
PowerINSPECT OMV 2016

Quick Start Guide



PowerINSPECT OMV

Copyright © 1993-2015 Delcam Ltd. All rights reserved.

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of the software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

Contents

Introduction to PowerINSPECT OMV	1
The OMV process	4
Tutorial example	5
Creating an inspection document	6
Understanding the screen layout	7
Manipulating the CAD view	9
Specifying the machine tool details	10
Aligning the CAD model to the part	11
Stage 1: Delete the default alignment	12
Stage 2: Create a geometric group	12
Stage 3: Create a probed plane	14
Stage 4: Create a probed line	15
Stage 5: Create a circle	17
Stage 6: Create a PLP alignment	18
Working with probe paths	20
Verifying probe paths	21
Simulating measurement	23
Resolving verification faults	24
Adding further details to the inspection sequence	26
Creating a cylinder	27
Creating a CNC surface inspection group	28
Positioning the probe	31
Using automatic orientation	32
Orientation Point items	34
Using orientation points	35
Measuring an inspection sequence	38
Exporting the inspection sequence to the machine tool	39
Running the NC program	41
Importing measurements into PowerINSPECT OMV	42
Displaying the measurement results	44
Creating an on-screen report	45
Creating an HTML report	46
Optimizing the alignment (Best Fit)	48
Exporting a section of the inspection sequence	50

Creating a probe path from imported points 52

Introduction to PowerINSPECT OMV

PowerINSPECT OMV performs rapid part verification on a machine tool — a process known as **On Machine Verification (OMV)**. This allows you to spot problems while the part is still on the machine tool.

Choosing what to inspect

PowerINSPECT OMV's inspection sequence specifies the features to be inspected and the order of inspection. You can add features to the inspection sequence by picking them from a CAD model, or by selecting them from the **Item** toolbar.

What can you measure?

You can measure the following types of feature on the part:

- Surfaces.
- Geometric features, including 2D features (such as circles) and 3D features (such as cones).

Aligning the part

PowerINSPECT OMV provides several methods of aligning the part to the CAD model:

- From File alignment — Loads a previously saved alignment from file. You can use this option when inspecting multiple examples of a part. When you use this alignment, each example must be identically positioned on the inspection table.
- Geometric PLP alignment — Creates an alignment using geometric features that have known nominal coordinates.
- Reference Point System (RPS) alignment — Creates an alignment using the coordinates of up to one hundred features.

- User Defined alignment — Aligns the part by specifying the relationship between the reference systems of the Machine Datum and the PCS (CAD Datum).

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

Inspecting the part

You perform inspections by exporting the inspection sequence as an NC Program, and then running the program on the machine tool. When the inspection is complete, you import the measurements into PowerINSPECT OMV for a detailed analysis of the results, including deviations from the nominals and the positions, sizes, and form of geometric features.

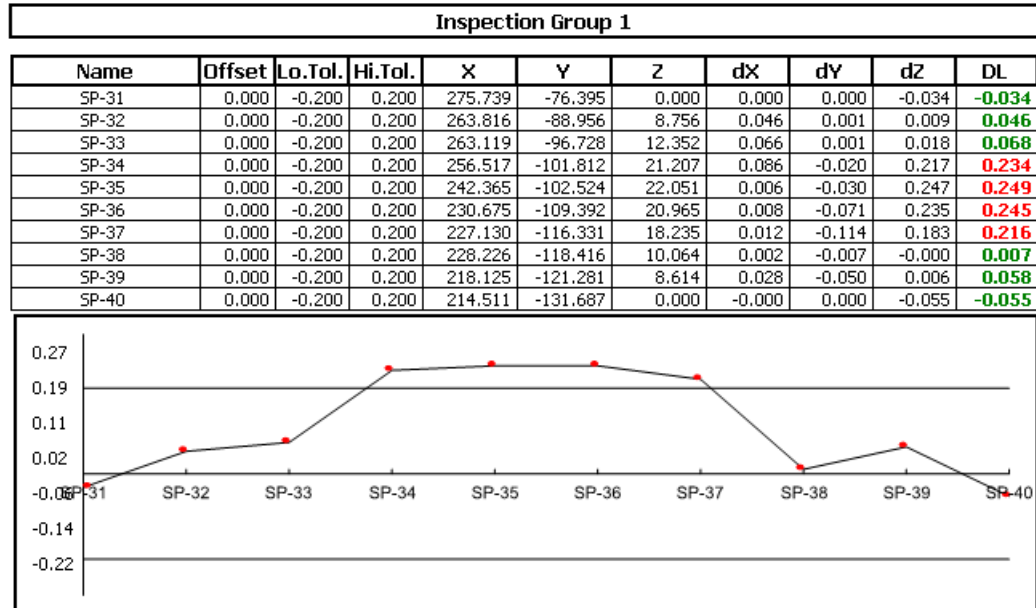
You can export NC programs and import results using a real-time connection with a serial cable, or by transferring a file of points to the measuring device.

Viewing the results

PowerINSPECT OMV provides several types of feedback:

- Inspection points and labels displayed in the CAD view use colour-coding to identify which items are within tolerance, above tolerance, and below tolerance.
- The **Info** tab provides full details of the selected item in the inspection sequence, including position, size, form value, and deviation from nominal information.

- The **Report** tab produces a customizable HTML report. In it, you can show all measurements for all features in your inspection, or you can choose which measurements to include. You can also generate printable reports using a PDF generator or Microsoft Excel.



The OMV process

PowerINSPECT OMV uses a standard method for identifying the features to be measured, building a measurement program and collecting measurement data, so you can use the same process for inspecting any type of part. The process for creating an inspection is detailed in the following steps:

- 1 Using the **New Document Wizard**, create an inspection document, and, if available, load the CAD model for the part you want to inspect.
- 2 In the **Machine** tab, specify the probe assemblies with which you want to inspect the part.
- 3 In the **Sequence Tree** tab, create an alignment for the part and specify the features to be inspected. For each feature, you must specify the points to be measured and the points to be probed.
- 4 In the **Simulate Program** tab, check the inspection sequence, and correct any problems.
- 5 In the **NC Program** tab, export the inspection sequence to an NC program file.
- 6 Transfer the NC Program file to the machine-tool controller. You can transfer the file using a memory stick, or a serial or network link between your computer and the machine controller.
- 7 Run the NC Program file on the machine tool.
- 8 In the **NC Program** tab, import the measurements into PowerINSPECT OMV.



If you are using a serial or network link, you may be able to import each point as it is taken.

- 9 View the results in PowerINSPECT OMV using the **Info** and **Report** tabs.

