit Alignment 1*

Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-1	0.000	-0.200	0.200	242.027	35.002	-11.251	0.008	0.026	0.283	0.284
SP-2	0.000	-0.200	0.200	242.362	63.281	-14.517	-0.004	-0.007	-0.057	-0.058
SP-3	0.000	-0.200	0.200	242.027	94.110	-18.913	0.006	-0.005	-0.051	-0.051
SP-4	0.000	-0.200	0.200	207.108	102.629	-22.686	0.007	-0.010	-0.123	-0.123
SP-5	0.000	-0.200	0.200	179.724	106.338	-25.147	0.001	-0.003	-0.016	-0.016
SP-6	0.000	-0.200	0.200	155.254	102.018	-25.512	-0.005	0.021	0.054	0.058
SP-7	0.000	-0.200	0.200	137.051	80.957	-22.777	-0.025	0.050	0.243	0.249
SP-8	0.000	-0.200	0.200	132.756	54.915	-17.143	-0.013	0.100	0.435	0.446
SP-9	0.000	-0.200	0.200	166.624	76.131	-20.468	-0.000	-0.005	0.040	0.040

Geometric Group 2

Circle 3 (Datum - *Best Fit From Points 1::*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Centre	X	0.100	-0.100	41.958	42.063	0.105	0.005
	Y	0.100	-0.100	67.361	67.319	-0.042	-
	Z	0.100	-0.100	0.000	0.010	0.010	-
Diameter		0.100	-0.100	13.001	12.960	-0.041	-

Circle 4 (Datum - *Best Fit From Points 1::*)

		Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
Centre	X	0.100	-0.100	225.000	224.800	-0.200	-0.100
	Y	0.100	-0.100	12.000	12.070	0.070	-
	Z	0.100	-0.100	-10.000	-10.005	-0.005	-
Diameter		0.100	-0.100	10.000	10.028	0.028	-

Inspection Group 3

Datum - *Best Fit From Points 1*

Name	Offset	Lo.Tol.	Hi.Tol.	X	Y	Z	dX	dY	dZ	DL
SP-1	0.000	-0.200	0.200	44.155	80.352	0.000	-0.000	-0.000	0.037	0.037
SP-2	0.000	-0.200	0.200	57.063	101.923	0.000	0.000	0.000	0.011	0.011
SP-3	0.000	-0.200	0.200	123.143	91.654	0.000	-0.000	-0.000	-0.220	-0.220
SP-4	0.000	-0.200	0.200	112.787	47.915	0.000	0.000	0.000	-0.247	-0.247
SP-5	0.000	-0.200	0.200	122.589	14.245	0.000	0.000	0.000	-0.264	-0.264
SP-6	0.000	-0.200	0.200	61.934	9.807	0.000	0.000	0.000	-0.112	-0.112
SP-7	0.000	-0.200	0.200	152.491	9.892	-0.000	0.000	0.000	-0.219	-0.219
SP-8	0.000	-0.200	0.200	185.142	10.132	-0.000	0.000	0.000	-0.044	-0.044

9. Device Repositioning

Using the Device Reposition Wizard 

The **repositioning wizard** in PowerINSPECT provides the ability to move the part and/or measuring device without losing the alignment. This allows the user to inspect a part that is larger than the device's measuring envelope.

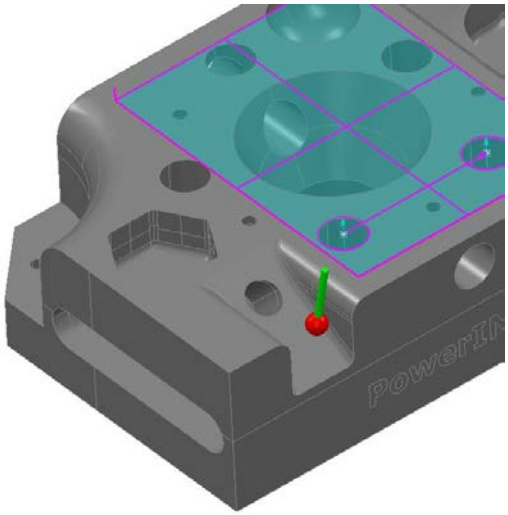
The wizard works by creating a new repositioning datum by specifying at least **three elements.** These elements can be a combination of **spheres and single points**. The part or device can then be relocated and the same elements measured again. PowerINSPECT matches the two positions and so can relate the new measurements to the original alignment.

This allows the user to carry on inspecting the part using the same co-ordinate system.

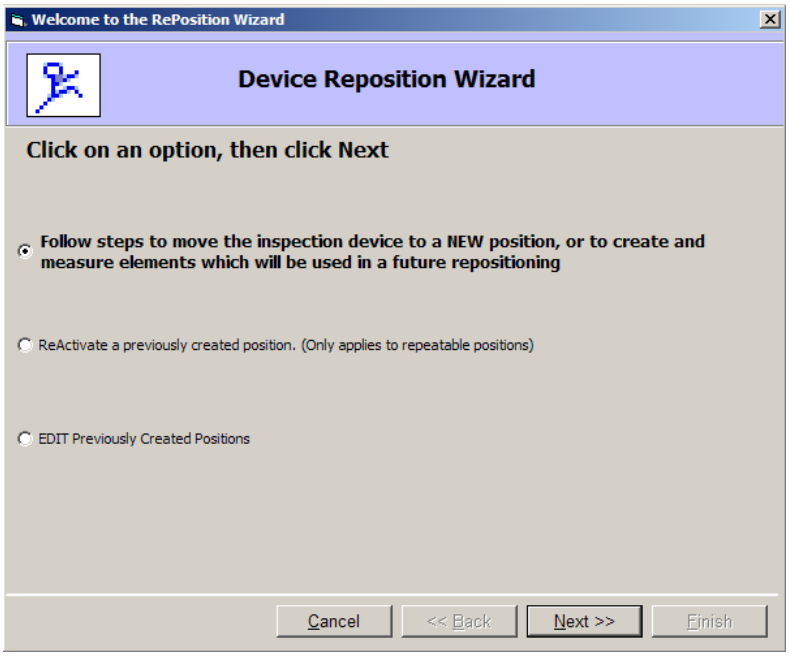
After creating a position, it can be edited and deleted from the session. The position can also be reactivated if you are certain that inspection device is in the exact same position.

- Create a **New Document using the Wizard**  and select 

.
- **Browse** for ***Demoblock2008.dgk***. Open this file.
- Create and confirm the **Alignment** using one of the previously learnt methods in **Chapter 3.**



- From the **Machine Toolbar**  , select **Repositioning Wizard**  .



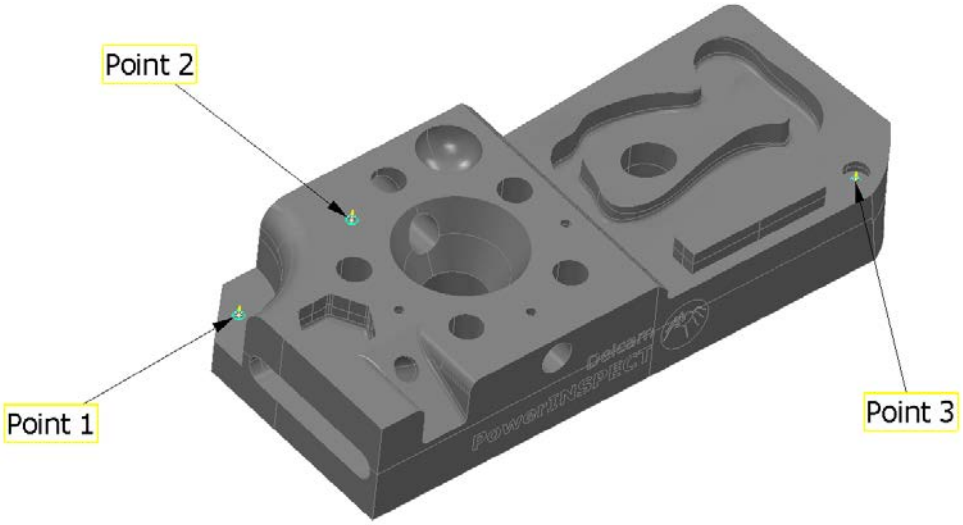
- Ensure the first option **(Create New position)** is selected and then click **Next.**

The elements are selected at this stage (at least three).

 is selected to create a **sphere element**.

 is selected to create a **single point element** (note: hard probe should only be used)

For this example, **three single point elements** will be selected on the **demoblock2008** part, as shown.

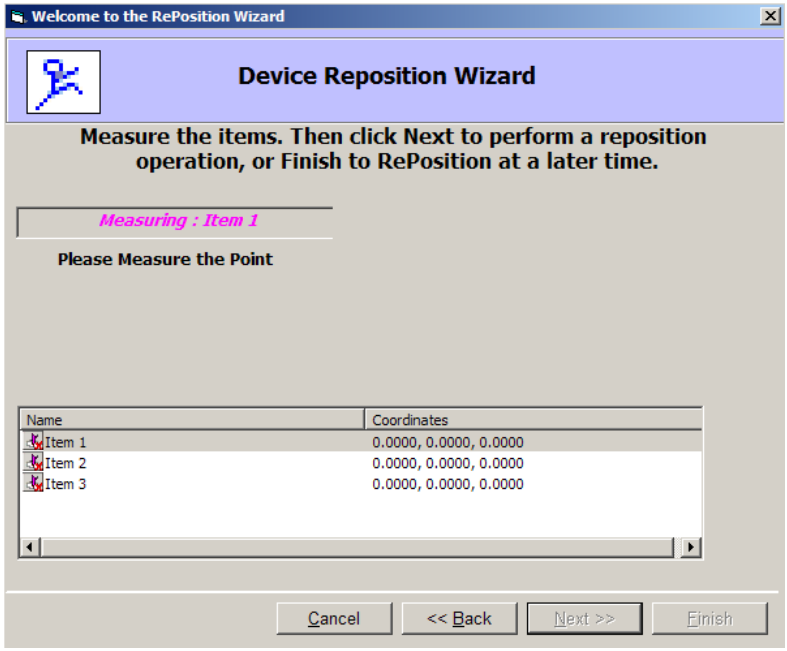


- Select  three times to select three single point elements.

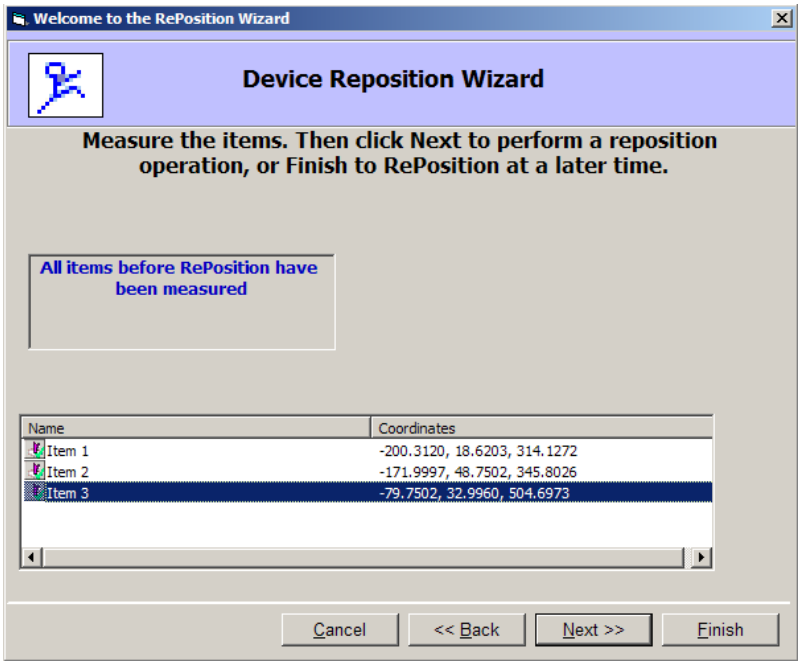
Name	Coordinates
Item 1	0.0000, 0.0000, 0.0000
Item 2	0.0000, 0.0000, 0.0000
Item 3	0.0000, 0.0000, 0.0000

- Select Next.

The three elements now require measurement



- Measure the three point items as indicated above.

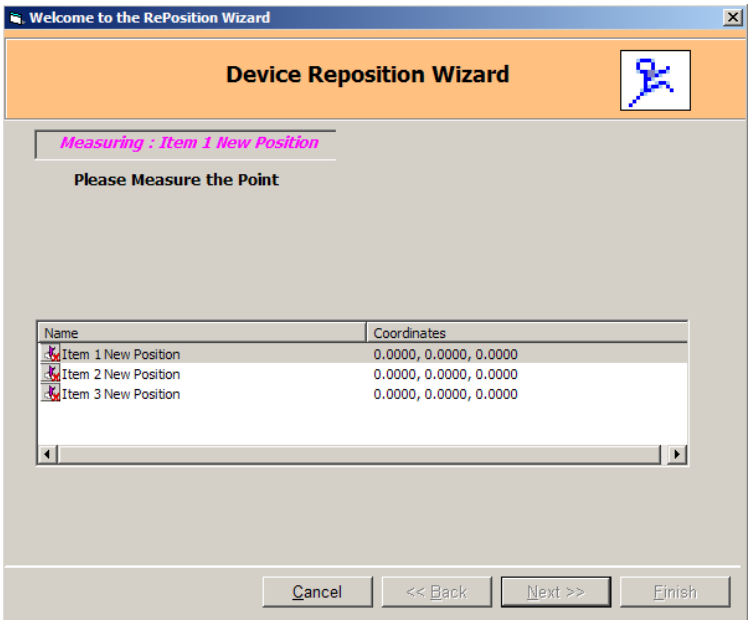


The measured coordinated for the three points are displayed.

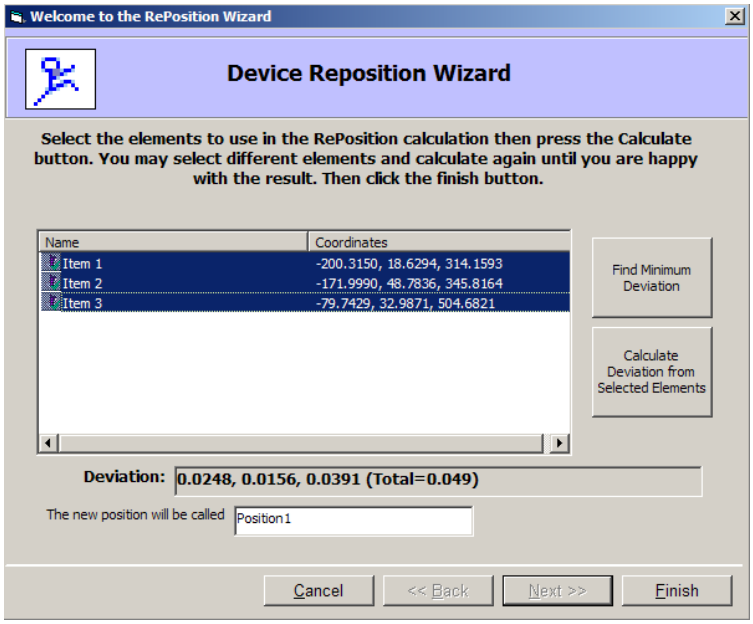
- Select **Next.**

The wizard now instructs the user to move the inspection device to a new location.

- Move the inspection device (or to simulate the same effect, **move the component block to a new location)**.
- Select **Next.**

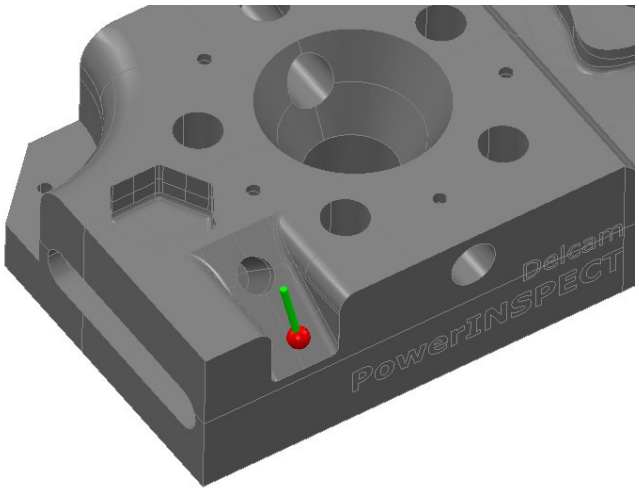


- Measure the **same three points** in the **same order** as initially taken and select **Next.**



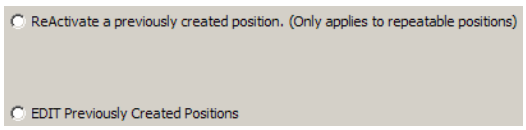
The deviation from the transformation is shown. Calculate Deviation can be used to calculate the deviation from selected items.

- Select **Finish,** to complete the **Device Reposition Wizard.**



Note:
The new position is saved and called **Position 1** (in this case).

Previously saved positions can be **Reactivated** and **Edited,** from the opening page of the Wizard.

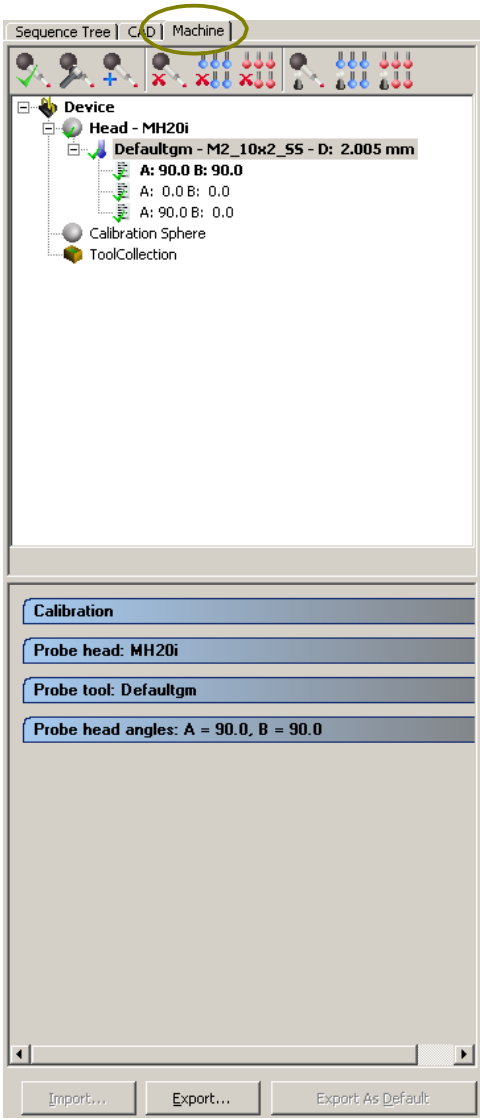


10. Machine Tab (Manual CMM)

Introduction to Machine Tab

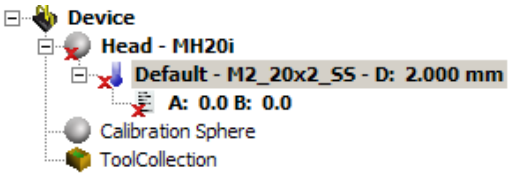
The **Machine Tab** is used to **manage the measuring device** when PowerINSPECT is connected to a **CMM**. *Please note that no information is displayed in the tab if PowerINSPECT is connected to an arm.*

The Machine Tab is available from the document window to enable the user to manage the **probe heads**, **probes** and **calibrations in one place.**



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 will demonstrate the creation of a new probe assembly and calibration.

It should ne noted that the exact procedure will vary from CMM to CMM. You should follow the prompts given on your machine. Ask your tutor if in doubt.

Creating a new probe assembly

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

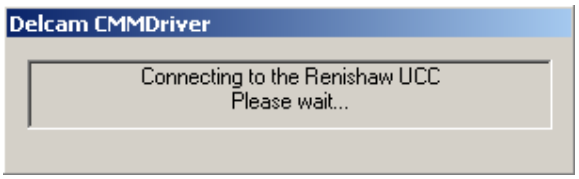
- Select **Create a New Document**. 

A new empty session has loaded

- Select the **Toggle Machine Toolbar**  from the **Main Toolbar**.

- Select the **Connection Button** 

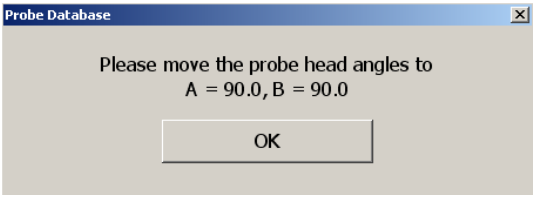
A window is displayed during the connection attempt to the machine.



If successful, 

is displayed in the Machine Toolbar (replacing the **Not Connected**).

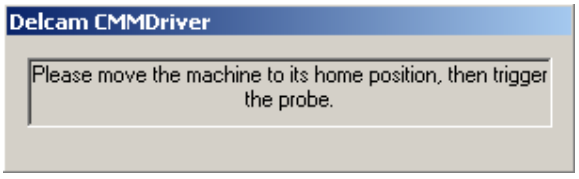
- If the user is prompted to move the probe head, Select OK



The Machine Toolbar now displays the **Not Homed** message.

- Select **Home** 

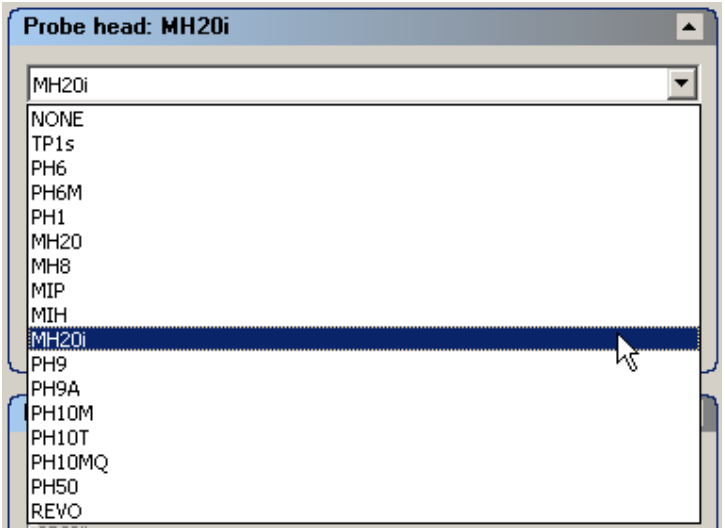
- Move the CMM to the positive end stops (machines do vary) and then trigger the probe as requested.