Tutorial example

This section describes how to:

- create an inspection (see page 6).
- understand the PowerINSPECT OMV screen layout (see page 7).
- manipulate the CAD model on the screen (see page 9).
- specify the probe and probe tools to be used to perform the inspection (see page 10).

Creating an inspection document



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

CAD

Knife

New... Details...

Surface Points

Surface Offset 0.000 Tolerance -0.200 0.200

Low High

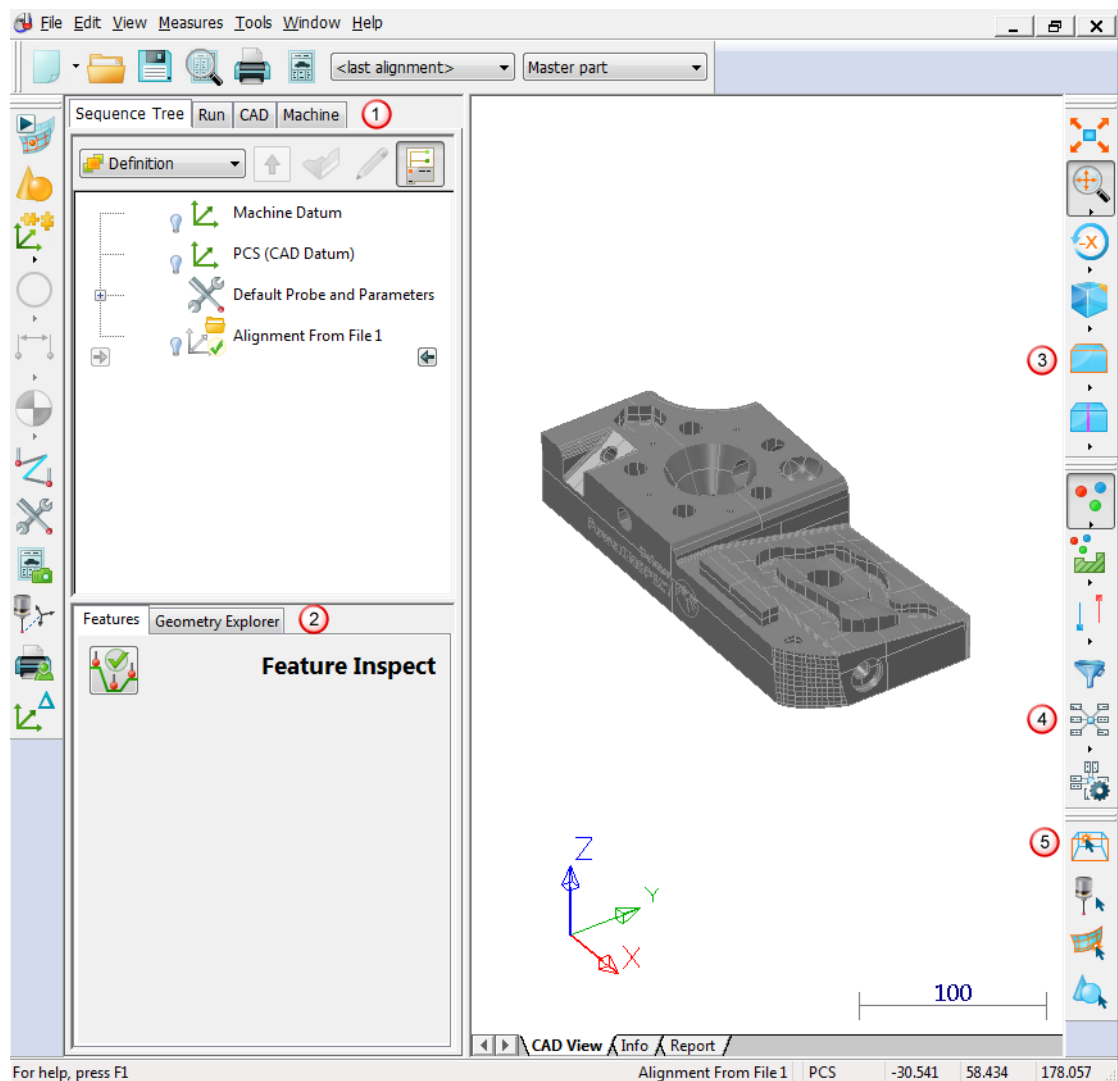
☒ Save parameters as default Units MM

< Back Next > Finish Cancel Help


- 4 Click **New** and select [knife.odx](#) in the [Samples](#) folder.
- 5 Click **Finish** to open the new document and display the model in the **CAD View** tab.

Understanding the screen layout

The main features of the screen are shown below:












Area	Description
1	<ul style="list-style-type: none"> The Sequence Tree tab lists the measured features. The Run tab controls simulation of the inspection sequence, and the export of NC Program to, and import of data to from the machine tool. The CAD tab contains details of the CAD file displayed in the main CAD view. The Machine tab controls simulation of the probe and stylus.

Area	Description
②	<ul style="list-style-type: none"> ▪ The Features tab specifies features and probe paths. ▪ The Geometry Explorer tab shows details of features picked for inspection using the Wireframe Checker - see ⑤.
③	The Shading options can aid visualization of the model.
④	The Label options display nominal and measurement information in the CAD View tab.
⑤	<p>The Wireframe Checker  button enables you to add items to the inspection sequence by picking geometric features directly from the CAD model.</p>

Manipulating the CAD view

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

To...	Use...
Zoom in and out of the CAD view.	 or 
Zoom in on an area by drawing a box around that area.	 or  or 
Pan the CAD view.	 or 
Rotate the CAD view. A globe is displayed in the CAD view to show the direction of rotation.	 or 

Where:

- The mouse button in red indicates which button you must press:



- One or more keys indicates you must press these keys on your keyboard at the same time as pressing a mouse button:



- The blue direction arrow indicates you must rotate the mouse wheel:



Specifying the machine tool details

The **Machine** tab specifies the details of the probe and the probe tools to be used to perform the inspection. This enables PowerINSPECT OMV to generate a graphical simulation of the probe paths so you can check for any problems.

To specify the machine tool details:


- 1 Select the **Machine** tab.
- 2 In the **Probe** and **Probe tool** sections, select the probe and stylus with which you want to measure the part.
- 3 Click **Export As Default** at the bottom of the **Machine** tab so that the probe and stylus you picked are automatically selected when you create new inspection documents.

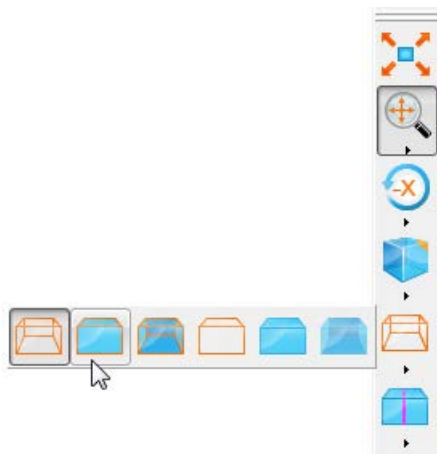
The **Machine** tab details are recorded in the [.pwi](#) file when you save an inspection, and are reloaded when you next open it.

Aligning the CAD model to the part

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

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

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



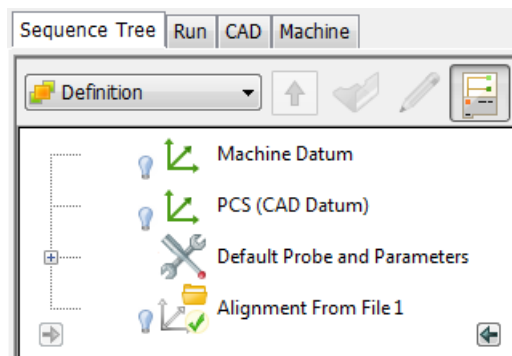
This shades the CAD model and makes it easier to see the surfaces and features you are working with.

Stage 1: Delete the default alignment

By default, PowerINSPECT OMV aligns the CAD model using an alignment saved to a file. In this example, however, we are going to create an alignment based on a plane, a line and a point (PLP), so we first need to delete this default alignment.

To delete the default alignment:

- 1 Select the **Sequence Tree** tab.
- 2 Select the default alignment in the inspection sequence.



- 3 Press the **Delete** key on the keyboard.

Stage 2: Create a geometric group

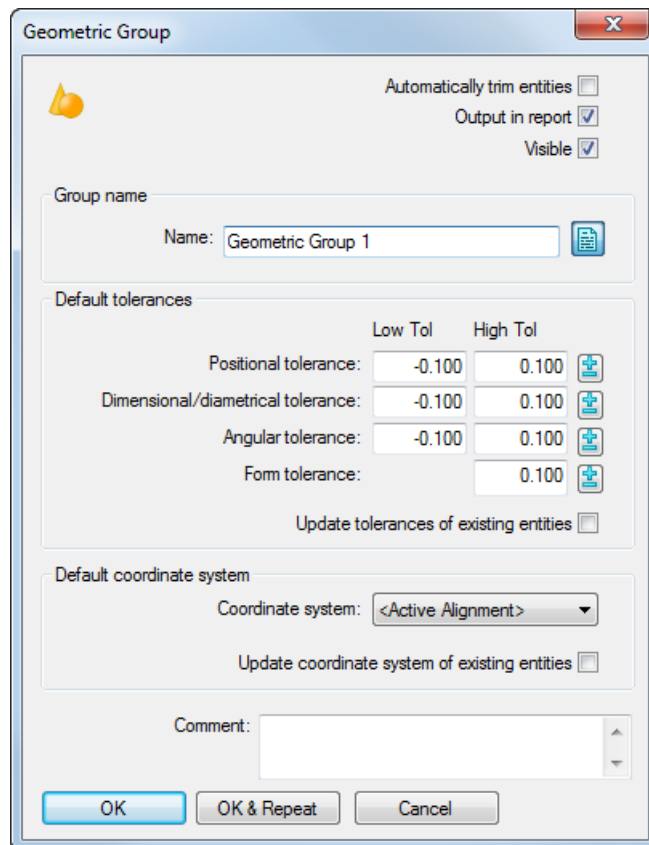
In PowerINSPECT OMV, geometric items are stored in geometric groups. These enable you to organize the inspection sequence, and to keep related items together. The first step in aligning a part to its model is to create a geometric group for the items used in the alignment.

To create a geometric group:

- 1 In the **Item** toolbar, click the **Geometric Group** button.



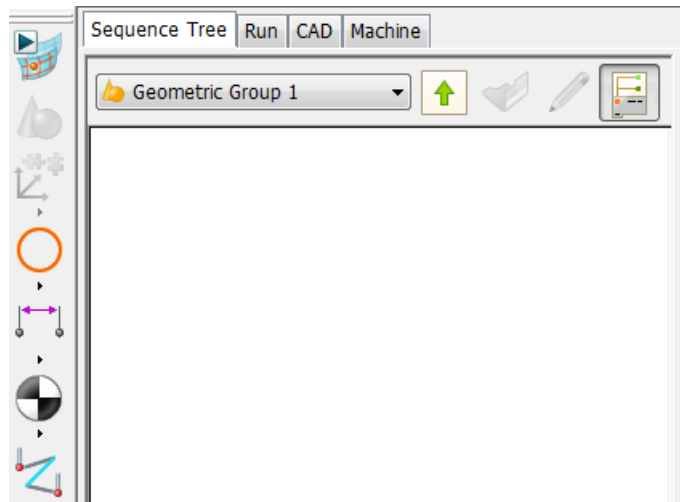
The **Geometric Group** dialog is displayed.



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

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

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



Stage 3: Create a probed plane

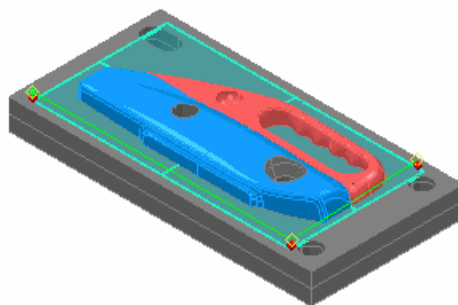
The first item required in a Geometric PLP Alignment is the plane. In this example, it is used to specify the orientation of the part's Z plane.

To create a probed plane:

- 1 In the **Geometry** toolbar, click the **Probed Plane** button.




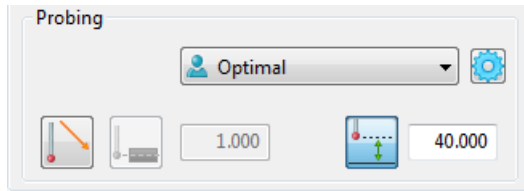
- 2 Double-click near three corners of the CAD model to define the plane.




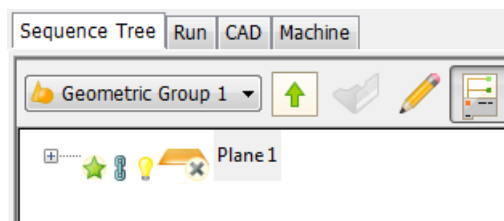
The details of the plane are displayed in the **Features** tab underneath the inspection sequence.