The following assembly will be created as an example. Where applicable, substitute the components to match your own hardware.

Expand the Probe Head Tab (if not already open) by selected the down arrow  .

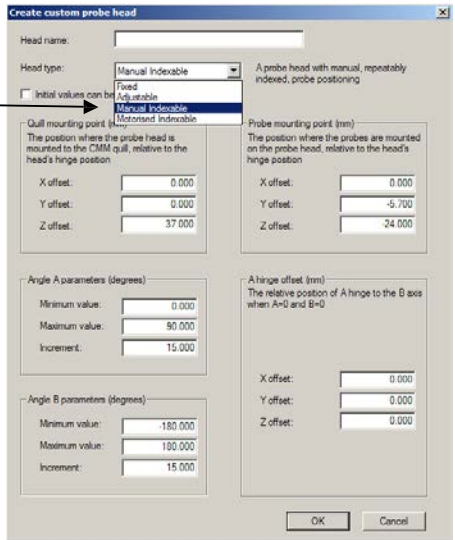
- From the drop down list, select the head **MH20i.**



Note: From this tab, a **new head** can be created by selecting 

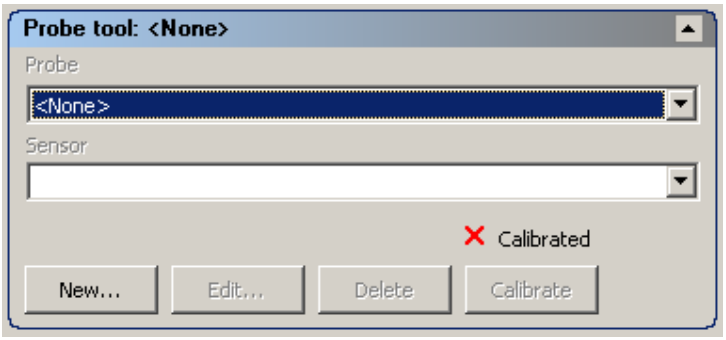
- Select Create to display the form.

Head type can be changed here (e.g. Fixed, Manual Indexable)



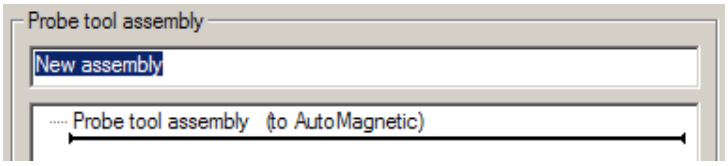
- Select **Cancel** as the previously selected MH20i will be used in this example.

Expand the Probe Tool Tab 

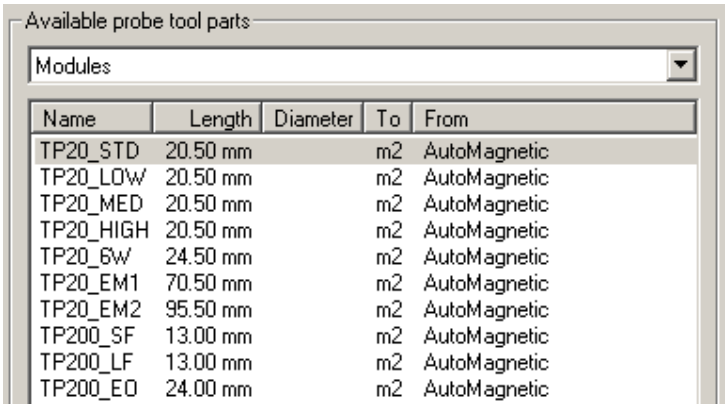


- Select  .

- At the top of the Tab, Enter a suitable name for this assembly (e.g. New Assembly)

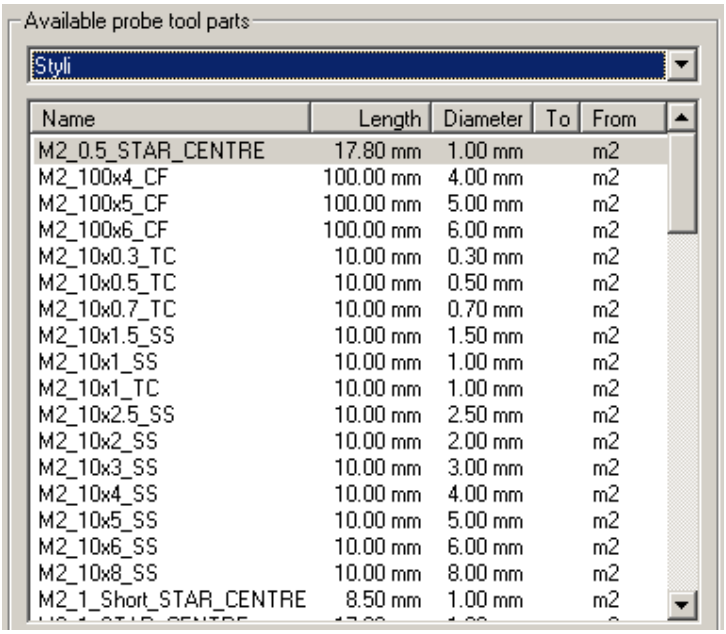


At the bottom the Tab, a list of modules are displayed.



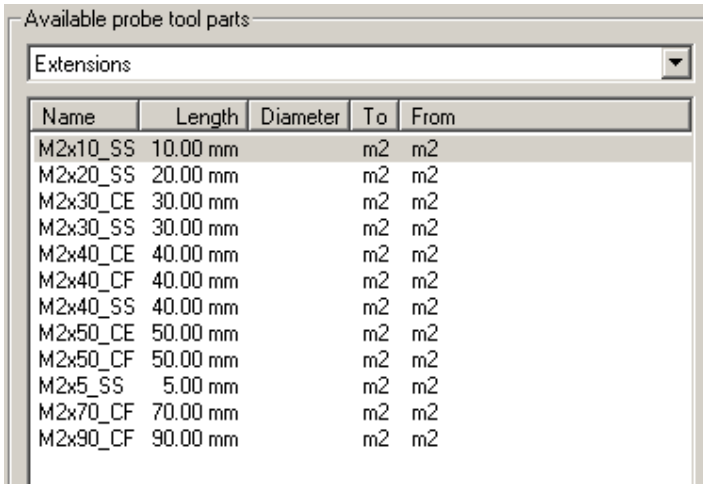
- Highlight the  module.

- Select 

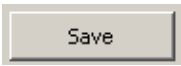


- Highlight the  styli.

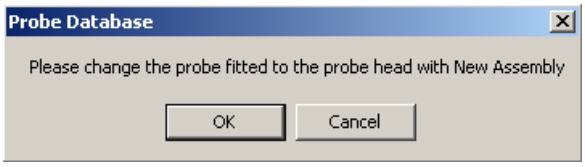
- Select 



The final component to be selected is an extension.

- In this example, no extension will be used, therefore select. 

This message allows the user to ensure the actual assembly/components are assembled.



- Ensure assembly is fitted, and then press OK.

The probe and probe tool that has just been defined will be kept in this **.pwi** file when saved. Therefore, this procedure does not need to be repeated for each new **.pwi** session.

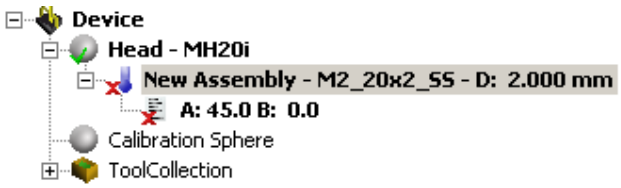
There are three options at the bottom of the Machine Tab.

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.

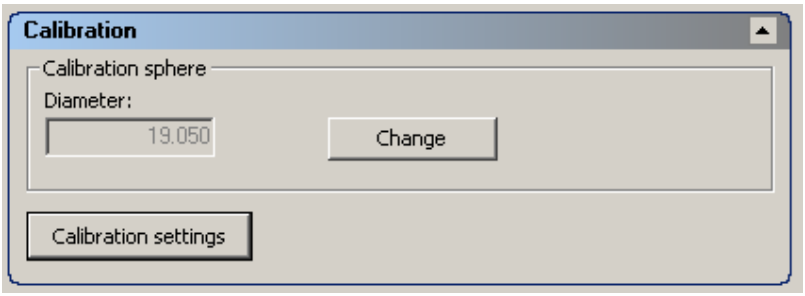
To specify the default settings, click **Export as default** on the **Machine** tab. This saves your settings to a **default.pdb file**, which is automatically loaded each time you create a new inspection session.

The **New Assembly** is now displayed at the top of the **Machine Tab**, summarising the tool setup.



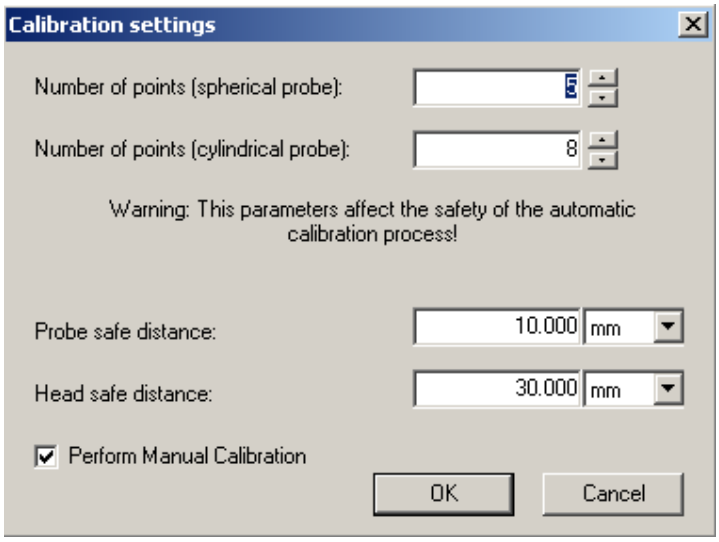
Calibration

Expand the Calibration Tab 



The current diameter of the calibration sphere is displayed here. This can be **changed** but only if the **Head** is calibrated first.

- Select 



The overall settings for the calibration can be changed in this form. Leaving the settings as default, select **OK.**

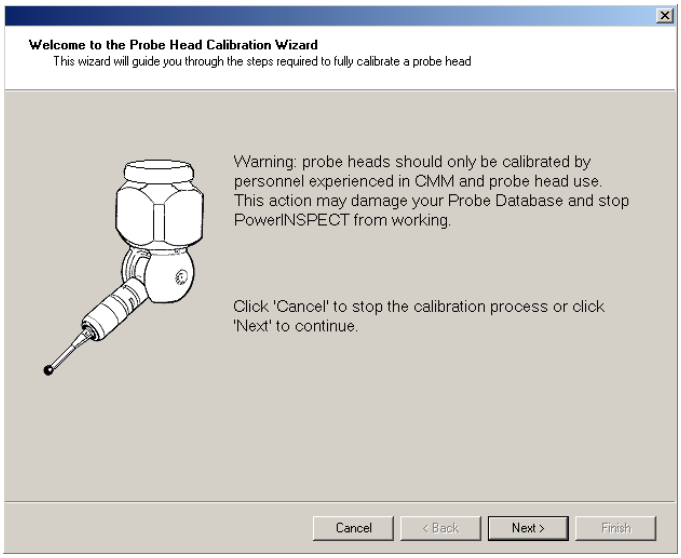
The Head must be calibrated first.

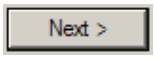
- Highlight the **Head-MH20i** in the assembly tree 

- Select Calibrate selected item 

The **Probe Head calibration Wizard** is lauched. As each stage progresses, the user is prompted with the required action.

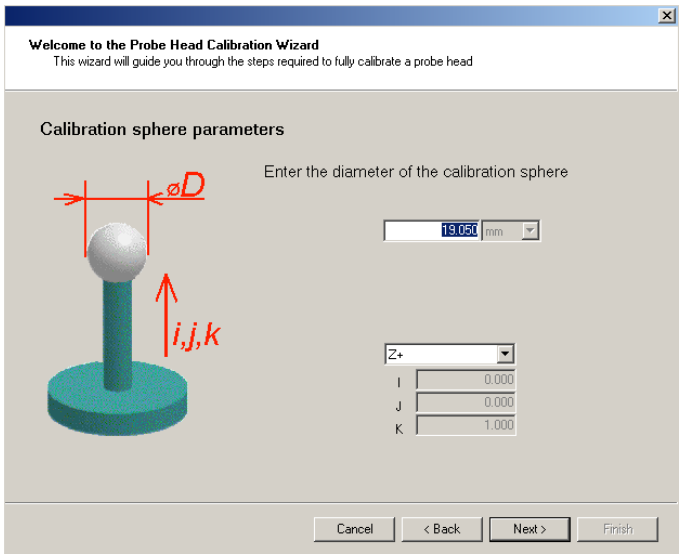
The first stage warns the user that this is a task that should only be undertaken by experienced personnel.

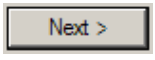


- Select 

The next stage prompts the user to suggest the size of the calibration sphere.

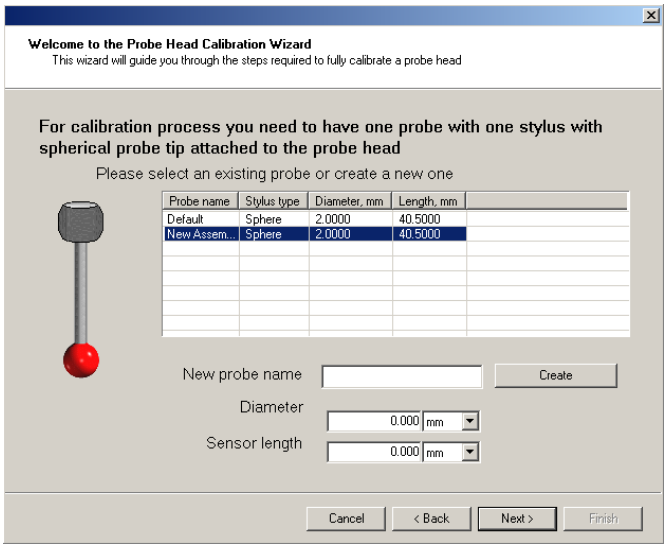
- Enter the size of the **Calibration Sphere**, and the vector direction of it's normal (if asked).

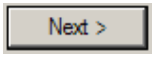


- Select 

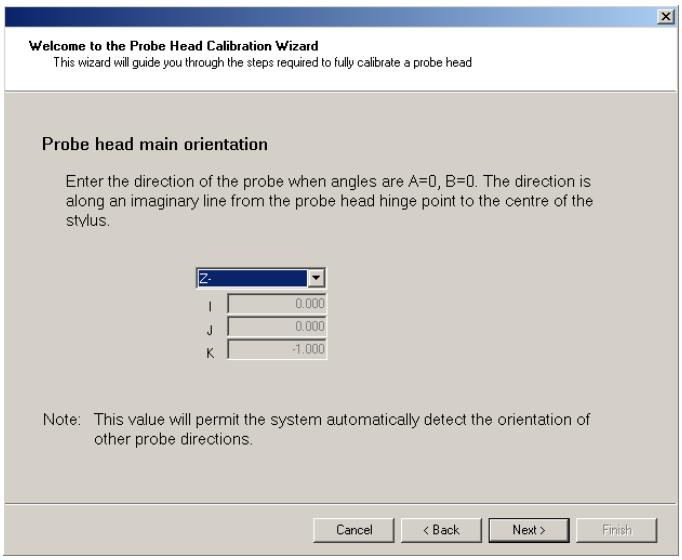
The next stage involves the creation of the **Probe tool** fitted to the **Head**.

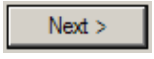
- Select the **New Assembly** (or if required, enter a new name and diameter)



- Select 

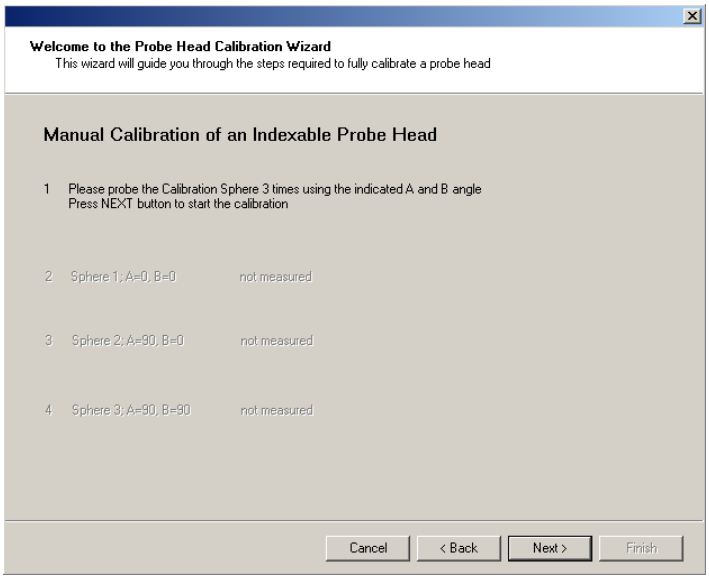
The User is then prompted to enter the direction of the probe when it is at the A= 0, B = 0 position. In this case it is directly down Z



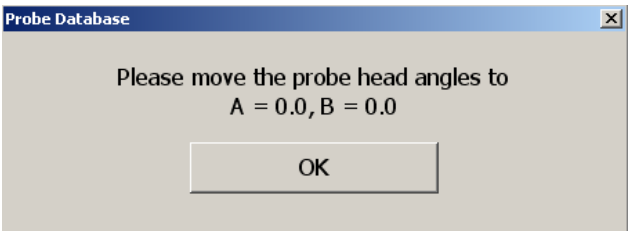
- Select 

The wizard asks the user to probe the calibration sphere.

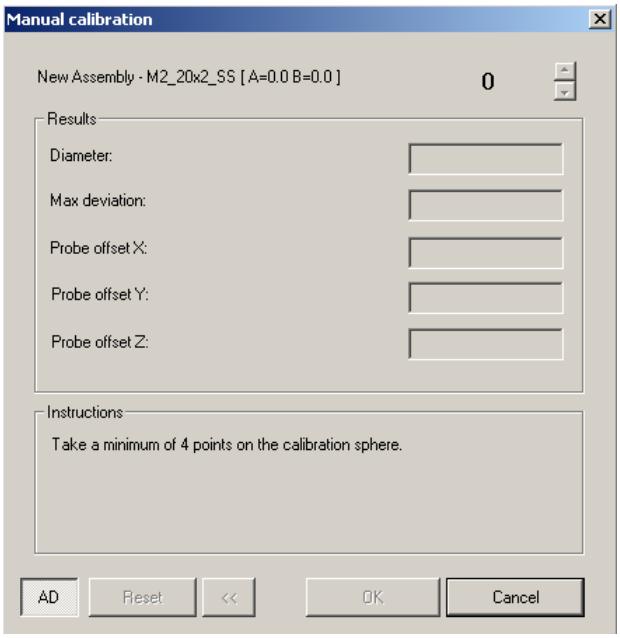
- Choose **Next** to start the process.



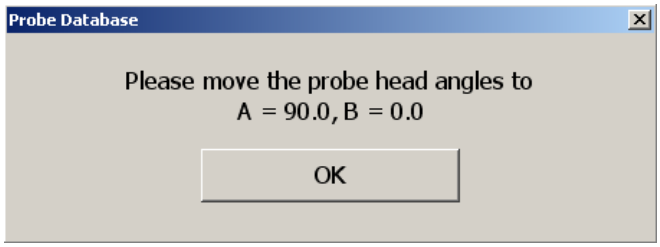
- Change if required, the Head angles to **A=0, B=0**



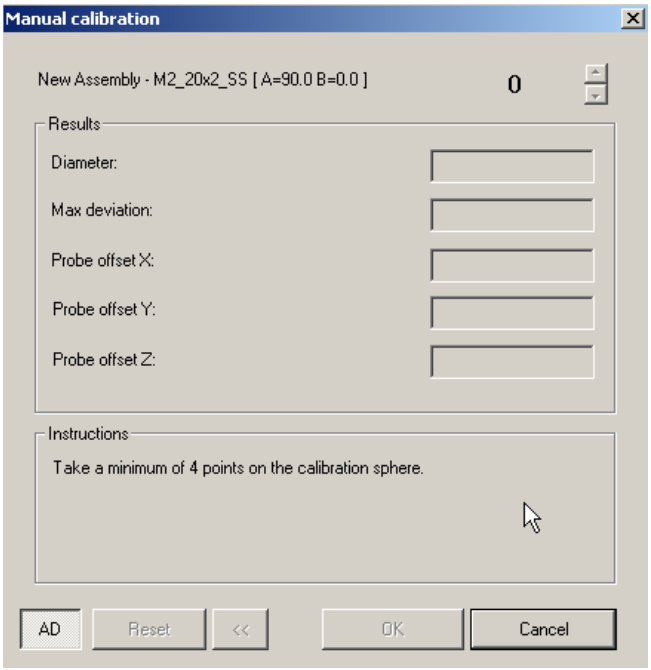
- Select **OK.**



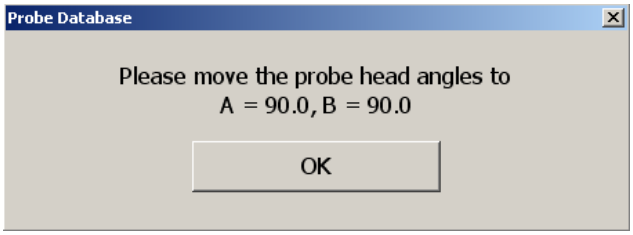
- Take at **least 4 points** around the sphere as shown
- Select **OK.**
- Change the Head angles to **A=90, B=0**.