- 3 In the **Features** tab, make sure that the **UserDefined** probing method of **Optimal** is selected.


- 4 Click  and enter a **Safe Height** of **40**, so the probe lead-in and lead-out clears the middle of the model.



- 5 Click  to confirm the details of the plane. The plane is added to the group and displayed in the inspection sequence.



An icon  is shown on the item to indicate it is unmeasured.


- 6 If you want to hide the plane in the CAD view, click the bulb  icon in the inspection sequence. Click the icon again to redisplay the plane.

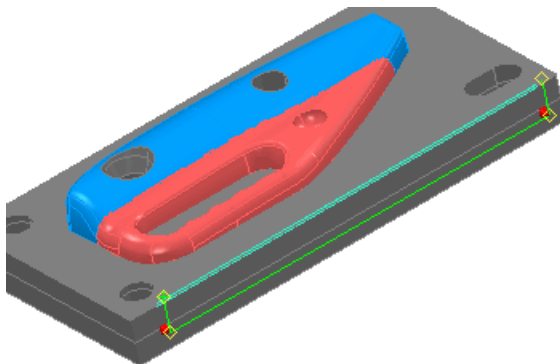
Stage 4: Create a probed line

The next item required for the alignment is a line. In this example, it is used to specify the orientation of the part's X plane.

To create a probed line:

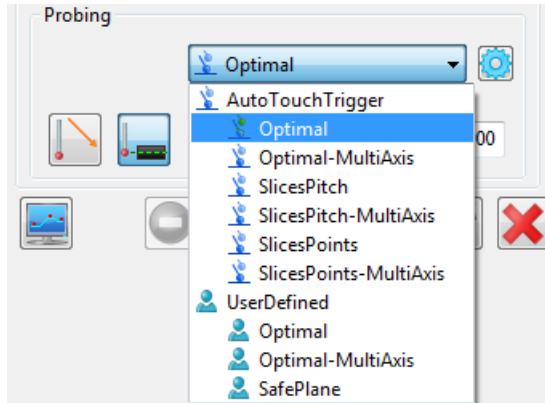
- 1 Rotate the model (see page 9) to display its back face.


- 2 In the **Geometry** toolbar, click the **Probed Line**  button and double-click near each end of the back face to define the line.

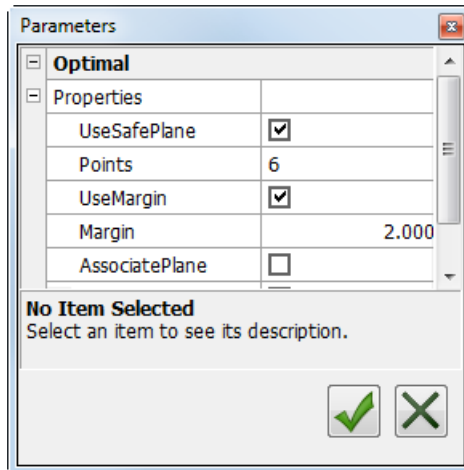



As before, the details of the line are displayed in the **Features** tab.

- 3 Select the **AutoTouchTrigger** probing method of **Optimal**. This specifies that you want PowerINSPECT OMV to automatically create the line's probe path.

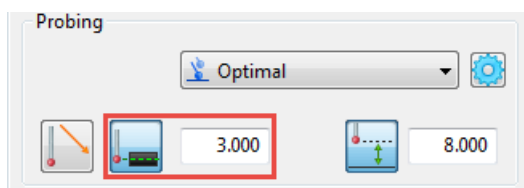


By default, the **AutoTouchTrigger** probing method of **Optimal** creates six probe points and spaces them equally along the line you created. Click  to display the **Parameters** dialog, then select the **Points** row and click the spin buttons to change the default number to **4**.




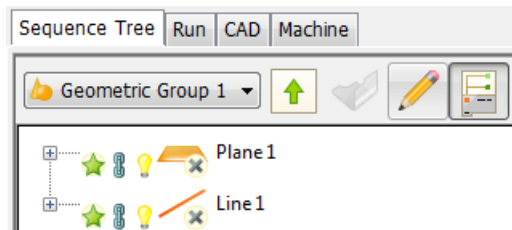
Click  to save your changes.

- 4 Click  and type **3** in the **Depth** box.



- 5 Click  and type **40** in the **Safe height** box.

- 6 Click  to confirm the details of the line.



Stage 5: Create a circle

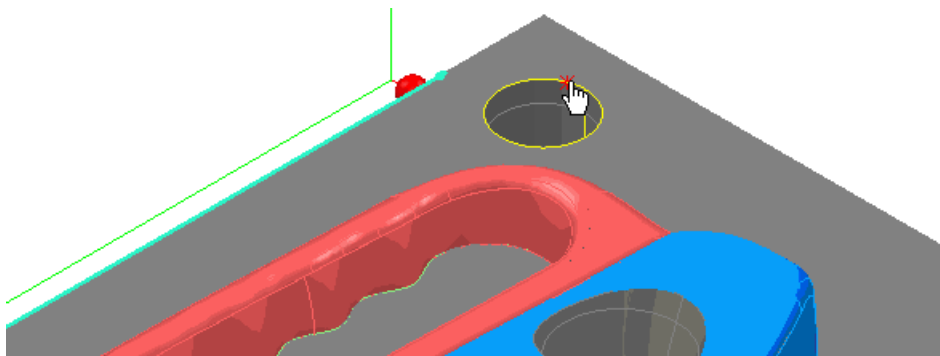
In addition to selecting the items you want to inspect and then specifying the probe points to be used, you can pick the features you want to inspect from the CAD model. This section demonstrates how to pick the circle item that used for the alignment.

To create a probed circle:

- 1 Select the **Wireframe Checker**  button in the **Mouse Context** toolbar.

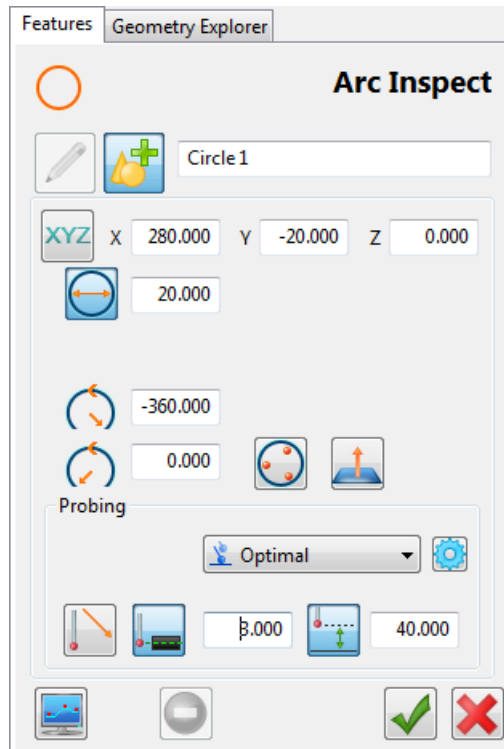



- 2 Move the mouse cursor over the CAD model. As you move the cursor, different features are highlighted in yellow.
- 3 Move the cursor over the circle shown below, and click the left mouse button to select it.