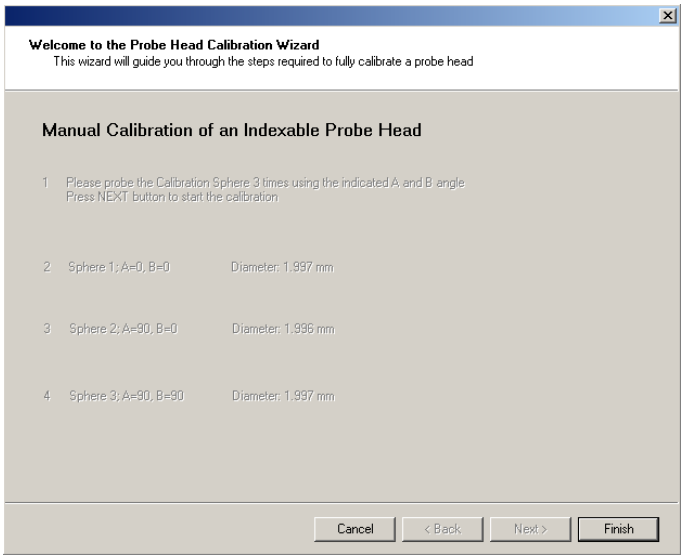
- Select **OK.**
- Take at **least 4 points**



- Select **OK.**
- Change the Head angles to **A=90, B=90**.



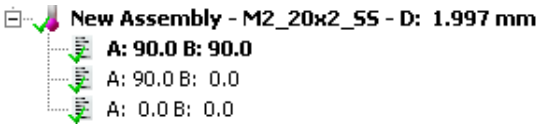
- Select **OK.**
- Take at least 4 points



- Select 

to complete the calibration process.

The assembly tree is now updated to reflect the calibration. The **head** and **three positions** used are calibrated and identified with a green tick.



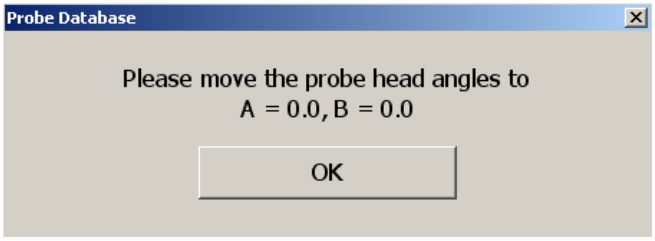
Currently, the position **A:90 B:90** is highlighted in **Bold**. This denotes this **active** position.

If a different position is required then it must be activated first.

- Highlight position 

- Select activate selected item 

- Move the Head to the position requested.
- Select OK

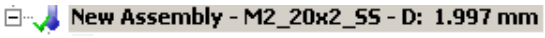


Position A=0, B=0 is now ready to use.

Adding Further probe positions

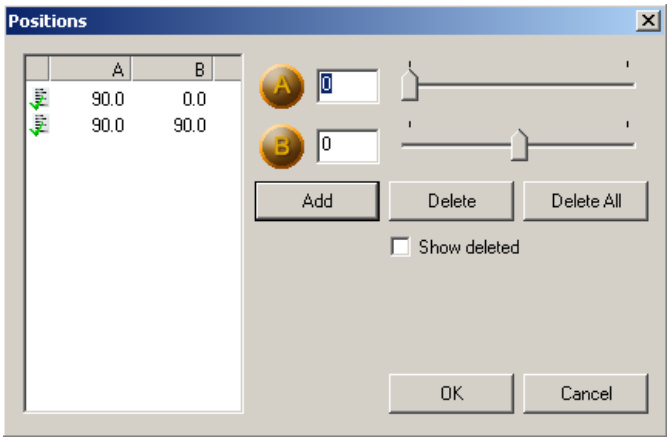
Further probe positions can be added.

- Highlight the **New assembly** in the tree.



- Select **Add new position** 

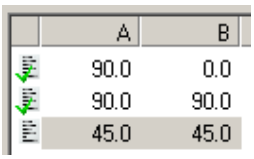
New positions can be added to this form.



Use the slider bars or enter directly the angles required.

- Enter **A=45, B=45**

- Select **Add** to add it to the current list of positions.

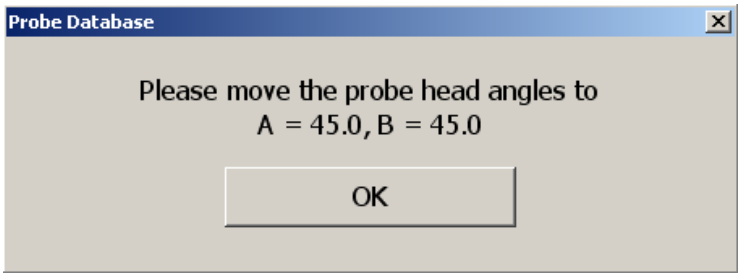


- Select **OK** to close the form

The new position is now listed in the assembly tree. However it is indicated by a red cross identifying it as **uncalibrated**

- **Highlight** the position 

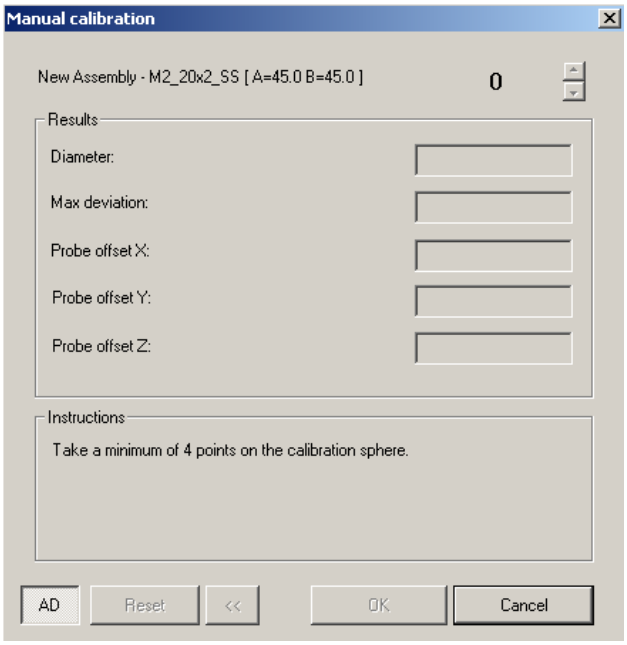
- Select **Calibrate selected item** 



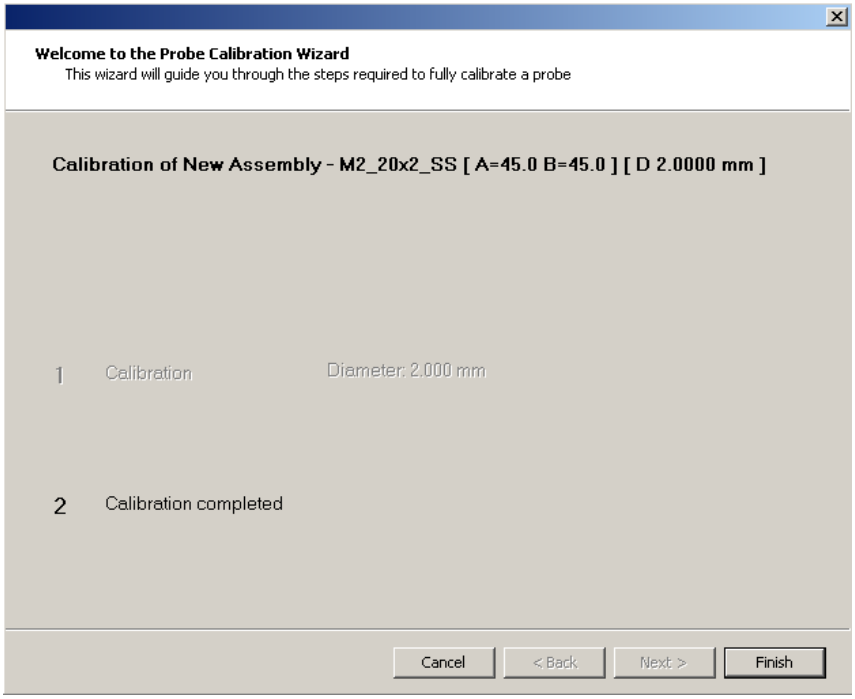
- **Change** the position of the probe to reflect **A=45, B=45**.

- Select **OK.**

- As with the previous calibration, Take at **least 4 points.**



- Select OK



- Select **Finish** to **close** the form

The new position is calibrated and active ready for use. A: 45.0 B: 45.0

11. PowerINSPECT OMV Demo

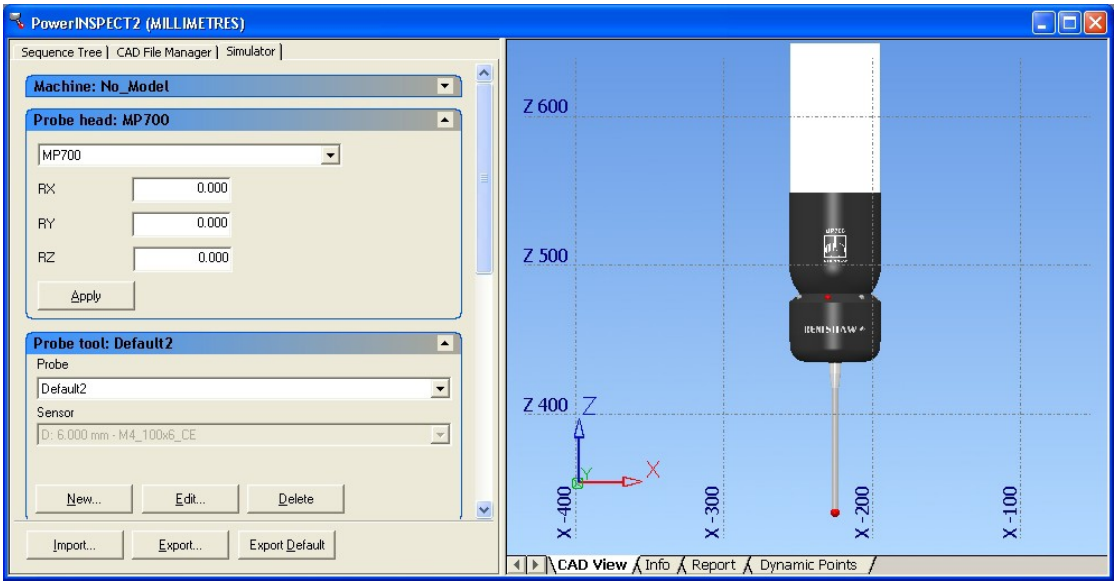
1. Create a new session using the knife.dgk CAD file

The CAD view manipulation is now the same as the other products.

2. Set up the simulator

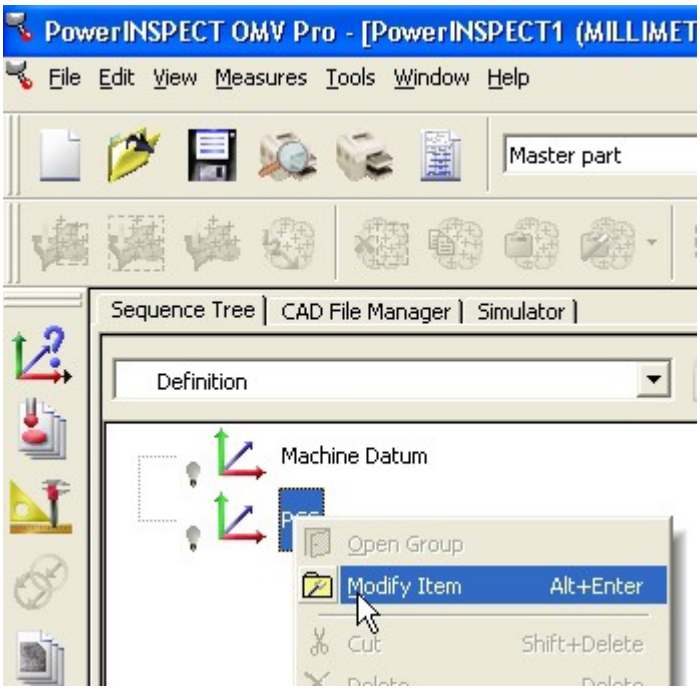
If you have a machine tool already set up, you should use what is available. Otherwise the recommended setup for demonstration purposes is an MP700 with a 100mm long, 6mm diameter stylus.

Set this in the Simulator tab. The simulator is behind the Sequence tree. Set it up to use No_Model mtd file, MP700 and the M4_100x6_CE stylus (this represents M4 connection, 100mm nominal length, 6mm nominal diameter ball, ceramic construction.)



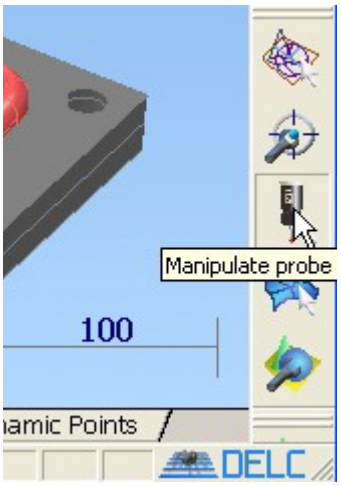
3. Preparation

Reset the PCS



Delete any transformations that have been set up.

Show the CAD view, rotation etc. Move the probe around using the 'Manipulate Probe' button.

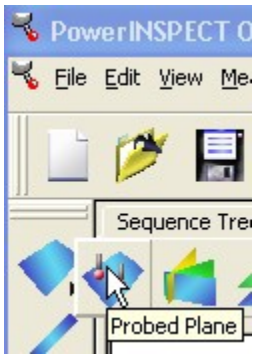


You can move the probe assembly by selecting the stylus with the left mouse button. ALT with left mouse will limit the movement to the CAD principal axes.

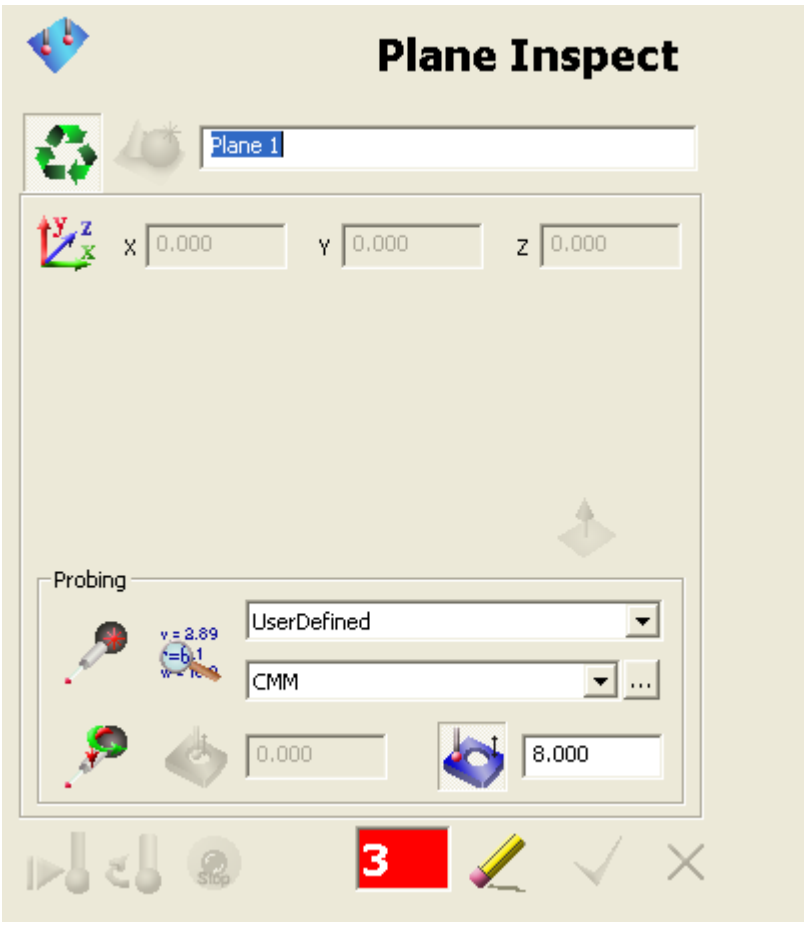
Create a geometric group. 

4. Probed Plane

Create a plane on the top surface using the Probed Plane icon.

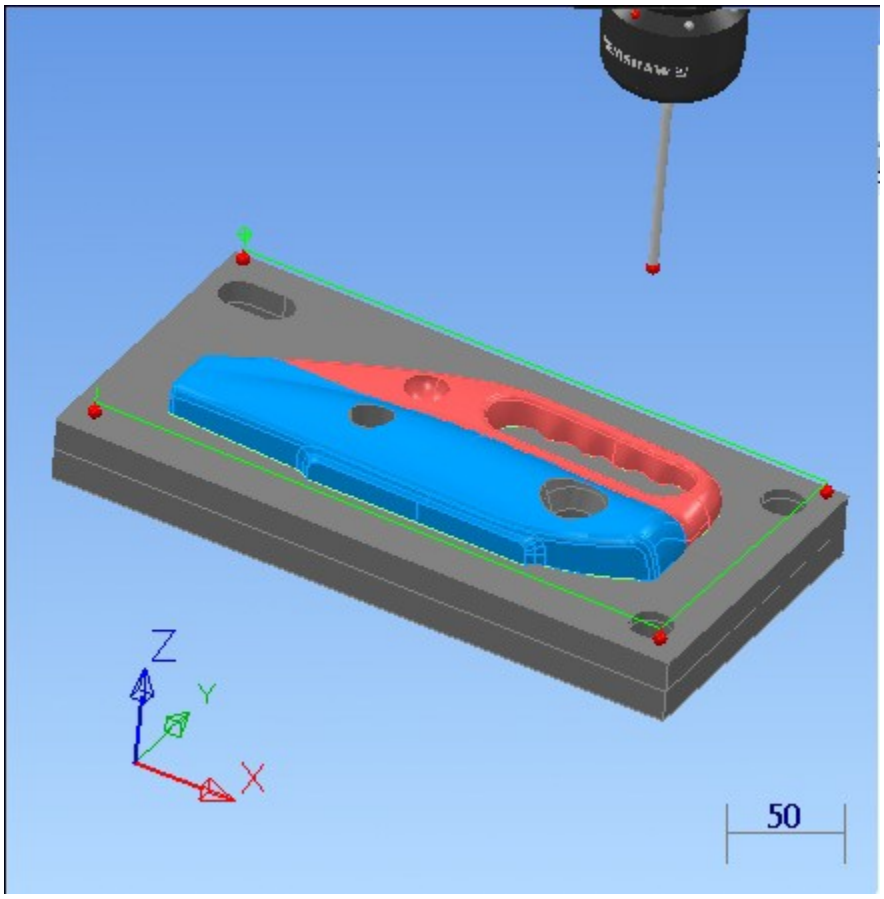


In the CNC panel, this will create a probed plane.



The strategy will be User Defined. This means that the probe path is generated by clicking points on the model.

Click the points on the model around the top plane, and commit the probe path.

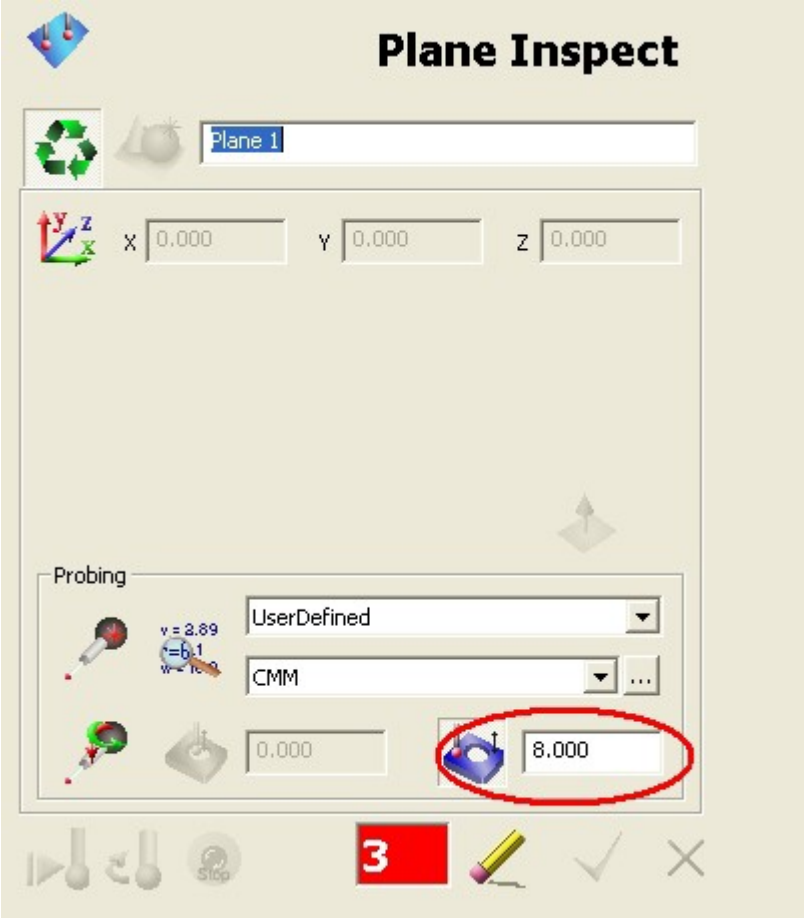


4.1 Editing the Probe Path

When you have committed the probe path, you can edit it by pressing the ‘recycle’ button. 