This creates a probe path, highlights the feature, and displays its details in the **Features** tab.

- 4 Click  and enter a **Depth** of **3**. Click  and enter a **Safe Height** of **40**.




- 5 Click  to confirm the details of the circle. You have now specified the items required to align the CAD model with the part.


Stage 6: Create a PLP alignment

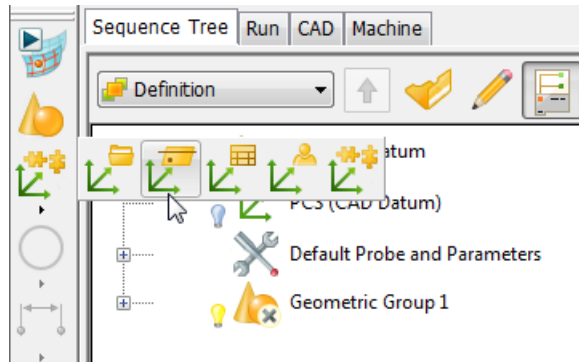
Geometric PLP alignments align a part to its model using a plane, line, and point for which you know the nominal coordinates. This type of alignment allows you to modify the alignment by amending the details of the geometric items or by remeasuring them individually.

To create the alignment:

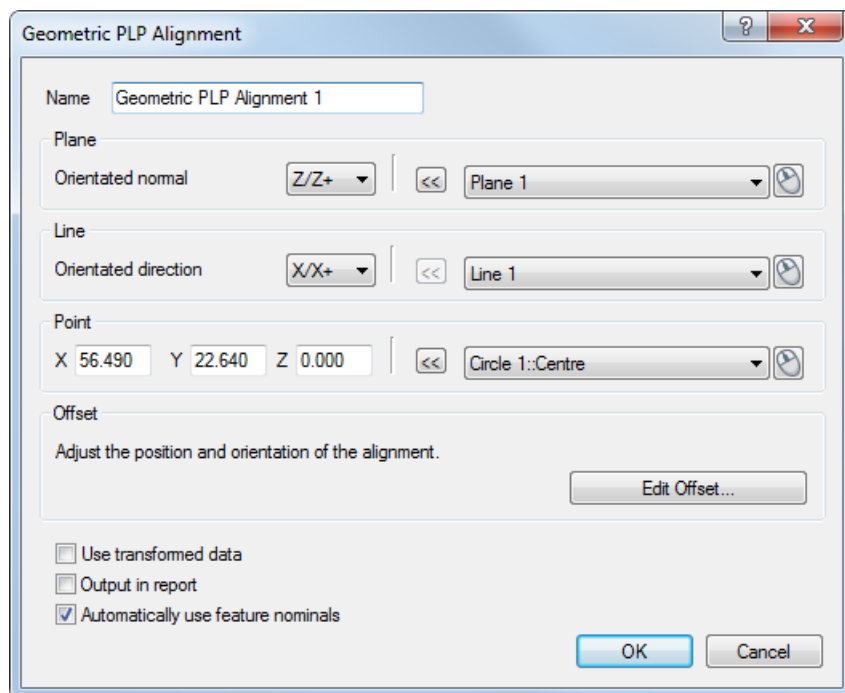
- 1 Click  at the top of the Sequence Tree to close the geometric group.

The **Active Group** list changes to *Definition* to show you have closed the group and are working with the top-level of the inspection sequence.

- 2 In the **Item** toolbar, click the **Geometric PLP**  button to create a PLP alignment based on the features you selected.



The **Geometric PLP Alignment** dialog is displayed. It automatically selects the items you defined and loads the nominals of the Point item.



- 3 Click **OK** to close the dialog and confirm you want to create the alignment using the default selections.









Working with probe paths

Probe paths control the movements and orientation of the probe while measuring a part. Before you export the inspection sequence to the machine tool for measurement, you must check the paths and correct any problems, such as collisions between the probe and the part. This section describes how to:

- Spot problems in the inspection sequence (see page 21).
- Simulate the inspection sequence (see page 23).
- Fix probe path faults (see page 24).

Verifying probe paths

By default, PowerINSPECT OMV checks the probe path of each item in the inspection sequence and displays one of the following icons to indicate its verification status:

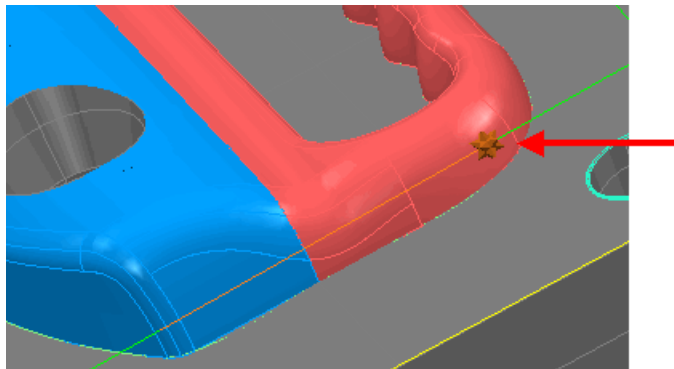
Icon	Description
	the probe path is unverified. This is displayed only when editing an item or simulating a probe path.
	the probe path contains no verification faults.
	the collision check for the path is invalid.
	the probe path contains a collision between the probe and the part. If a path contains both a collision and an out-of-bounds fault, only the out-of-bounds fault is displayed.
	the probe path contains a missing contact.
	the probe path contains an illegal contact.
	the probe path contains an out-of-bounds move.
	the probe or probe head specified for a probe path cannot be found.



Probe orientation changes and link moves between probe paths that are automatically created by PowerINSPECT OMV are not verified.