4.2 Lead in and Lead Out

You can change the Lead in and Lead out using the Safe Plane button.



Pressing the icon  turns the behaviour on and off. The number is the safe plane distance. A value of 40mm will allow you to clear the top of the model.

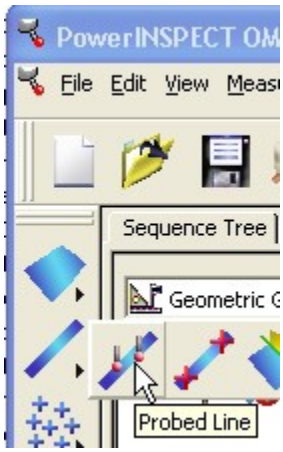
4.3 Simulating the strategy

When the plane is correct, you can show the movement of the probe

using the 'simulate strategy' button.  You can use this in conjunction with the View- Simulator commands, or the keyboard shortcuts F9, F10, F11 which show and hide the machine, the head, and the stylus.

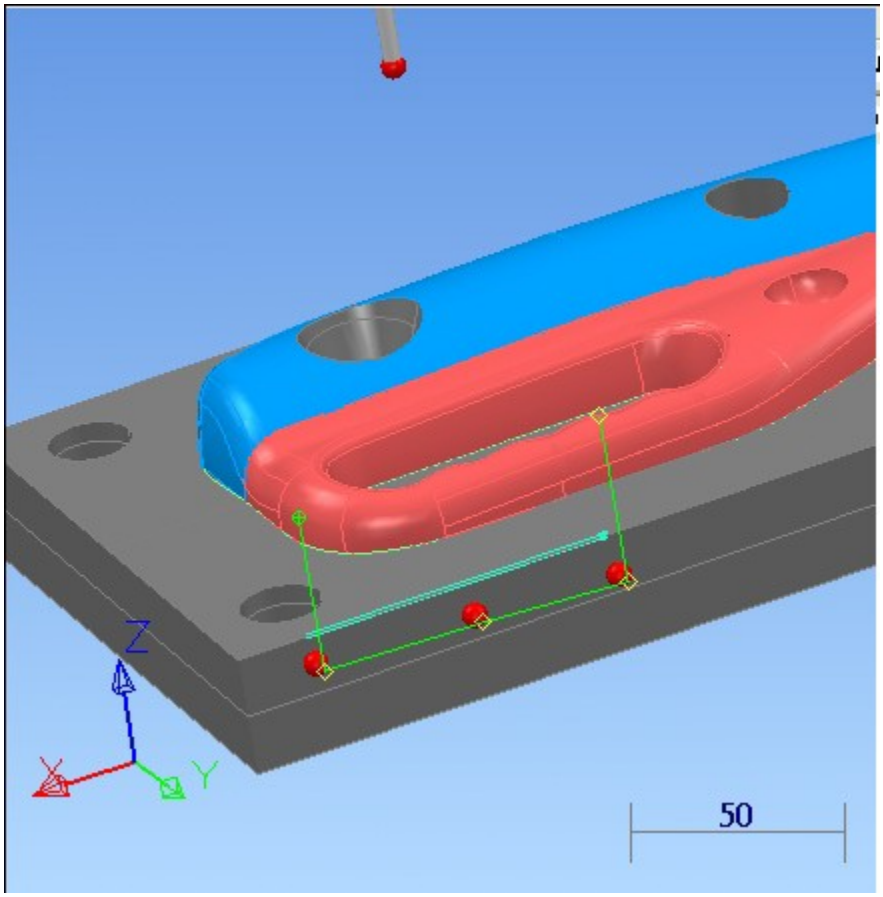
5. Probed Line

Create a probed line using the probed line button.



Again, this will create a feature with a user strategy.

Click points on the edge of the model to create a probed line.



Set a safe plane distance of 40mm again, to ensure that you clear the top of the model.

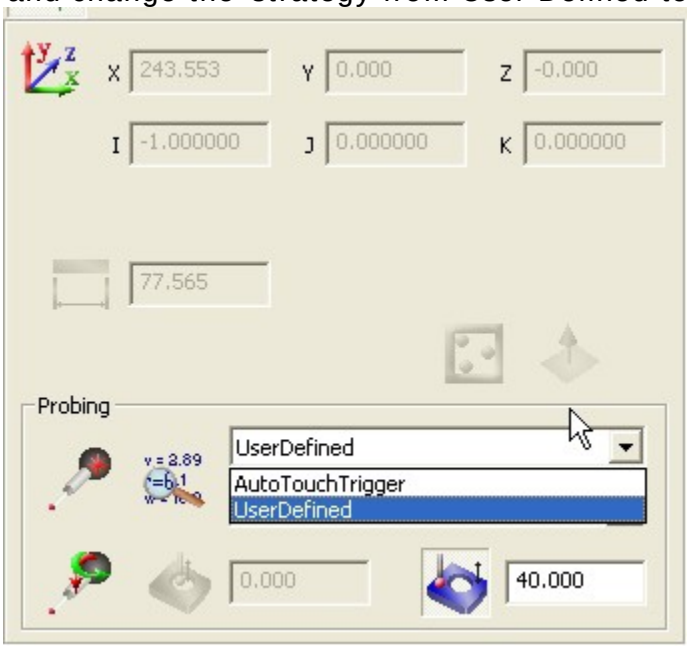
5.1 Converting from a User Strategy to an Auto Strategy

With most geometric features, you can change between a user strategy (clicked points) and an auto strategy (automatically distributed points.)

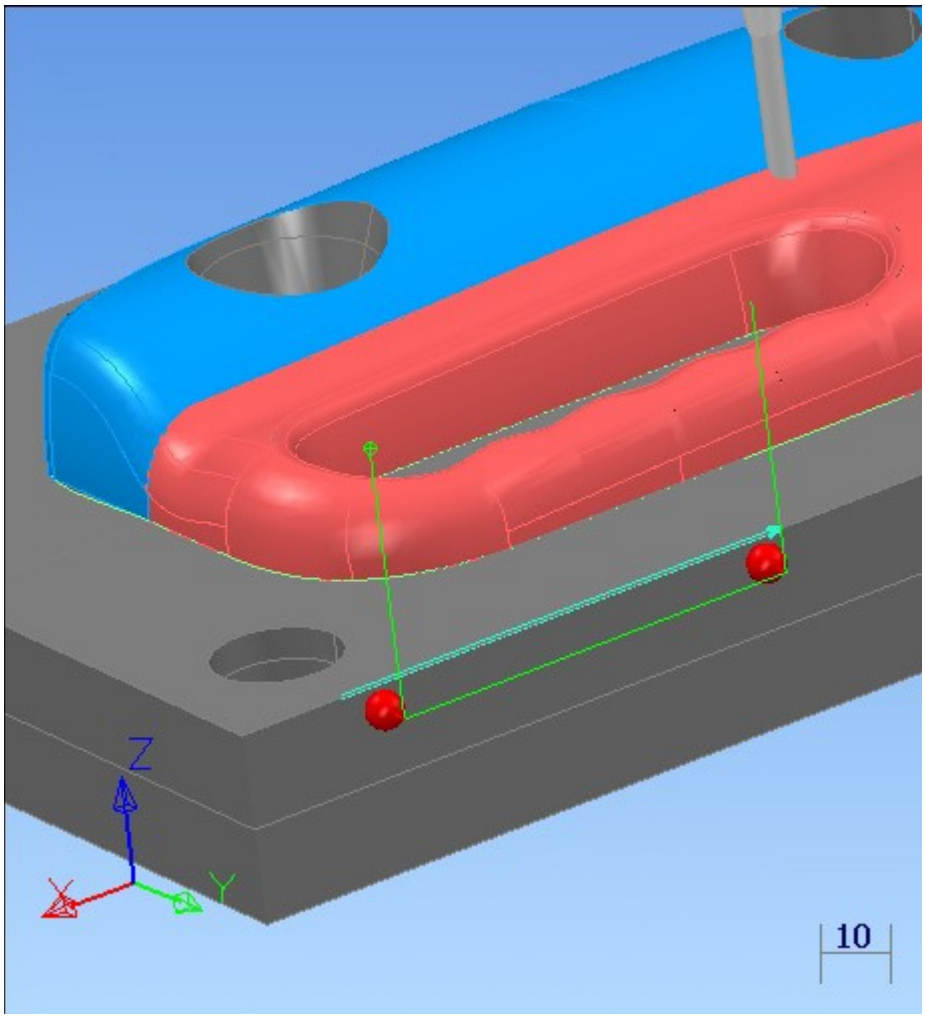
This useful because

- an auto strategy can ‘tidy up’ the probe path from a user strategy
- a user strategy can take an auto strategy as a starting point

To change from the user strategy to the auto strategy, edit the feature and change the strategy from User Defined to Auto Touch Trigger.



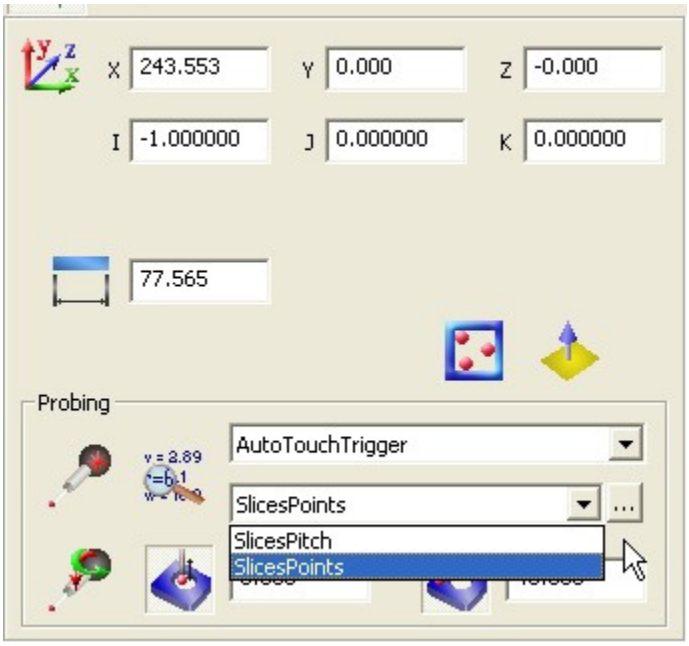
When you do this, the geometry of the feature will be unchanged, but the probe path will be spread evenly along the feature.



5.2 Changing the method

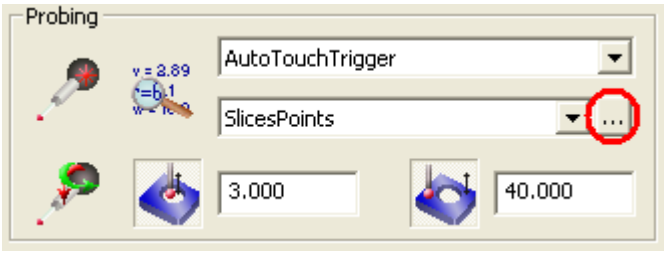
There are two methods for this feature- SlicesPitch and Slices Points

SlicesPoints defines the probe path in terms of the number of points. SlicesPitch defines the probe path in terms of the distance between the points.



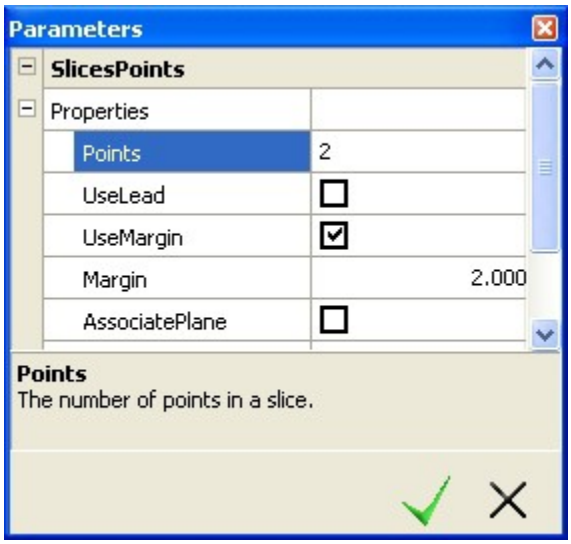
5.3 Changing the number of points

You can change the number of points used in the feature by pressing the strategy editing button.



This will raise a dialog where you can edit the parameters for the probe path, including the number of points.

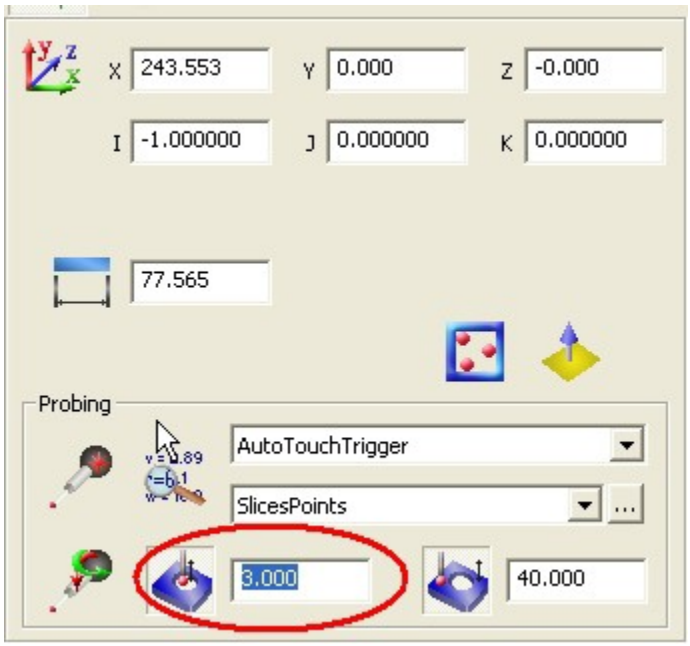
Edit this to the desired number of points.



5.4 Changing the Probing Depth

You can change the probing depth- the depth into the plane- using the probing depth icon. 

Pressing the button toggles this parameter on and off. The number in the field is the depth dimension.

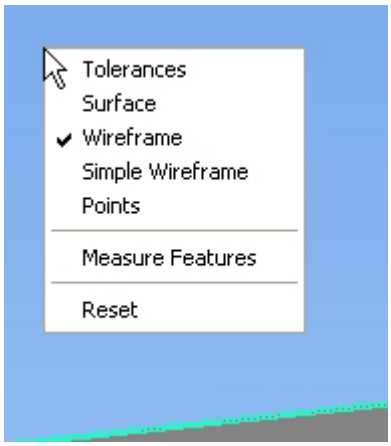


6. Probed Circle

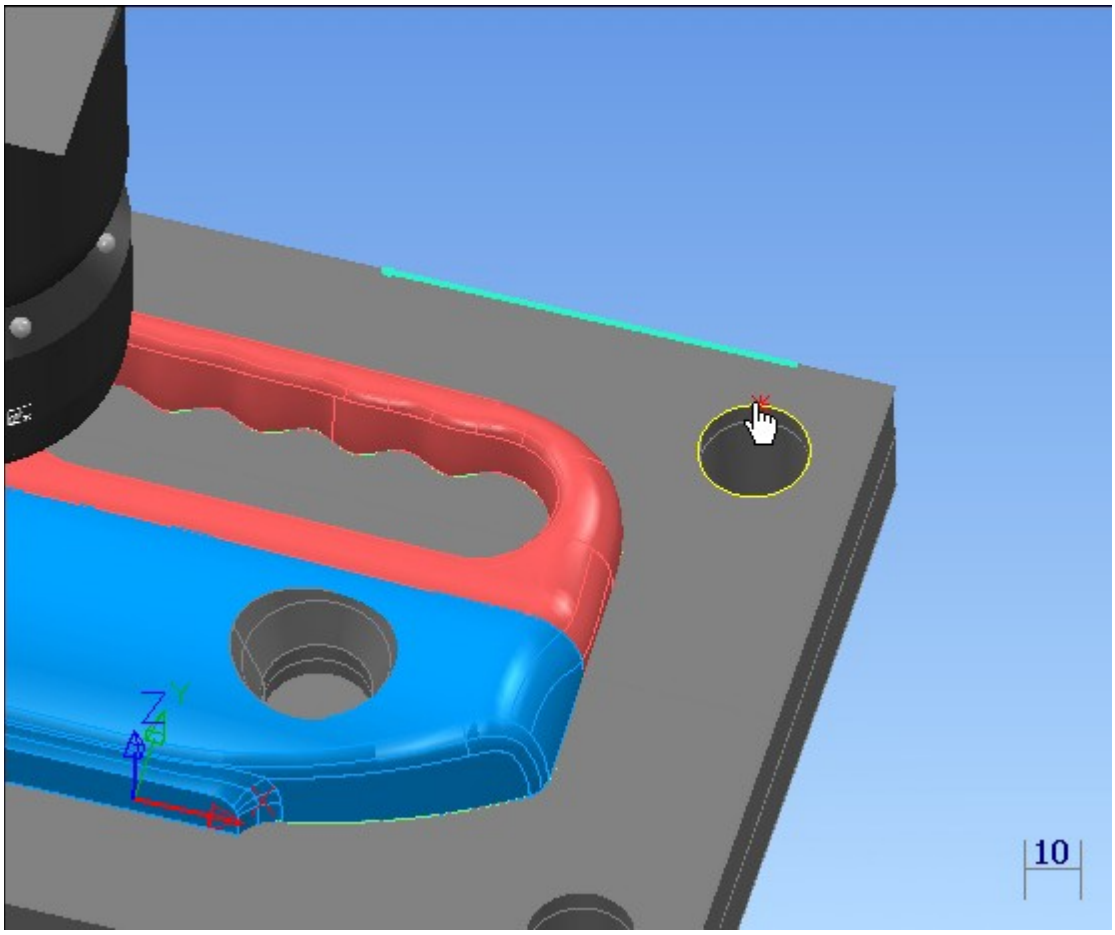
Use the wireframe checker  to select geometry from the CAD model.

By default, the wireframe checker is in ‘wireframe’ mode. This is used to pick complete 2D features such as circles. You can check this by ‘right clicking’ in the CAD view.

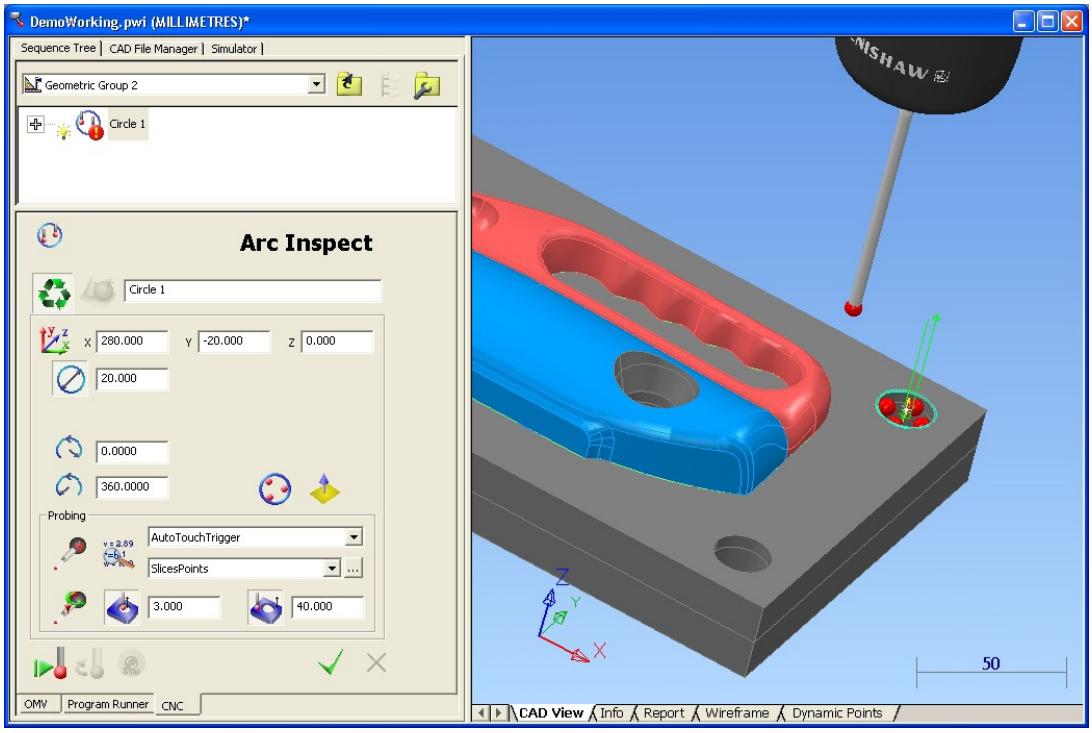
Surface is used to select complete 3D features, such as cylinders. Wireframe is used to select complete 2D features, such as circles.
Simple wireframe is used to select partial 2D features, such as the semicircles at the end of a slot.
Point are used to select single points, which can be built into other features by picking more than one point.



In Wireframe mode, move the cursor over the model and you will see that geometry is highlighted as you more over a particular feature are shown.

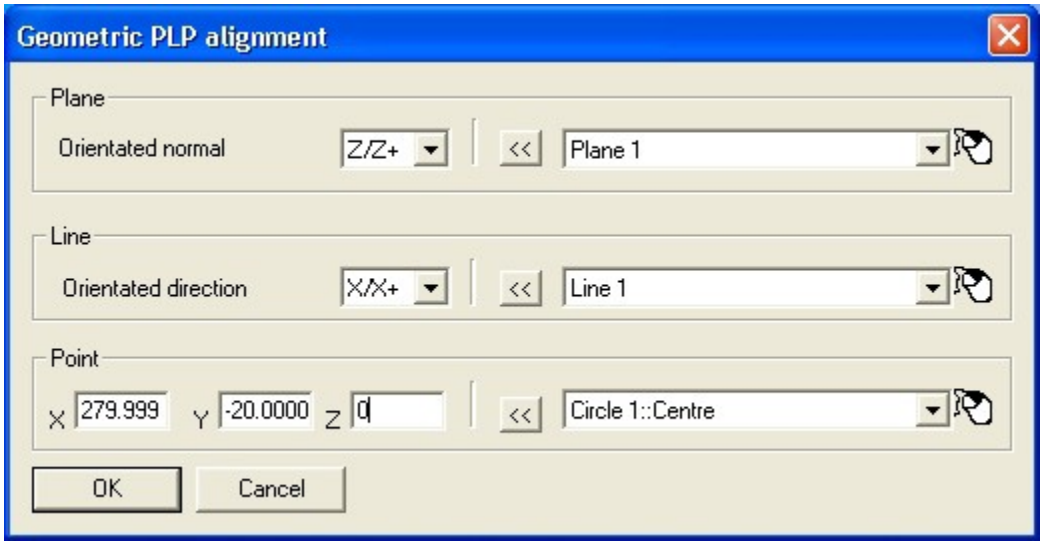


Pick the circle, and PowerINSPECT will create a feature with a probe path. Set the probe safe distance to 40mm, and set the probing depth to 40mm.



7. Creating the Alignment

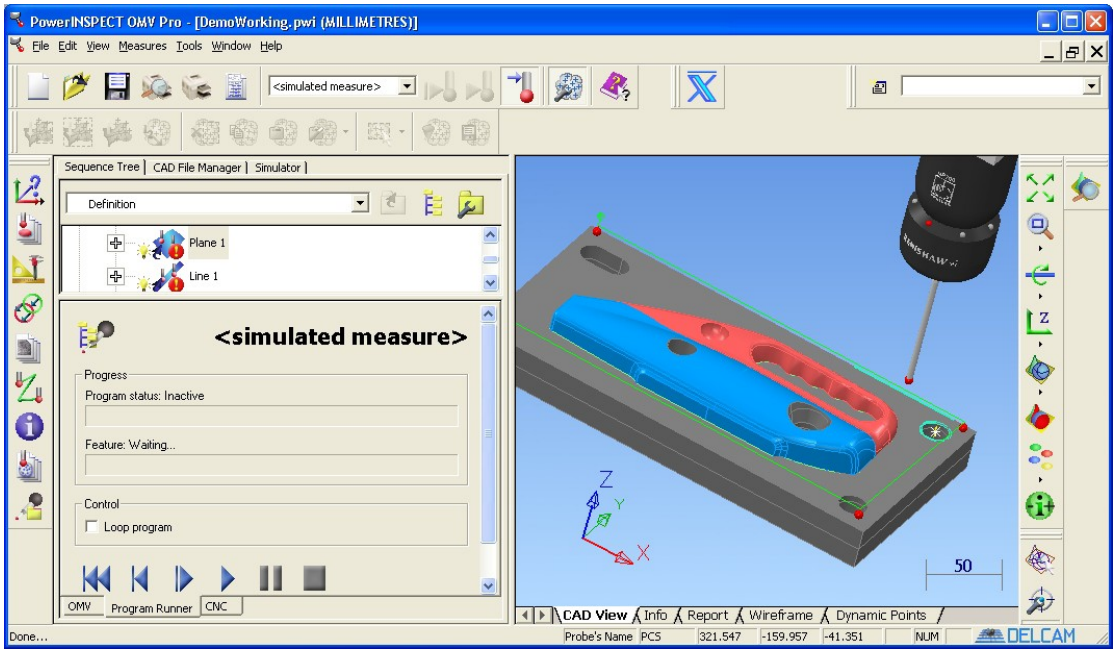
Go up to the definition level, and create a PLP alignment based on these three features.



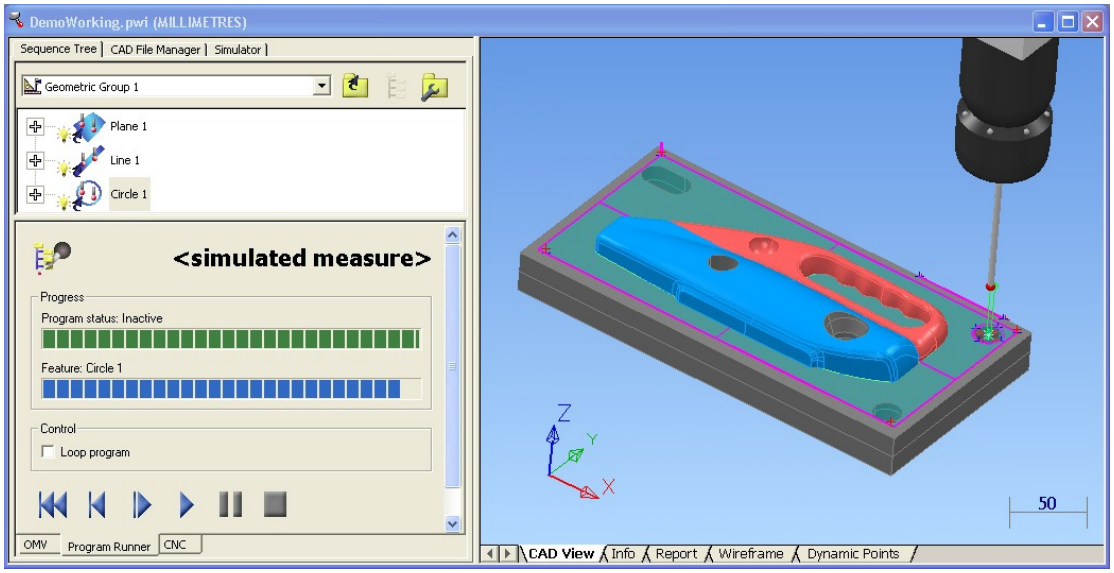
The plane sets the Z orientation. The line defines the X axis.
The centre of the circle defines a point for positioning.

8. Simulating the Measurement

You can simulate the measurement of a group of features- and see simulated results- by using the Program Runner with the Simulated Measure. The play, rewind and stop buttons etc allow you to see the movement of the probe throughout the sequence.

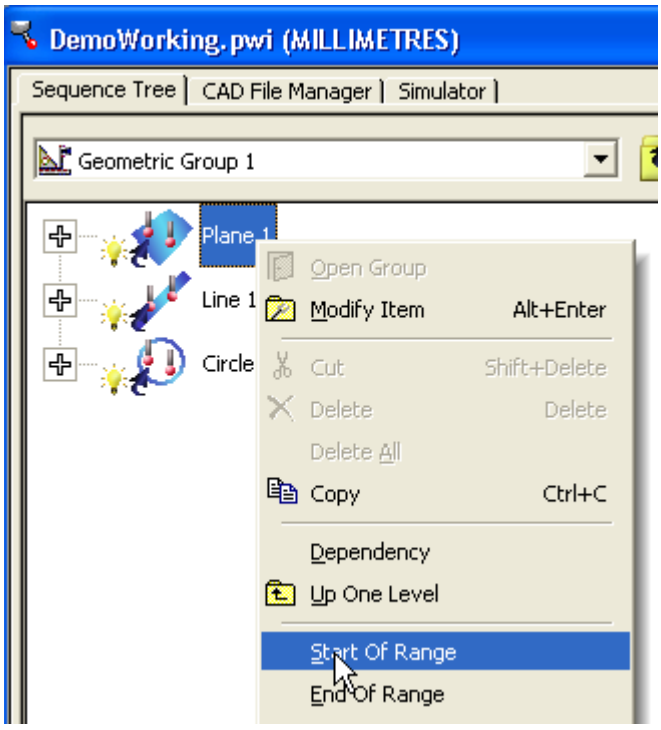


This simulation will also give you simulated results.

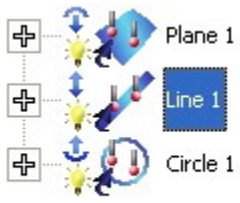


8.1 Simulating a Partial Sequence

You can set range markers to mark the start and stop of the range for simulation.



Set the start and end of the range. The features will be marked with a series of arrows, to indicate whether they are included in the current range.

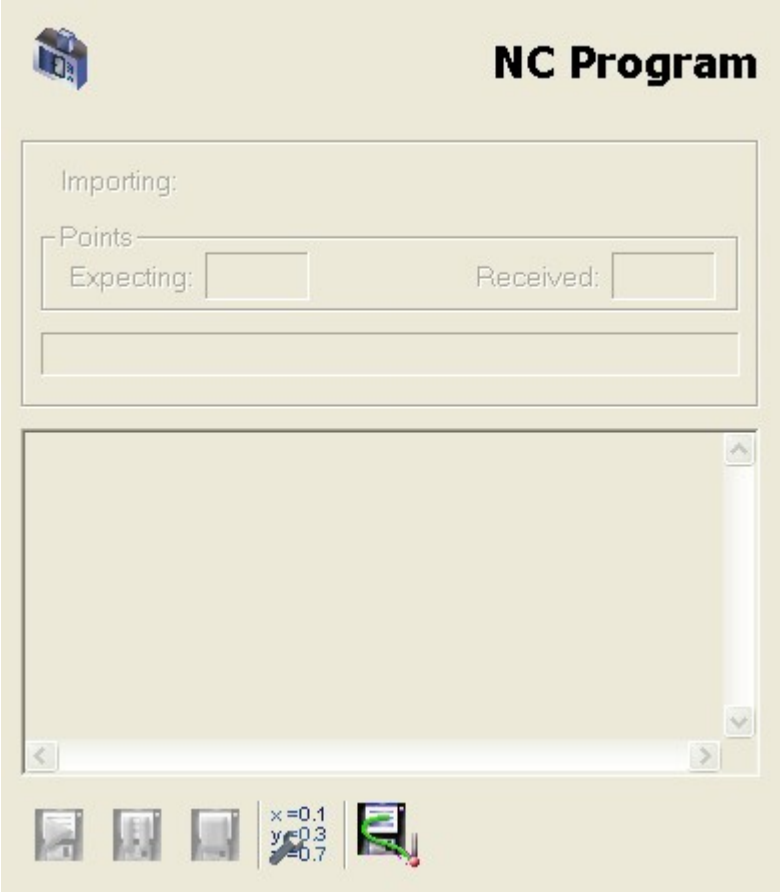


8.2 Showing and Hiding Probe paths

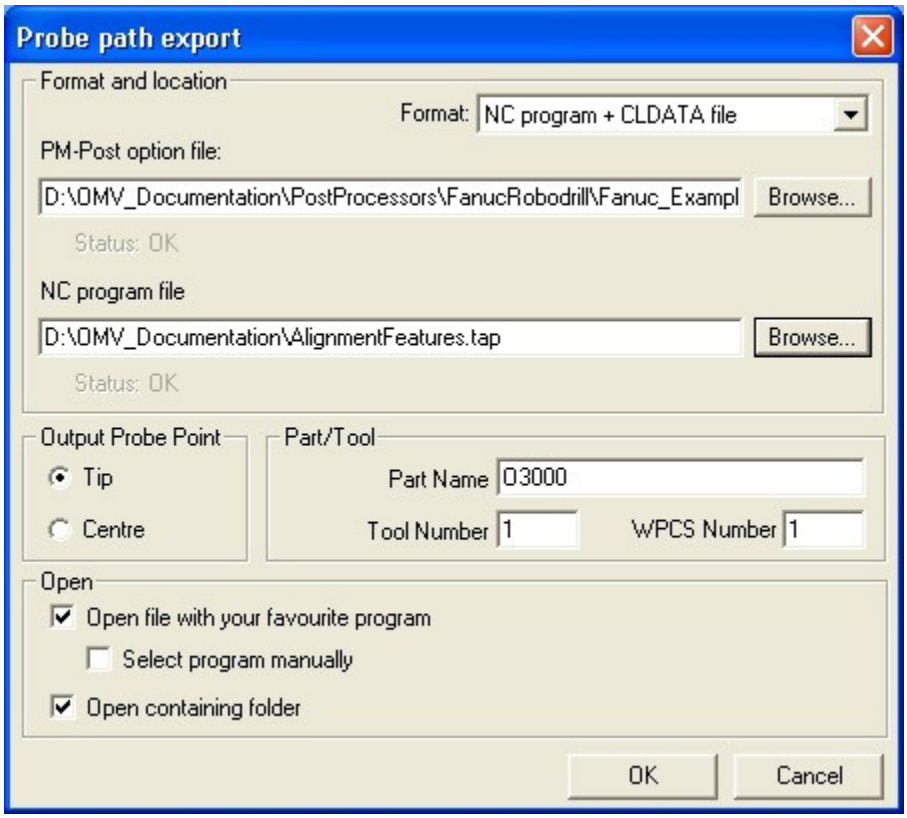
You can set which probe paths you want to display using View- CNC Probe Paths.

9. Creating an NC Program

You can create an NC Program using the OMV tab.



The export button  allows you to export an NC program. This brings up the export Window.



Choose the appropriate postprocessor and choose the name for the file. Set the output probe point to Tip.
Select the Tool Number, the name of the NC program, and the Workpiece Coordinate System number. (On Fanuc and Siemens this equates to G54, G55 etc.)

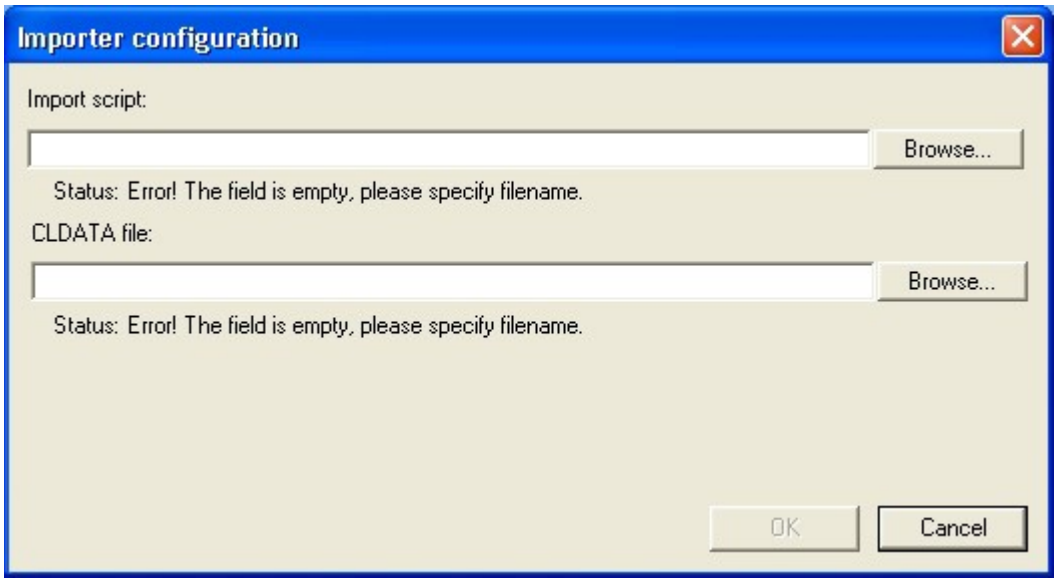
9.1 Transferring the program to the CNC controller.

The method will depend on the Controller.
On Siemens 840D you may be able to export directly to the controller, if you have networked onto the location.
You may be able to transfer the NC program to disk etc, and then copy to the controller.
On simpler Fanuc controllers you will probably use WinDNC.

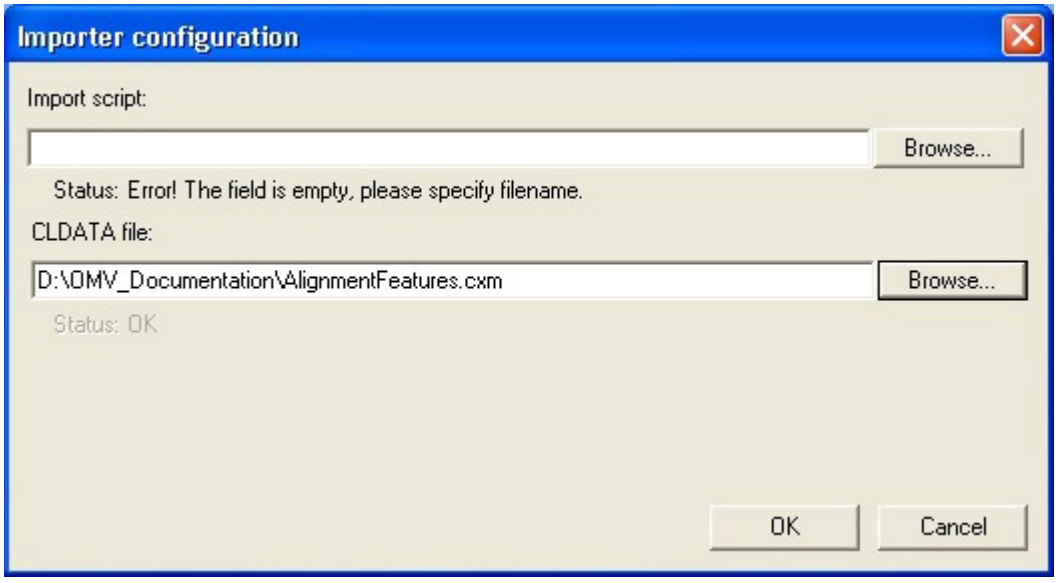
9.2 Setting up the import for results.

This depends on whether you are using a serial or file connection.

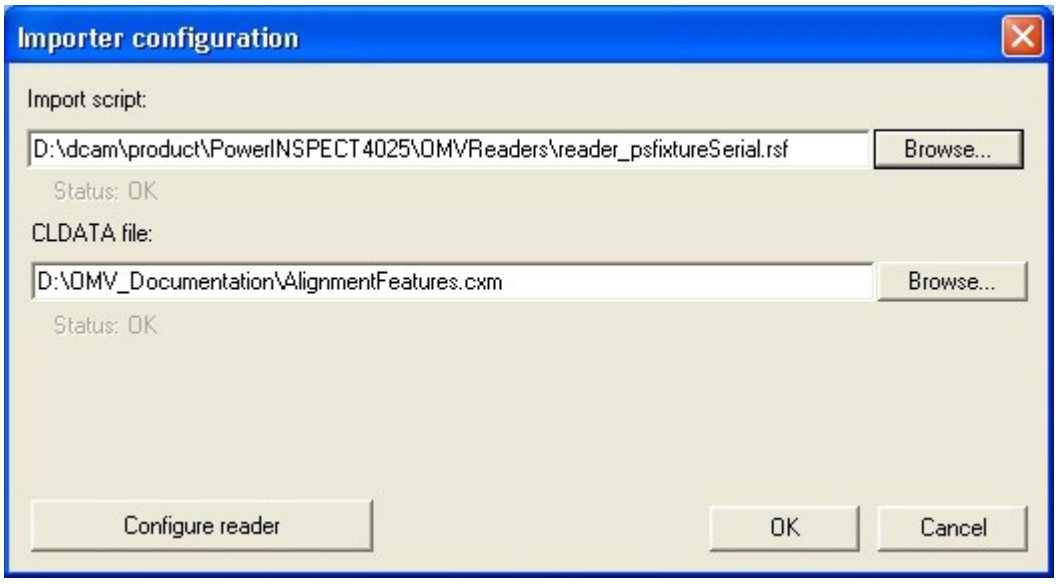
In either case, press the Configure Import button to display the Import Configuration dialog.



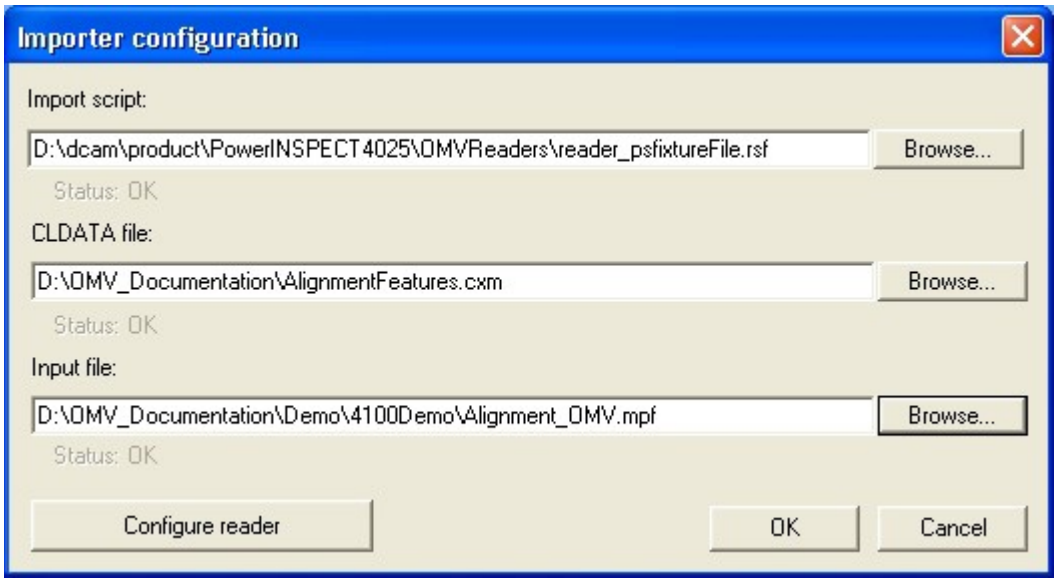
In any case, use for the CLDATA file the file that you used for the NC export. This should match the CLDATA file created in Section 9 above.



Then select the Import script. This will either be serial or from file. In the case of the serial import, you will have to set the communication parameters.



Alternatively, this will be from file, in which case you will have to identify the results file.



9.3 Running the Import

When the configuration has been set up, you will be able to run the import.

Run the import using the play button  .

When the import is ‘live’ you will be able to stop or pause the import using the other two buttons.