If any probe path faults are found, you must fix them (see page 24) before running the inspection sequence on a measuring device:

- You can view the location of an item's probe path faults by selecting the item in the inspection sequence. This displays its probe path and any faults in the CAD view. For example:



- You can view the details of all the faults for an item by selecting the item in the inspection sequence and then clicking the **Info** tab. For example:

Plane 1

Information

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

CNC Probe

Entity	Instruction	Status	Probe	Sensor	Orientation	Source
Plane 1	Sensor definition	OK	MP700_100x6_CE	M4_100x6_CE	A=0, B=0	Default Probe and Parameters

Probe Path Faults

Verification Status		Faults found
Fault Type	Start Point	End Point
Collision	(264.599, -40.262, 8.000)	(267.116, -103.140, 8.000)



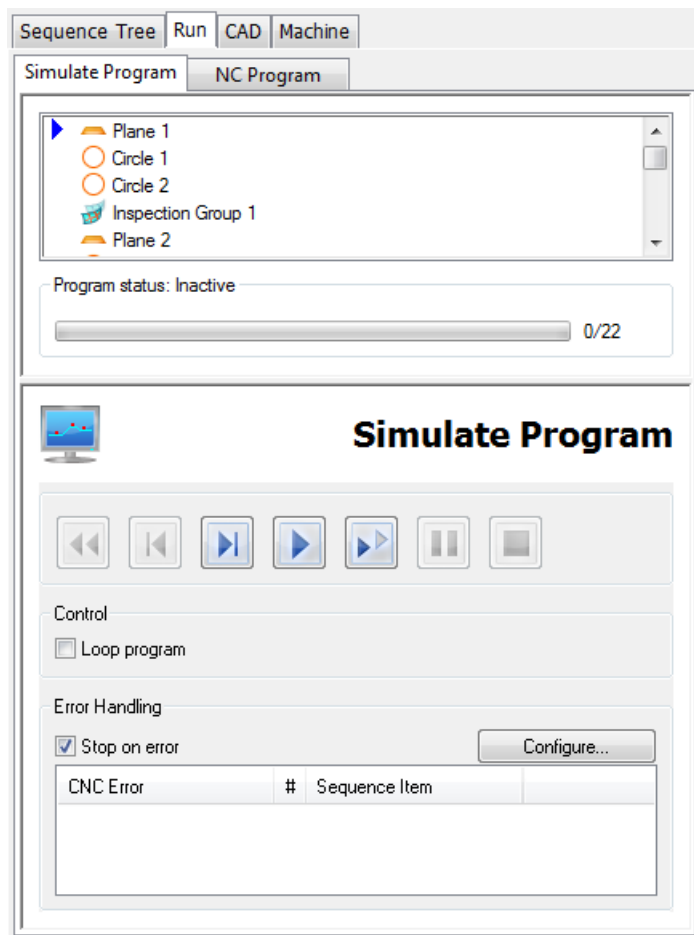
*Probe path verification is enabled by default. Select the **Tools** > **Probe Path Verification** menu option to deactivate it. Select the menu option again to reactivate it.*


Simulating measurement

The **Simulate Program** tab enables you to simulate all or part of the inspection sequence. Use it to visualize how the inspection and to check for problems in the probe path before running the sequence on the measuring device.

To simulate the inspection sequence:

- 1 At the top of the Sequence Tree, select the **Run** tab.
- 2 Select the **Simulate Program** tab.



- 3 If the probe is not visible in the CAD view, press the **F11** key to display it.
- 4 Deselect the **Stop on error** check box.
- 5 Click  to start the simulation in the CAD view.

As PowerINSPECT OMV simulates the inspection sequence, the movements of the measuring device are shown in the CAD view, and the program status and current feature are displayed in the Progress area at the top of the **Run** tab. Any problems with the probe path are displayed in the **Error handling** list. For more information, refer to the online help.

- 6 Use the other buttons on the **Simulate Program** tab to control the simulation. Click:



to simulate the inspection sequence, or selected range, one item at a time.



to simulate the inspection sequence, or selected range, one move at a time.



to pause or suspend the simulation. Click once to pause the simulation when measurement of the current feature is complete. Click twice to pause the simulation when the current move within the probe path is complete.



to stop the simulation immediately.



to rewind the simulation back to the start.



to step back one feature at a time.

Resolving verification faults

Use the details displayed about the fault to locate and resolve the cause of the fault. For example:

Fault	Resolution
Out of bounds move (yellow).	Reposition the part.
Collision with the part (orange).	Edit the item's probe path and try: <ul style="list-style-type: none"> deleting or moving touch points in the probe path. inserting intermediate points into the probe path.

Fault	Resolution
An unexpected unverified probe path (grey).	Check that: <ul style="list-style-type: none"> ▪ the probe used for the probe path is available. ▪ the probe supports the orientation specified in the probe path.
The probe is not displayed in the Simulate Program tab and most entries in the Features tab are not editable.	The probe specified for the selected item is not available in the probe database.

Adding further details to the inspection sequence


We'll now look at adding further details to the inspection sequence, including:

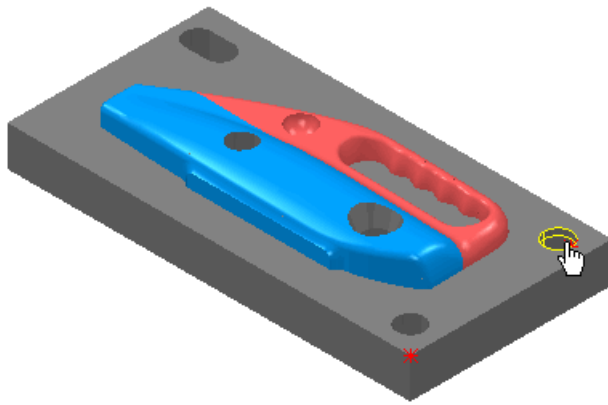
- a 3D feature (see page 27).
- a surface inspection group (see page 28).
- how to export only part of the inspection sequence (see page 50).

Creating a cylinder

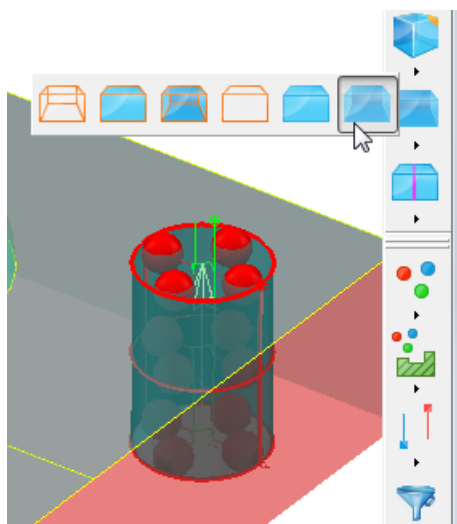
In addition to creating items from 2D features, the Wireframe Checker (see page 17) can also be used to create 3D items, such as cylinders. To do this, you must change the Checker's picking mode.