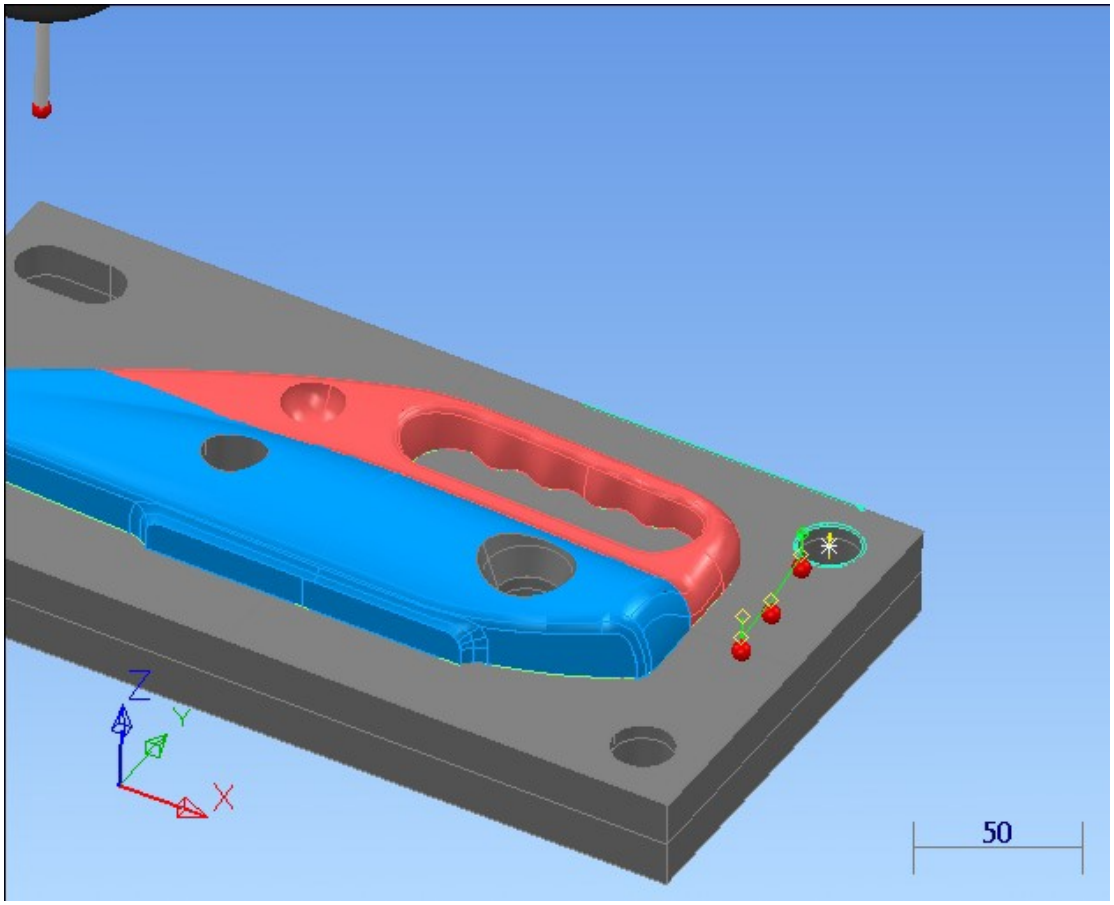
Before Import, make sure that the Measure is set to Master Part.

10. Creating a Surface Inspection Group

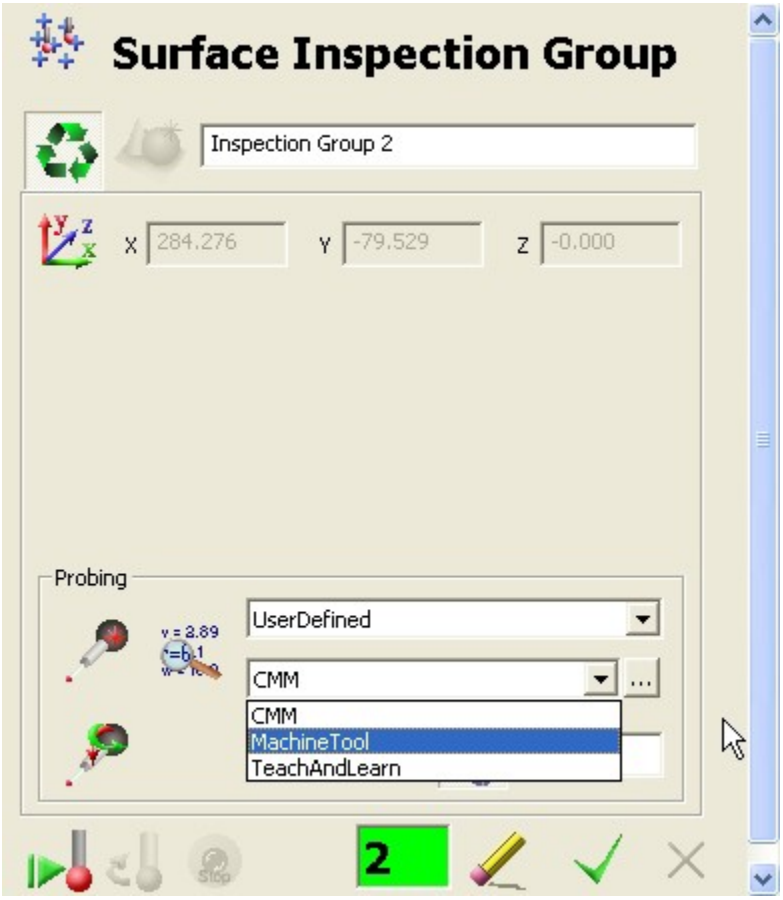
Create a surface inspection group using the Surface Inspection Group button. 

This will automatically create a user method.

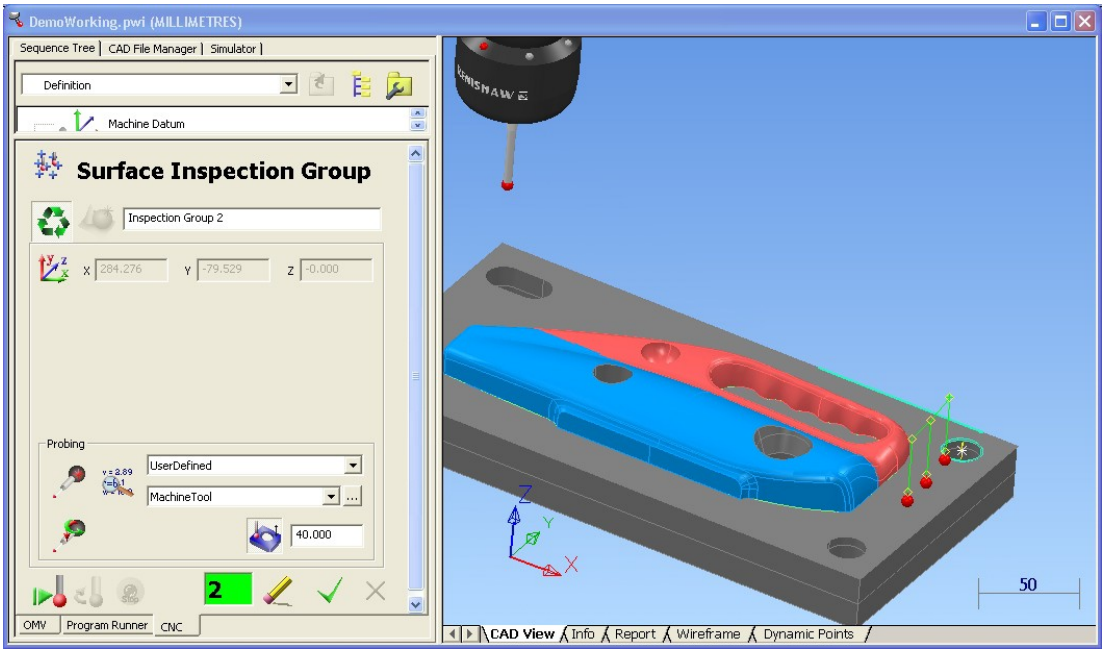
Create the first couple of points on the main surface.



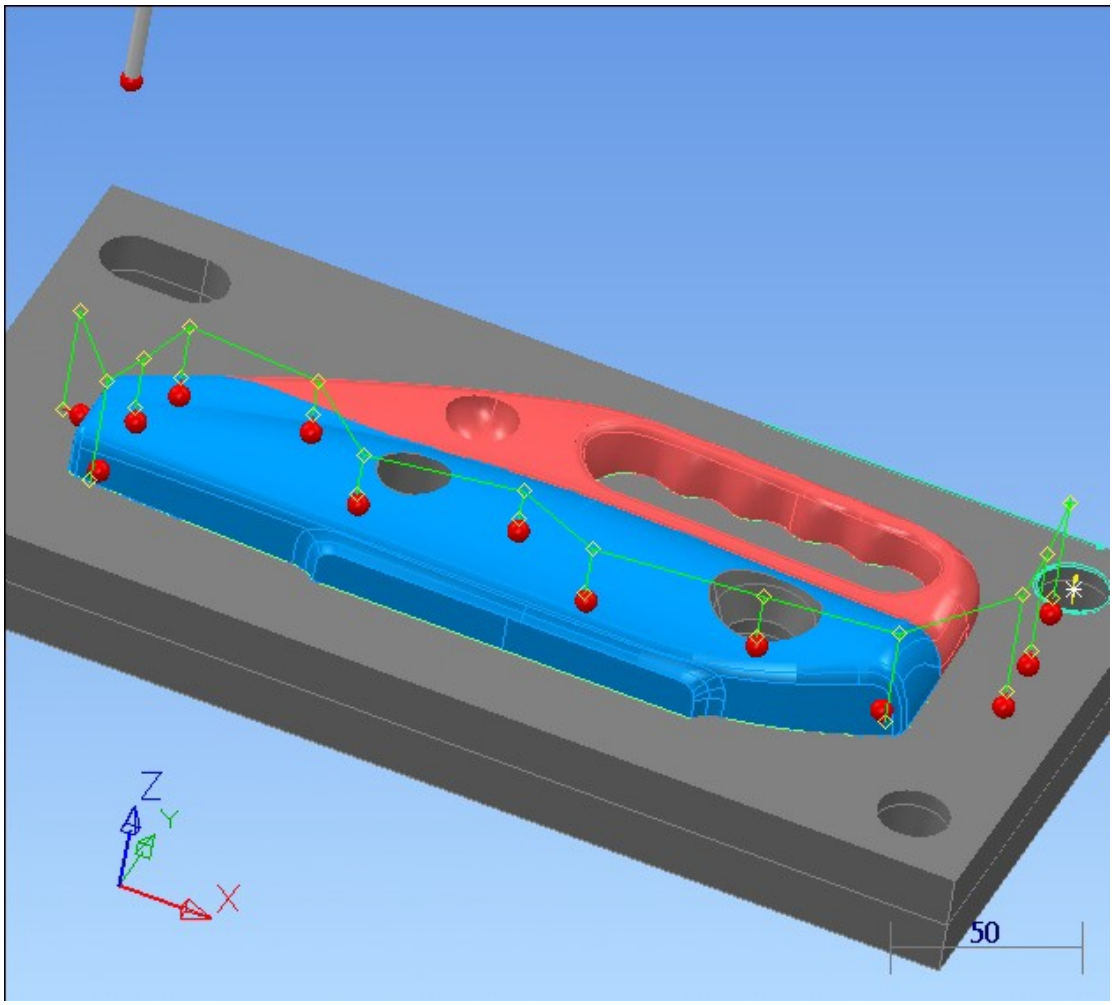
Then in the CNC tab, change the method from CMM to Machine Tool.



Set the method to Machine Tool, and set the safe plane height to 40mm. This means that the probe will come up to a plane 40mm above the surface between each point.



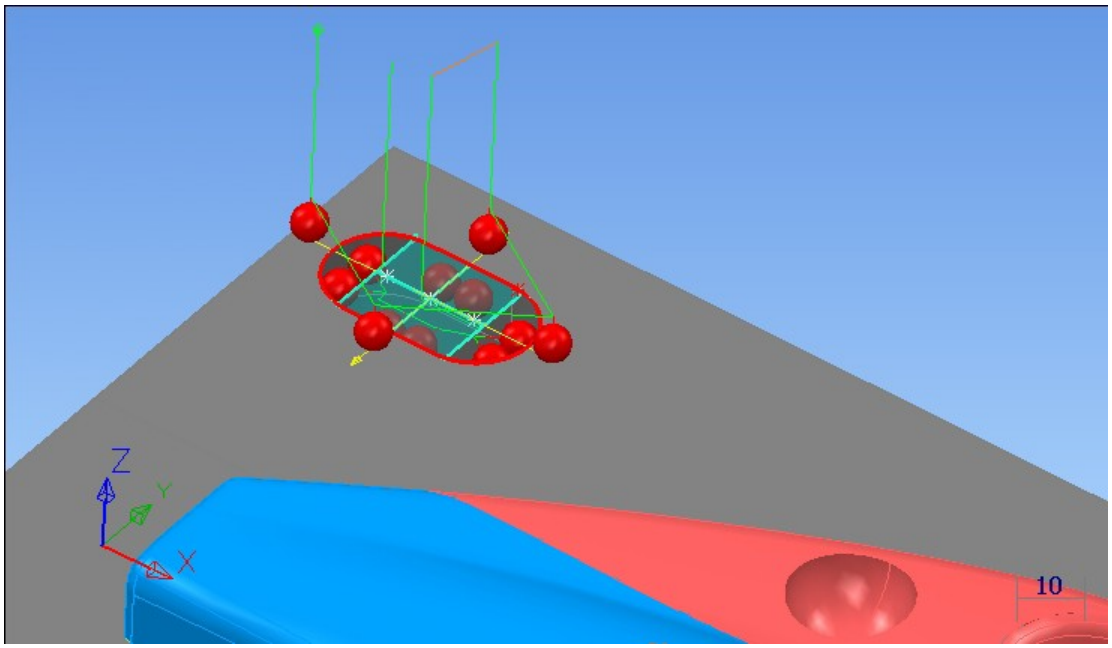
This means that it is easier to avoid collisions. Create a few points around the model and save the strategy.



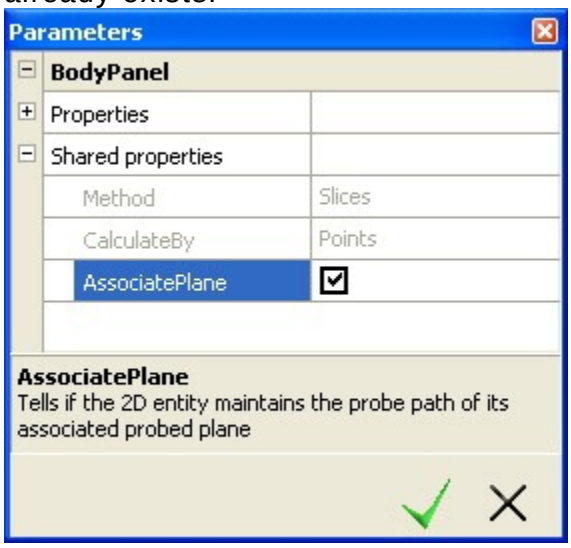
11. Slot

Create another Geometric Group, and select the slot. Set the safe plane to 40mm, and set the probing depth to 3mm.

You will notice that the slot is created with a reference plane.

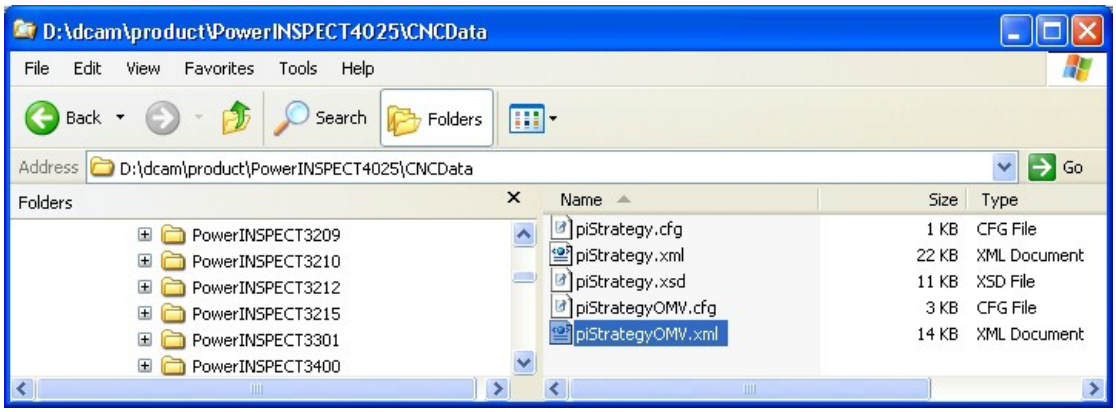


This is because the slot by default has a ‘body panel’ strategy, which uses a separate reference plane for each feature, even if a suitable plane already exists.



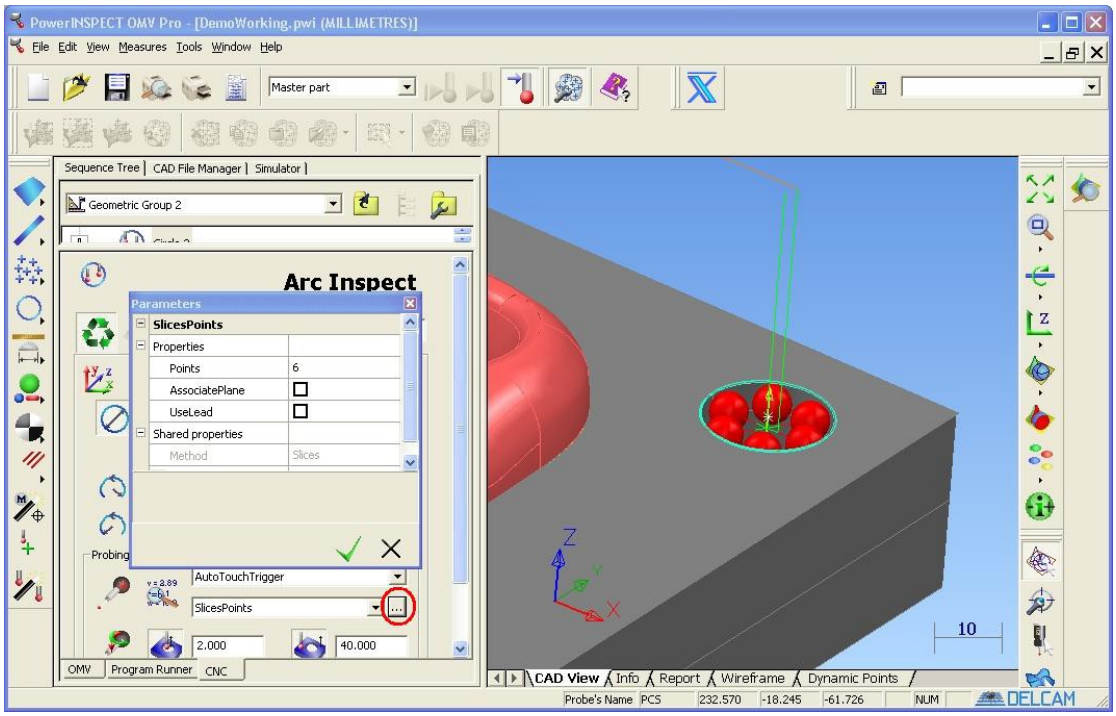
The ‘Associate Plane’ flag means that a new ‘linked’ plane will be created for each 2D feature of this type.

You can change this behaviour in the session by changing the strategy for the item, or permanently by editing the strategy file. This is called piStrategyOMV,xml, and is installed in the dcam\product\PowerINSPECTXXXX\CNCData directory.



12. Circle 2

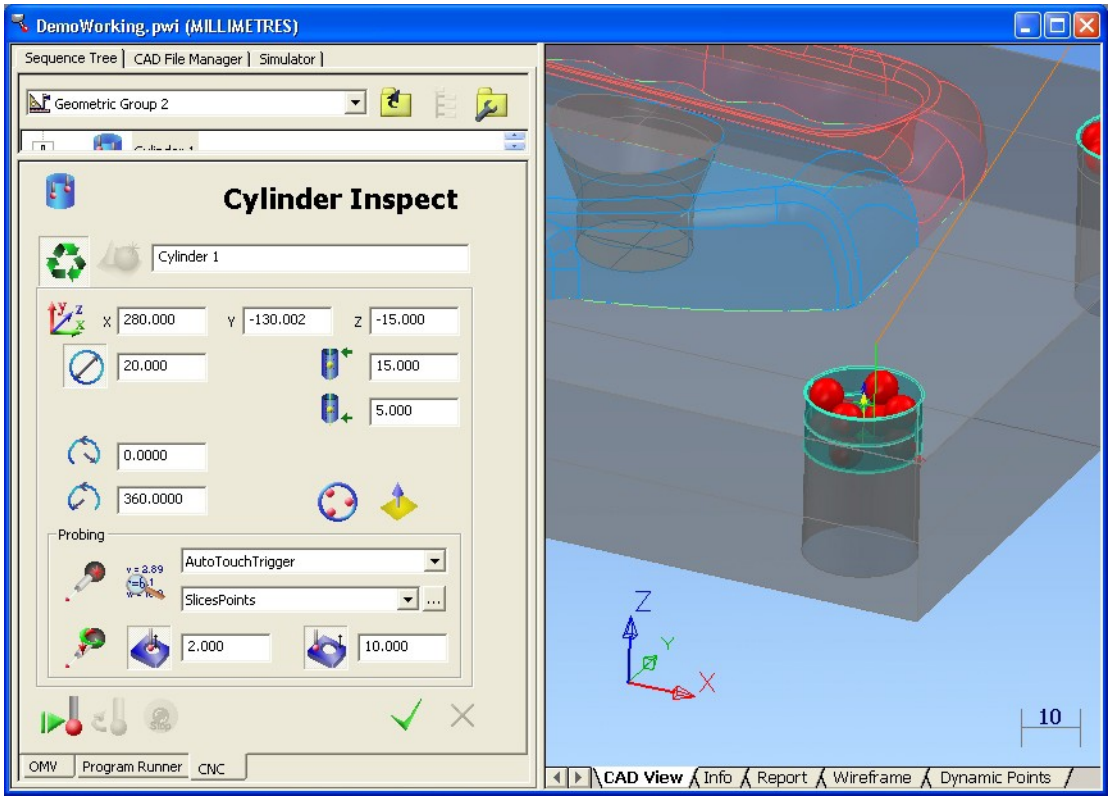
Create a second circle, this time using a different number of points (use the strategy button to do this.)



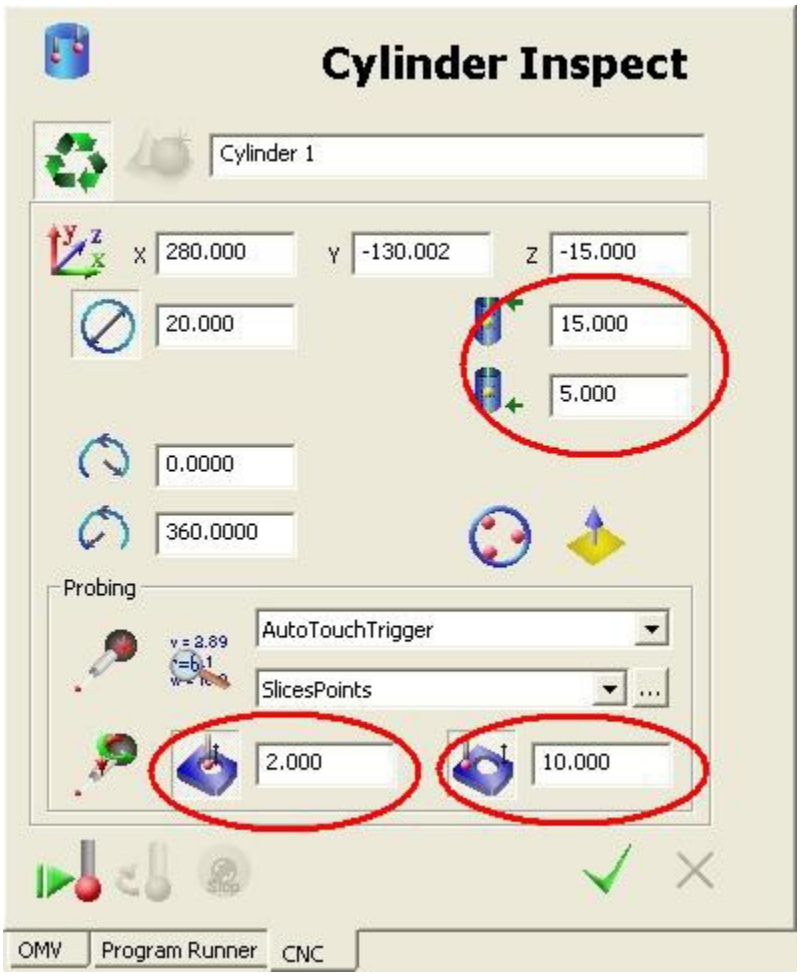
13. Cylinder

In the Wireframe Checker, change to surface mode to allow you to pick 3D features, such as cylinder and cones. Right click in the CAD view, and select Surface.

Select the cylinder in the model (pick on the wireframe lines, rather than in the spaces) and pick the cylinder.

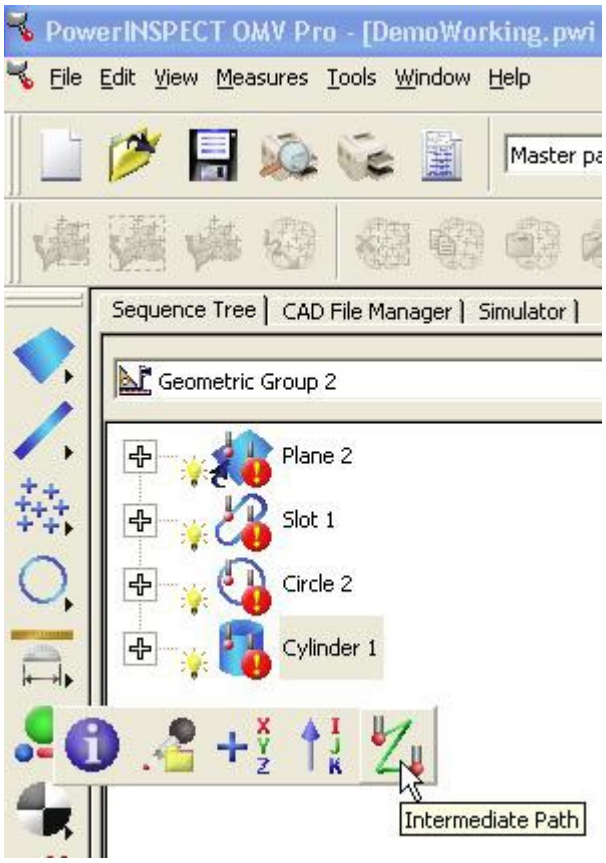


Edit the height of the cylinder, so that the upper height is 15 and the lower height is 5. This shortens the probed cylinder.
Edit the probing depth, and change this to 2 mm. This brings in the probed points from the end of the feature.
Finally, set the probe safe plane is 10mm.



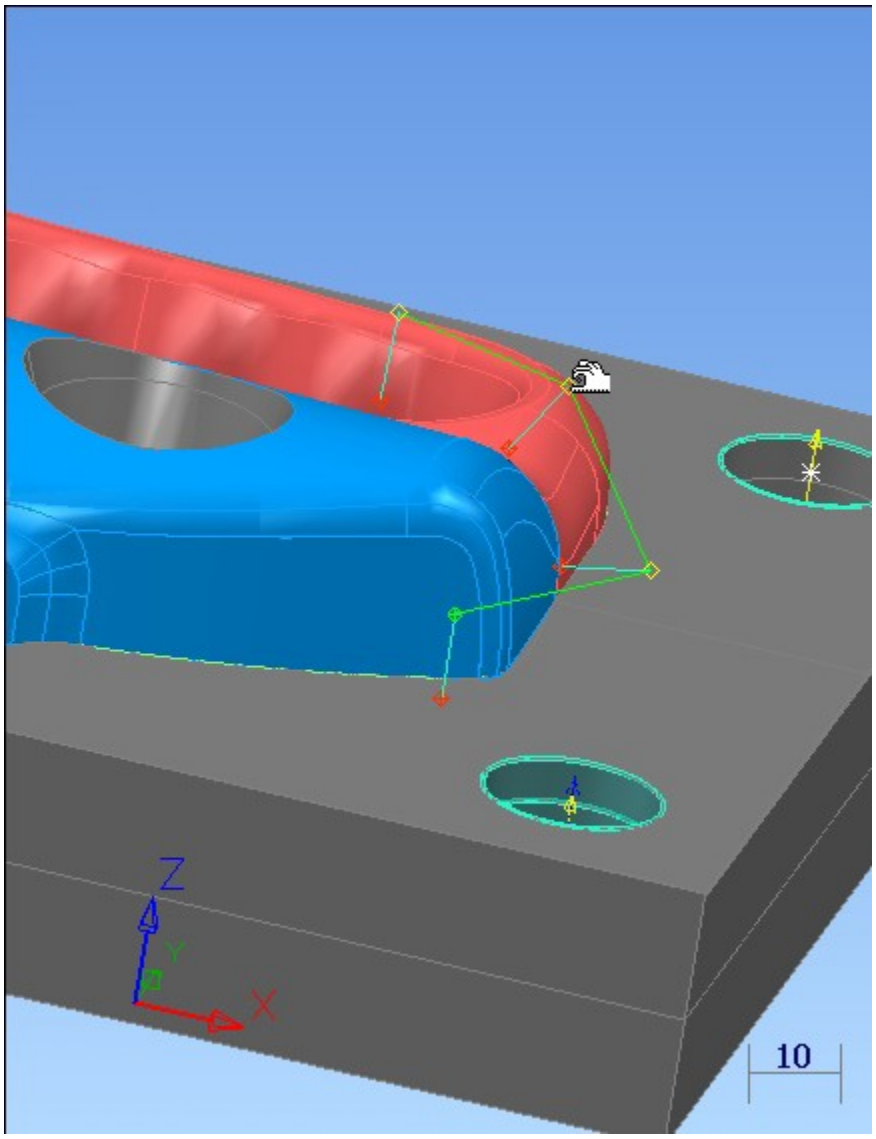
14. Intermediate Path.

Create an intermediate probe path.



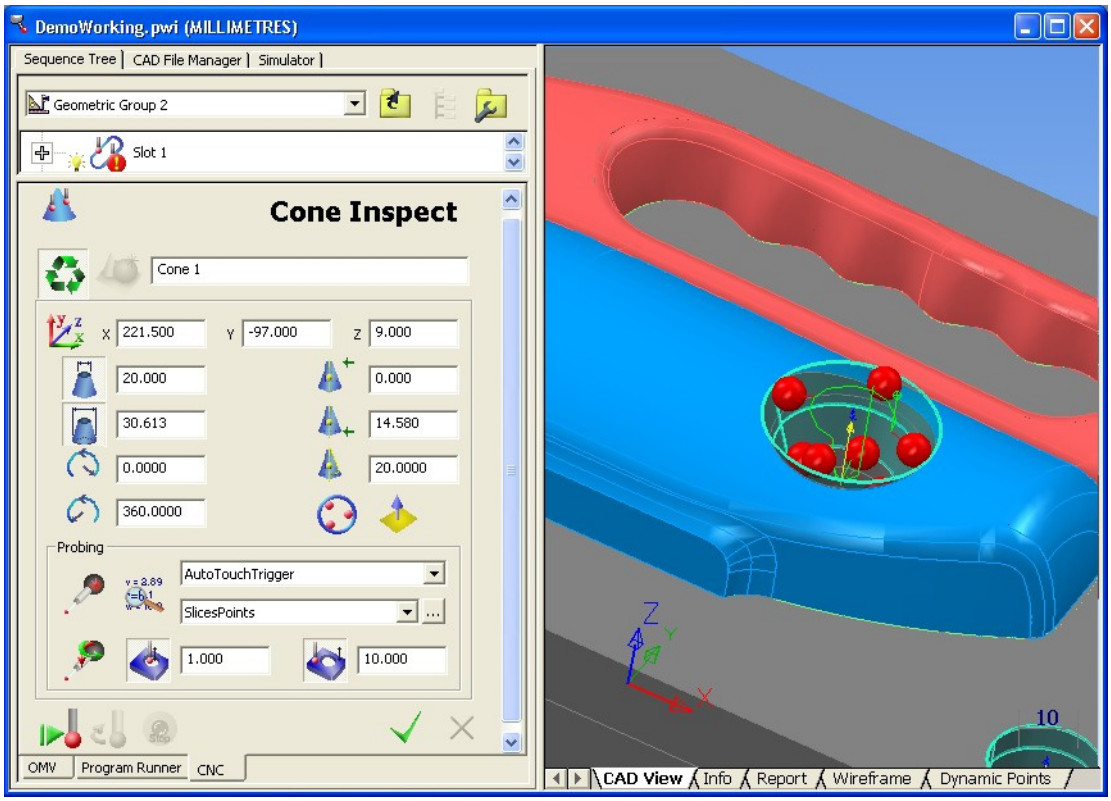
This allows you to create a probe path as a user method, but the points are not actually probed on the surface- they are offset from the model by a distance which is set by the 'probe safe plane' parameter.

Create an intermediate path going from the main plane to the top of the model near the cone. You can also edit the probe path by moving the points with the cursor.



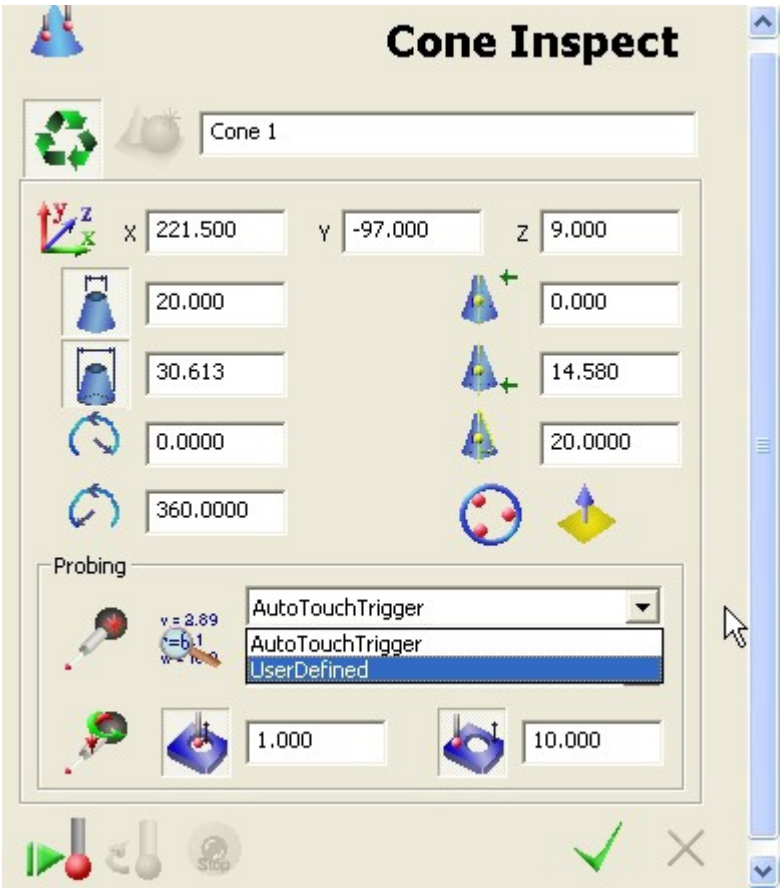
15. Cone with modified probe path

Now create the cone on the top of the model.

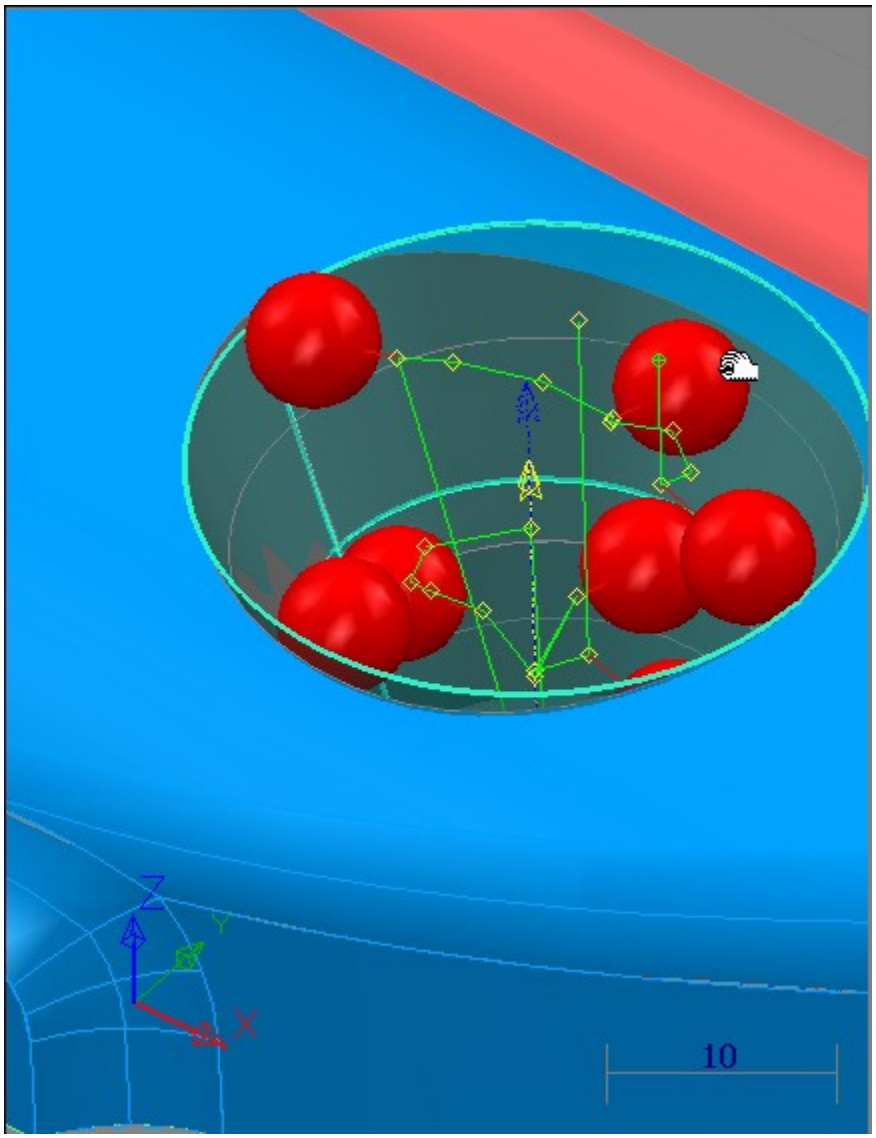


Although the cone is created properly, you will notice that some of the points are very near the ends, or actually in space.

We will modify the cone to use a user strategy, and manually move the points to a better location.



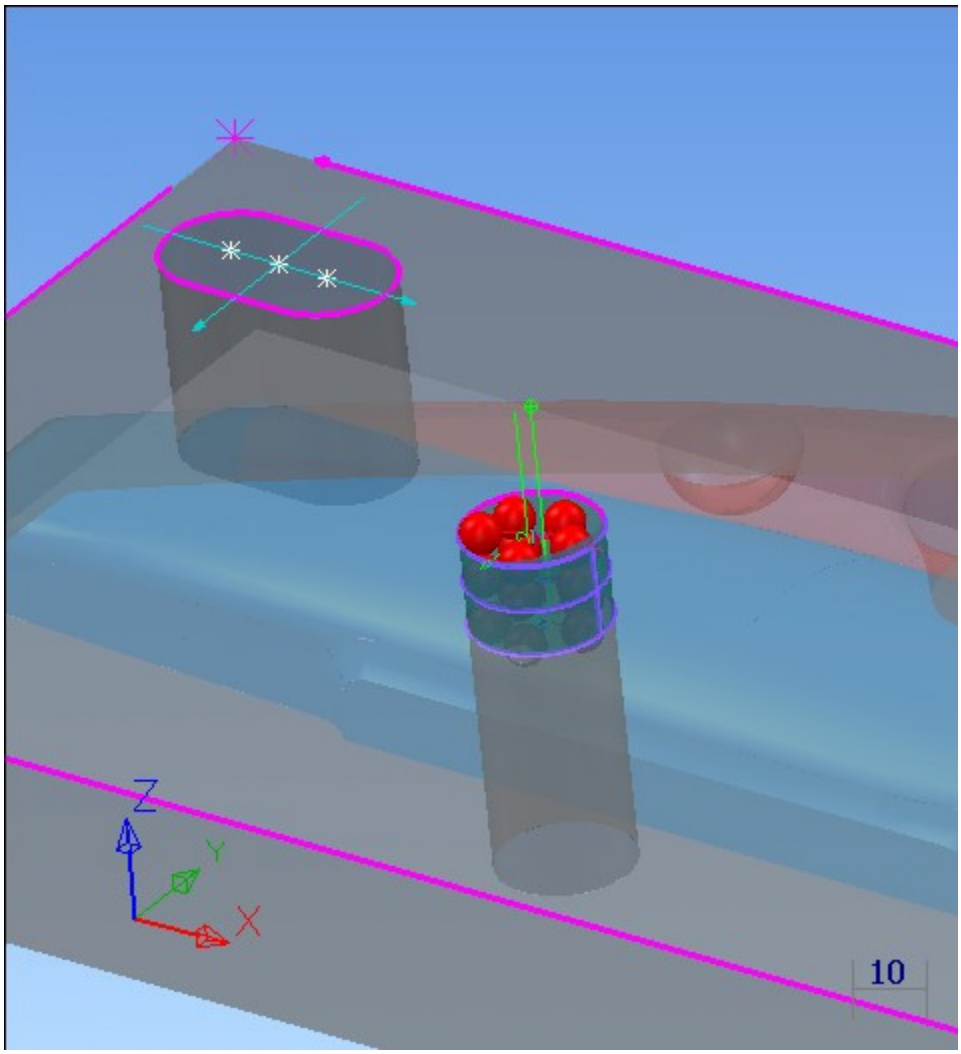
Change the method to User Defined, and then you will be able to move the probed points manually.



The auto strategy can be converted to a user strategy and vice versa.

16. Cylinder with modified probe path

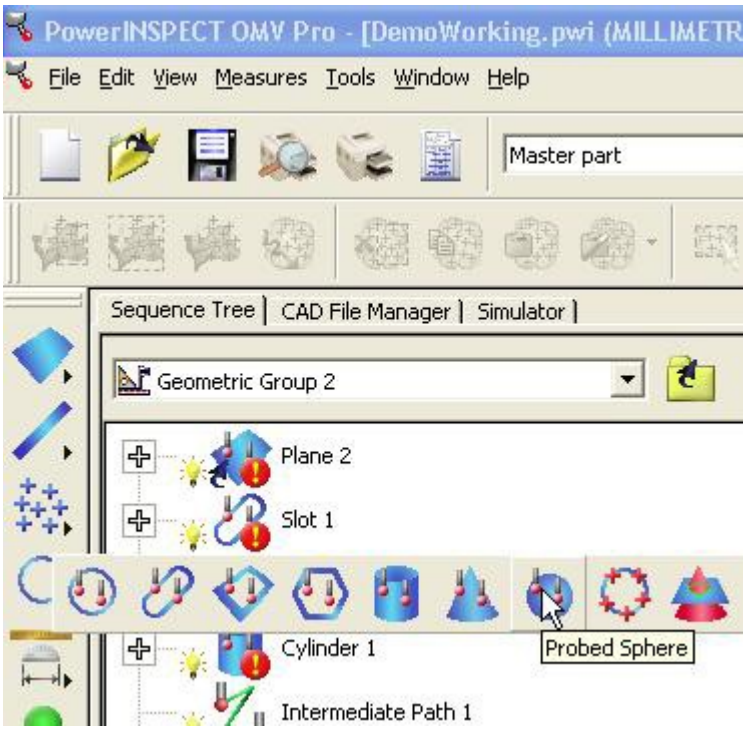
Pick the cylinder on the top of the model. As before, change the height of the cylinder so that only the top 10-15mm is used. Then, as with the cone, edit the method from Auto Touch Trigger to User defined, and move the points so that they stay on the surface.



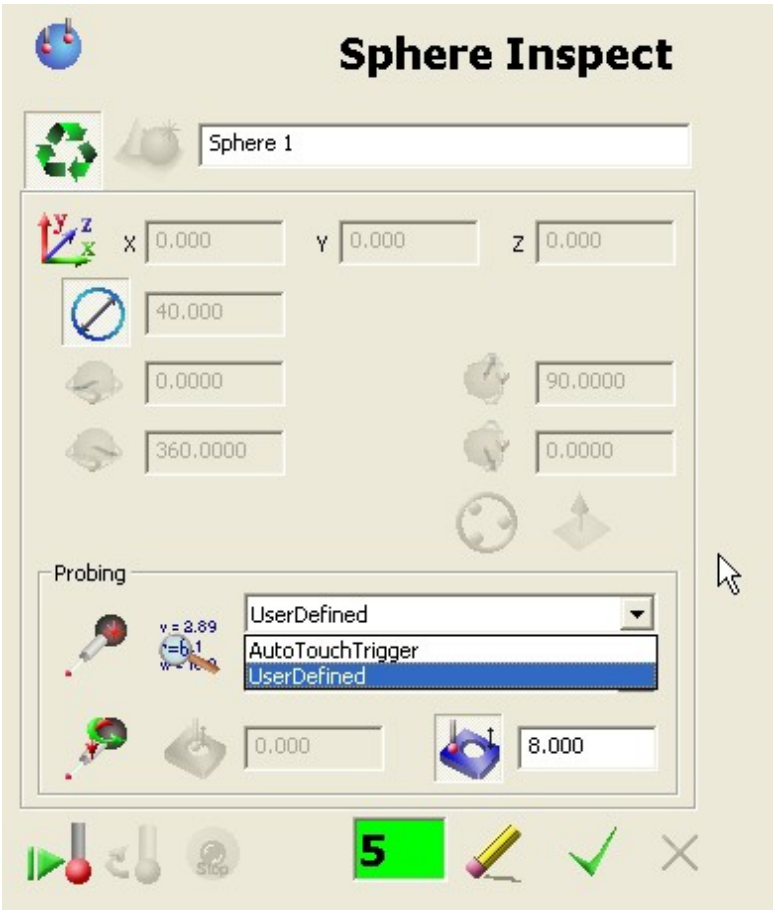
17. Sphere

This time, we will start with a User Defined Strategy and change to an auto touch trigger.

Select the Sphere as a Probe Sphere from the interface.



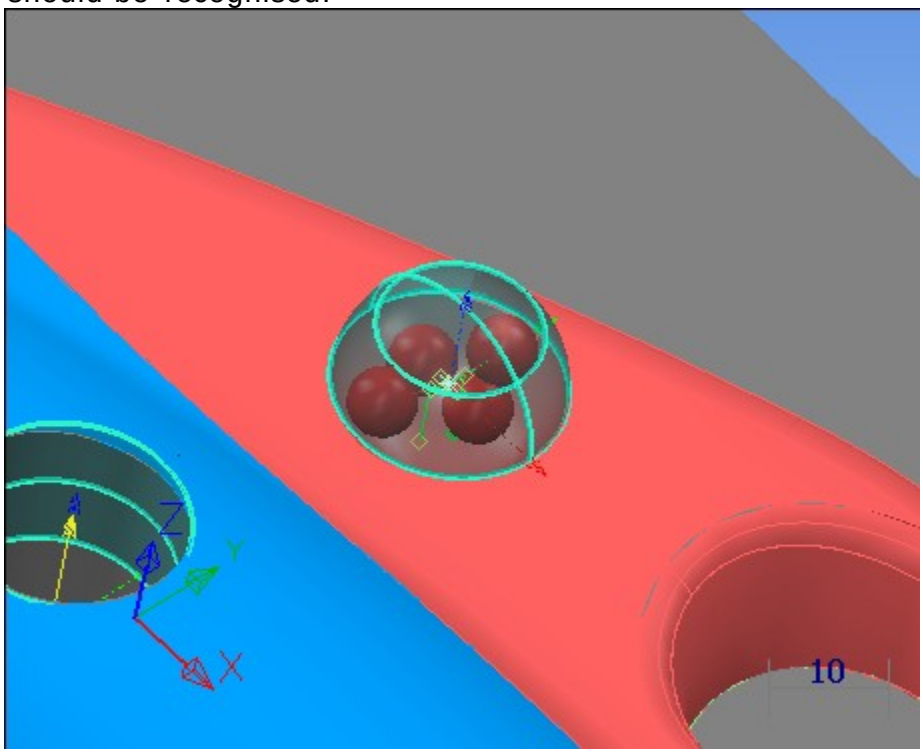
Change the method to User Defined



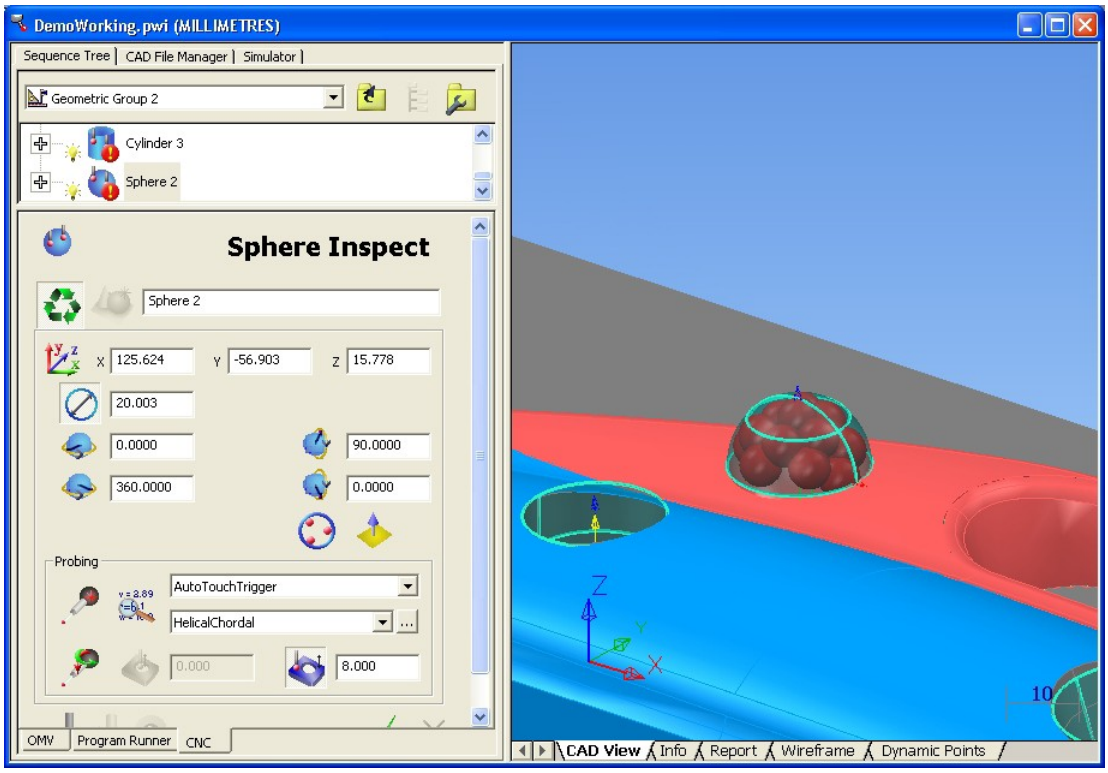
Now delete the automatically created probe path using the ‘remove point’ button. 

Do this until the counter is empty 

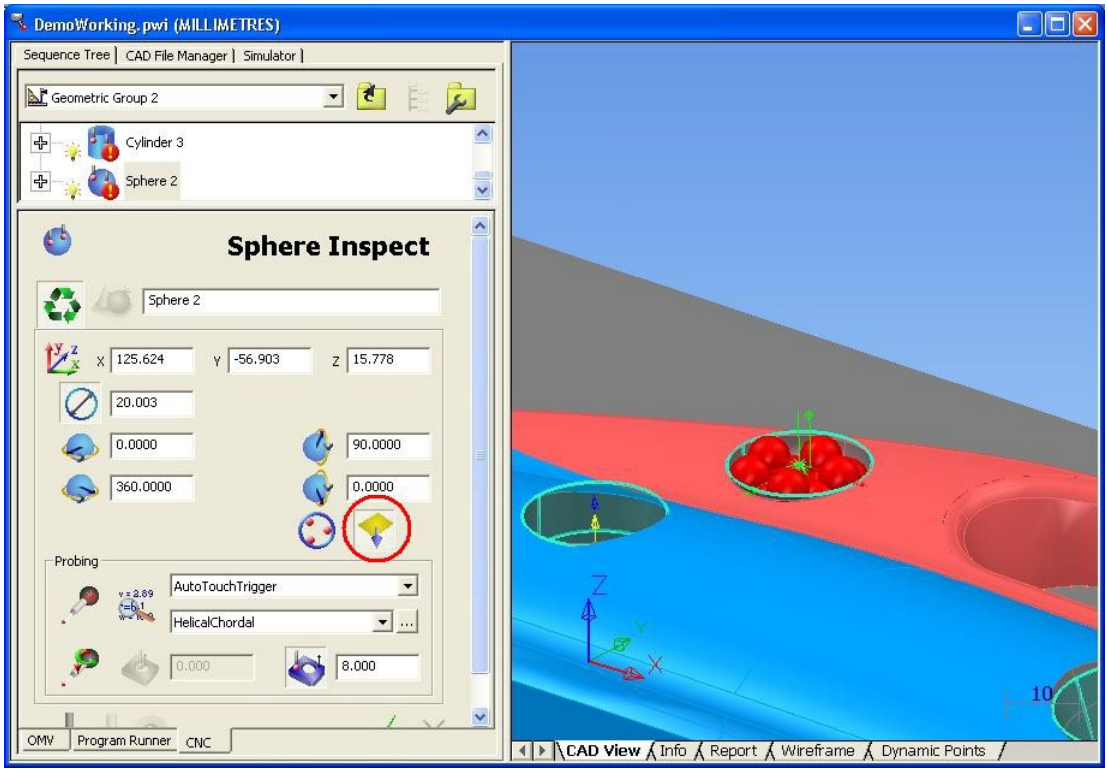
Now place 4 points on the sphere. When you pick 4 points, the sphere should be recognised.



Next change the method from user defined to Auto Touch Trigger. This will distribute the points evenly on the feature.



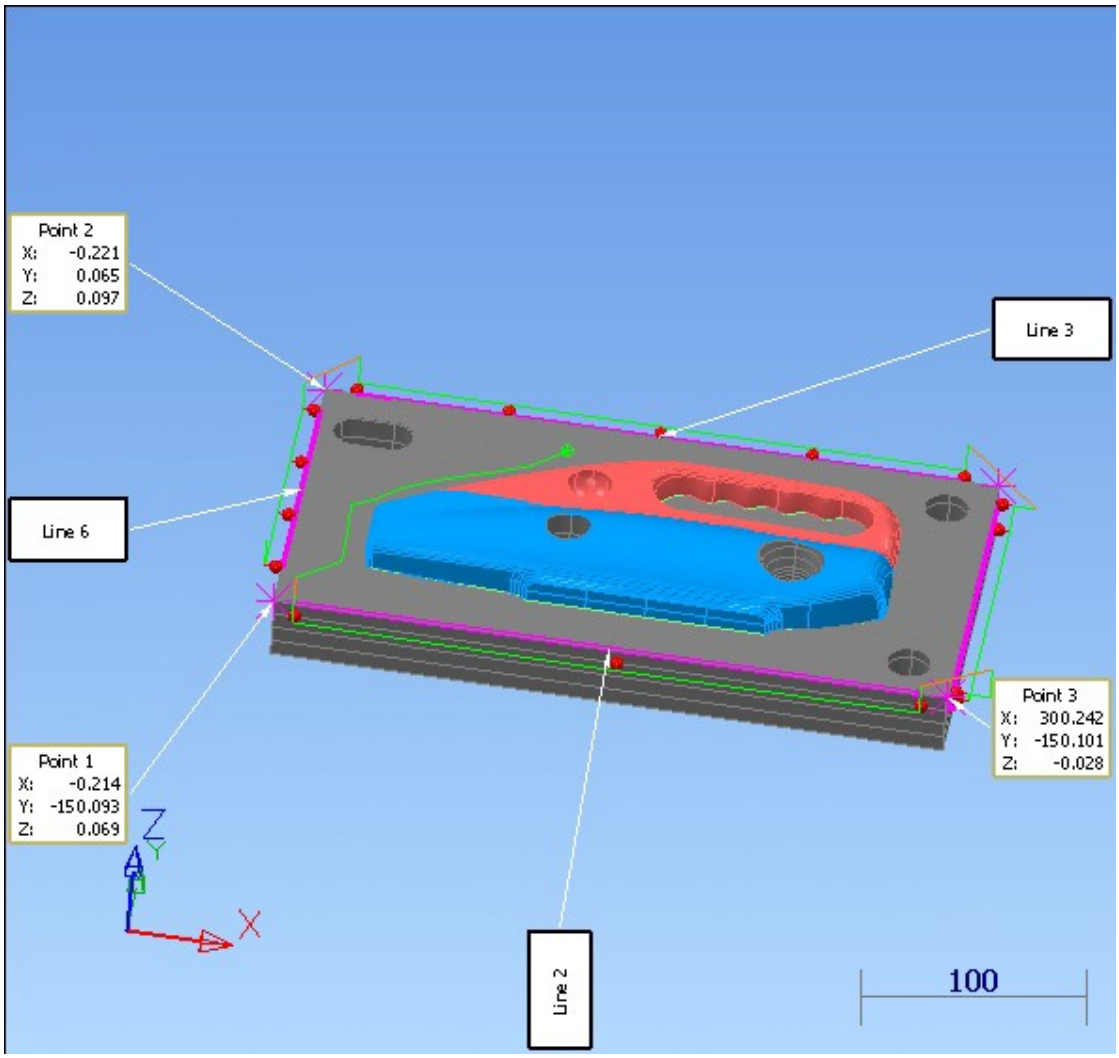
Now you have to fix the orientation using the ‘inverse probe path button’.



Finally, move the boundaries to 90 degrees and 10 degrees, to take account of the bounding of the intersecting surfaces.

18. Remaining Geometry

Create an intermediate path from the sphere to the front edge, followed by lines (or planes) around the outer faces.

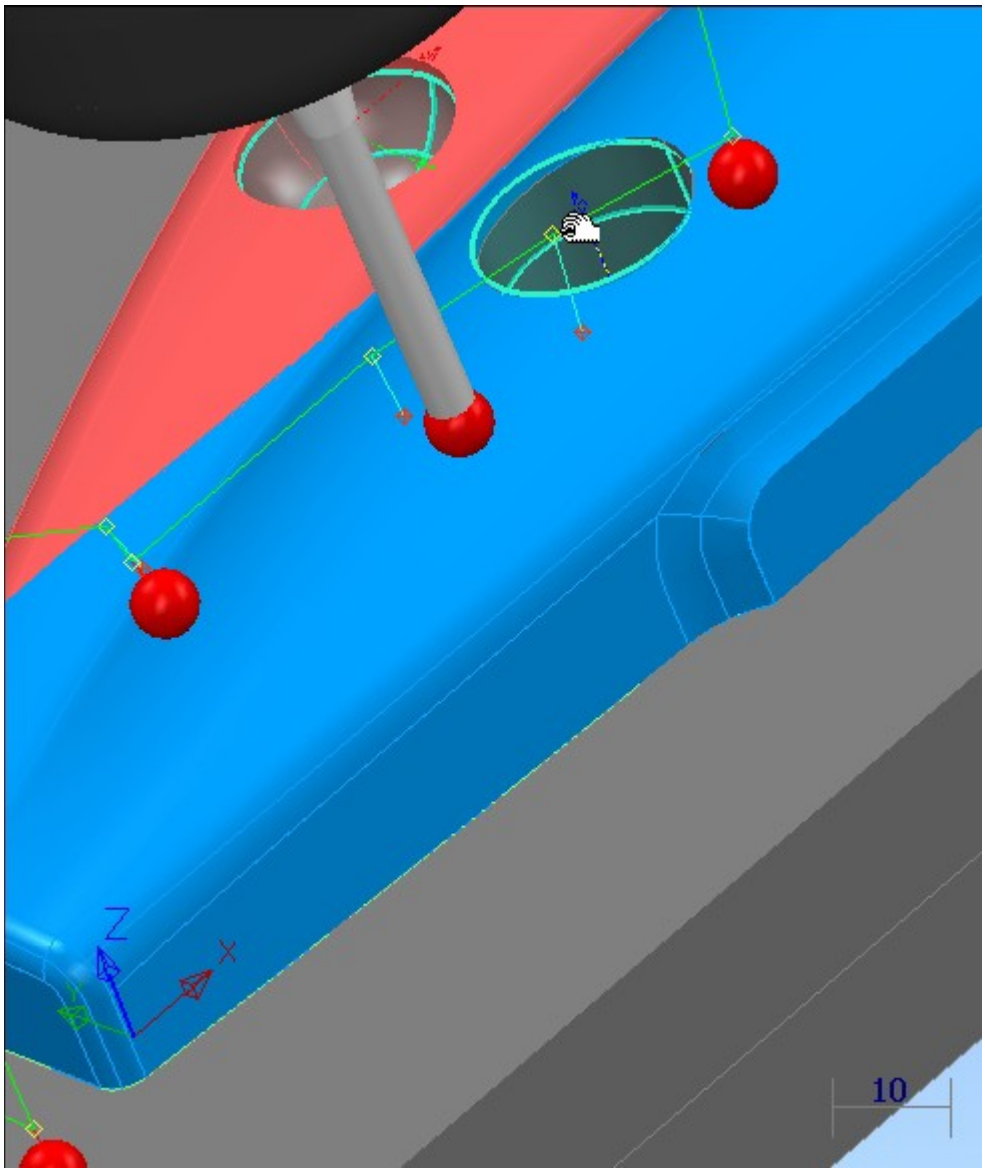


19. Surface Inspection with CMM Strategy

Finally, create one of more inspection groups using the CMM strategy.

As with the Machine Tool strategy, this is a user strategy- you pick the points individually- but this time the points will skim the surface. This means that you have to be more careful to avoid collisions.

To help avoid collisions, you can create intermediate points using the SHIFT key while double-clicking the points. This will create an intermediate point away from the surface- as with the intermediate probe paths- and as before you can manually change this distance by pulling the points.



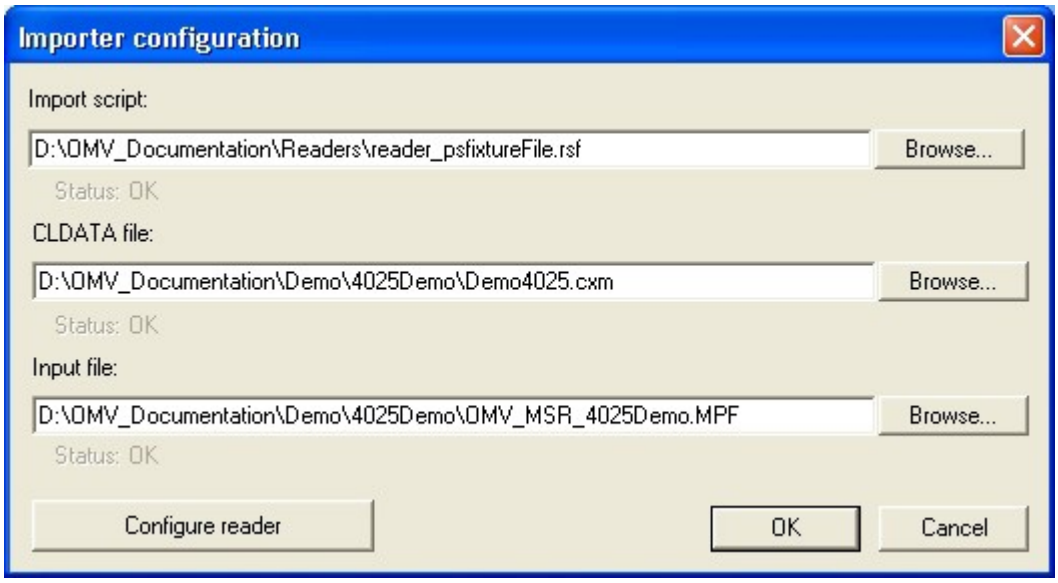
20. Showing OMV Import

With the files on the demo CD, you can show the OMV import using points from file. Copy the files from the OMV_Documentation\Demo\4025Demo directory.

In the OMV tab, use the reader_psfixtureFile.rsf file (this is installed with PowerINSPECT in the dcam\product\PowerINSPECT4025\OMVReaders directory.

For the CLDATA file, use the Demo4025.cxm file.

For the data input, use the OMV_MSR_4025Demo.mpf file.



Press the ‘play’ button on the OMV import, and this should read in the data as below.