To create a cylinder item:

- 1 Select the **Features** tab.
- 2 Select the **Wireframe Checker**  button.
- 3 Right-click the CAD view and select **Surface** from the context menu. Left-click the CAD view to close the menu.
- 4 Move the cursor over the cylinder shown below, and left-click to select it.



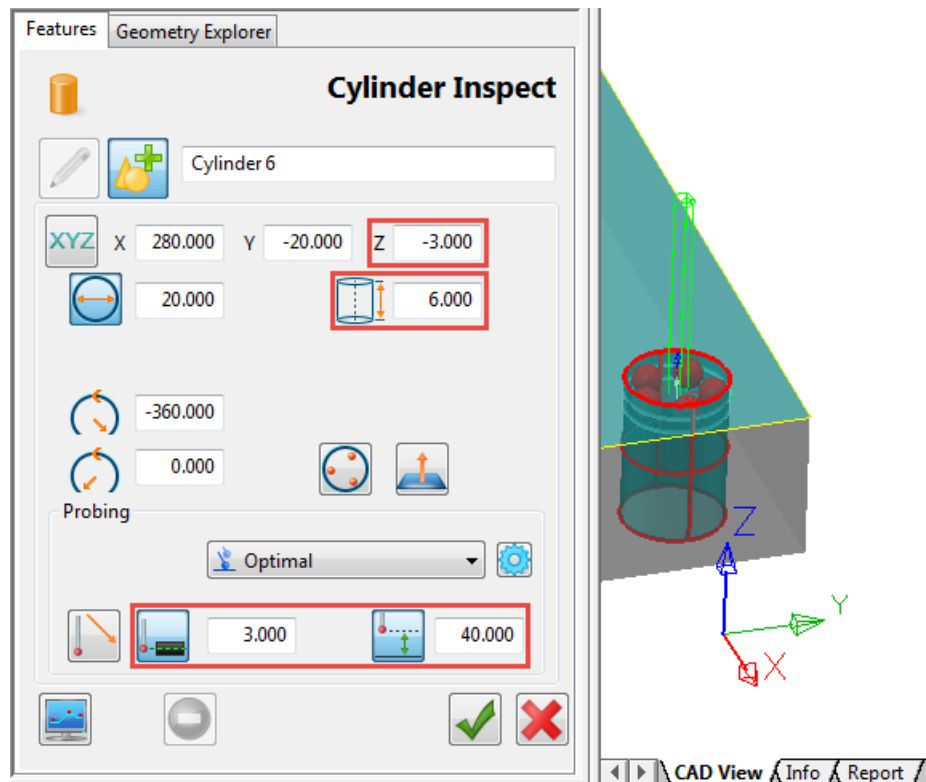
- 5 To see the whole cylinder, select the **Show Transparent** button in the **Shading** toolbar.




- 6 In the **Features** tab, enter the following values:
Z -3.0
Depth - 6 mm

Probing depth - 3 mm

Safe Height - 40 mm.





- 7 Click  to create the Cylinder item. Because items cannot be added to the Definition level of the inspection sequence, PowerINSPECT OMV automatically creates and opens a new geometric group to hold the cylinder item.

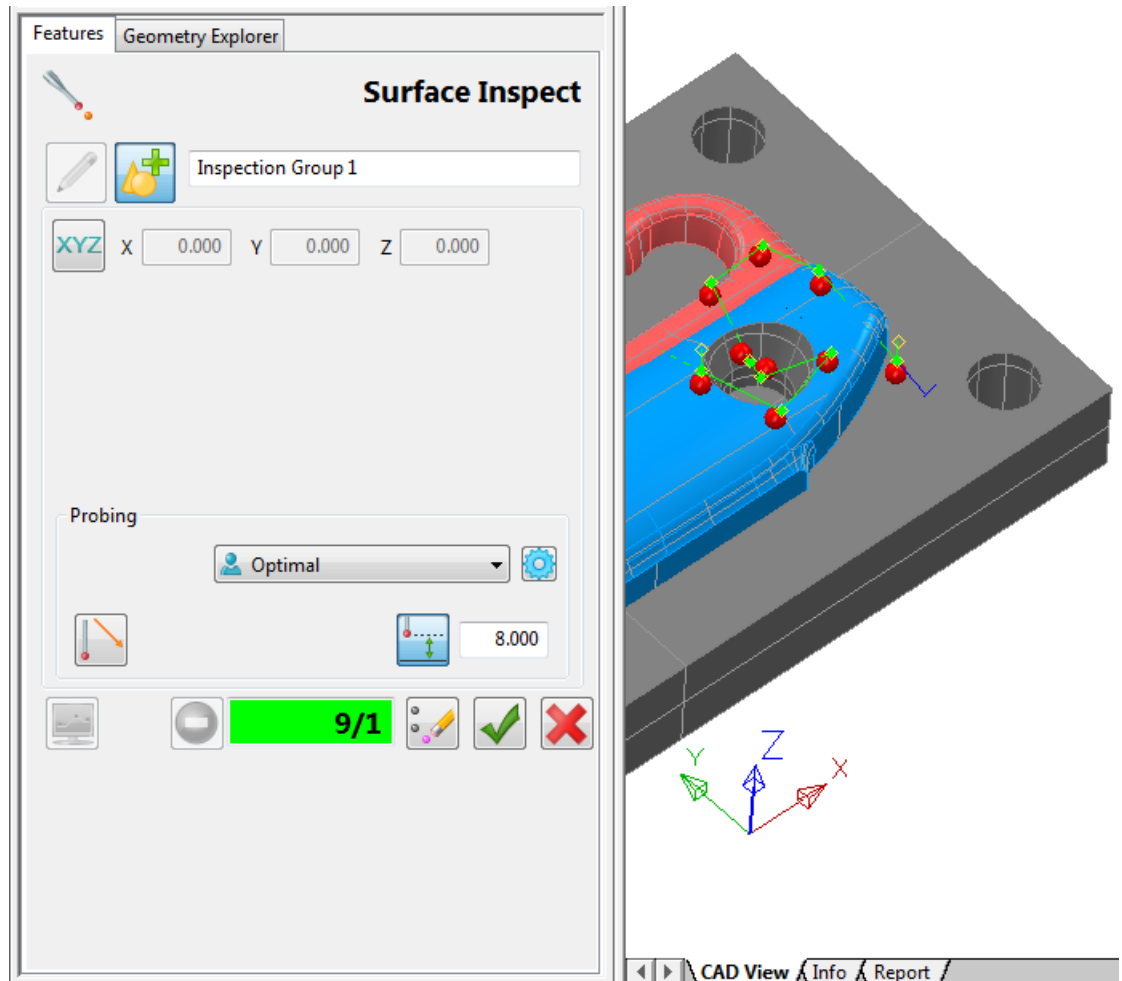
Creating a CNC surface inspection group

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

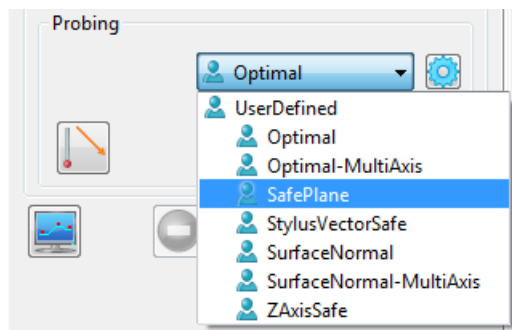
To create a surface inspection group:

- 1 In the Sequence Tree, click  to close the geometric group and display the Definition level of the inspection sequence.
- 2 In the **Item** toolbar, click  to create a CNC surface inspection group in the inspection sequence.

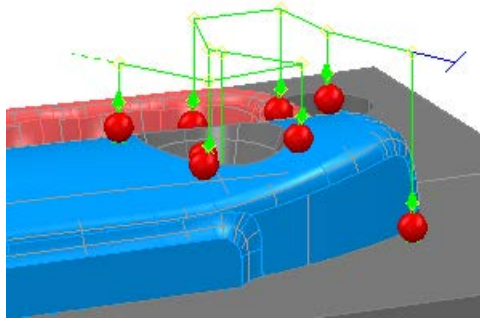
- 3 Double-click on a face of the CAD model to specify the points you want to inspect. Position the first point on the main face, then add points on the surface. For example:



- 4 In the **Features** tab, change the probing method from **Optimal** to **SafePlane**.



The path changes to move the probe to the safe plane between each point. For example:



- 5 Enter a **Safe Height** of **40** mm:



- 6 Click  to create the item.

Positioning the probe



Positioning the probe is an important part of creating an inspection sequence. It prevents damage to equipment, and also ensures the accuracy of your measurements. PowerINSPECT OMV enables you to control probe positioning using two methods:

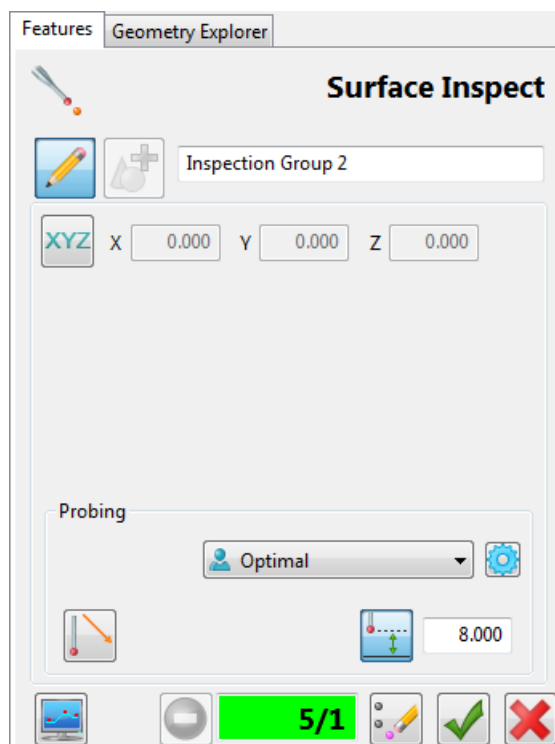
- Automatic orientation (see page 32).
- Orientation point items (see page 34).


Using automatic orientation


Automatic orientation positions the measuring device so that the probe is normal to the feature to be measured. Use it when you do not want to specify the probe orientation with which a feature is to be measured.

For example, to set the automatic orientation for an inspection group:

- 1 Select the **Sequence Tree** tab.
- 2 Select the **Features** tab.
- 3 In the inspection sequence, select the inspection group.
- 4 Click  to edit the item's settings.
- 5 Click the **Auto-Orientation**  button.
- 6 In the Probing method list, select the **UserDefined** probing method of **Optimal**.




- 7 Click the **Use probe safe parameter**  button to specify the safe height to be used when measuring the feature. For an **Optimal** method, this sets the approach distance before the first probed position, and the retract distance after the last probed position.

- 8 Click  to view the parameters that determine the probe path for the selected probing method, then close the **Parameters** dialog.



Only advanced users should change the probing parameters. If the standard probing methods included with PowerINSPECT OMV do not meet your needs, contact your sales representative to discuss your requirements.

- 9 Click the **Save geometric feature**  button to save your changes.

Orientation Point items

Use the Orientation Points item to control the position and rotation of the probe during multi-axis probing. Use it when you want to set or change the probe orientation in the inspection.



Orientation points are not available with all licenses.







To create an Orientation Point item:


- 1 In the **Item** toolbar, click . The **Orientation Point** dialog is


displayed in the **Features** tab, and the orientation point symbol is displayed in the CAD view at the CAD Datum.




*To hide the symbol in the CAD view, click the light bulb displayed next to the **Orientation Point** item in the inspection sequence.*



- 2 Complete the orientation point settings:
 - **Reference datum** — Select an entry in this list to specify the datum you want to use as the orientation point origin.
 - **Load** — Click  to load the orientation point settings from a previously saved file.
 - **Save** — Click  to save the current settings to a file.
 - **Align** — Click  to align the axes of the orientation point with the **Reference datum**.
 - **Reset** — Select  to move the orientation point to the origin of the **Reference datum**. Deselect  to leave the orientation point in its current position.
 - **Position** — Enter coordinates in the **X**, **Y** and **Z** boxes to specify the location of the orientation point relative to the **Reference datum**. The position of the orientation point in the CAD view is automatically updated when you select another box.
 - **Translate** — Use this box to change the location of the orientation point relative to the current **Position**. For example, to change the **X** position of the point by 10 mm, type **10** in the box, and click .


- **Rotate** — Use this box to rotate the orientation point relative to the **Reference datum**. For example, to rotate the orientation point by 45 degrees in the **Y** axis, type **45** in the box, and click .

- **Undo** — Click  to cancel the previous change.

- **Redo** — Click  to cancel the previous undo.

The orientation point symbol moves in the CAD view as you enter each change.

- 3 Click  to create the orientation point, or click  to cancel your changes.

- 4 Click  to simulate the orientation point. Alternatively, use the **Simulate Program** tab to simulate the orientation point in the inspection sequence.

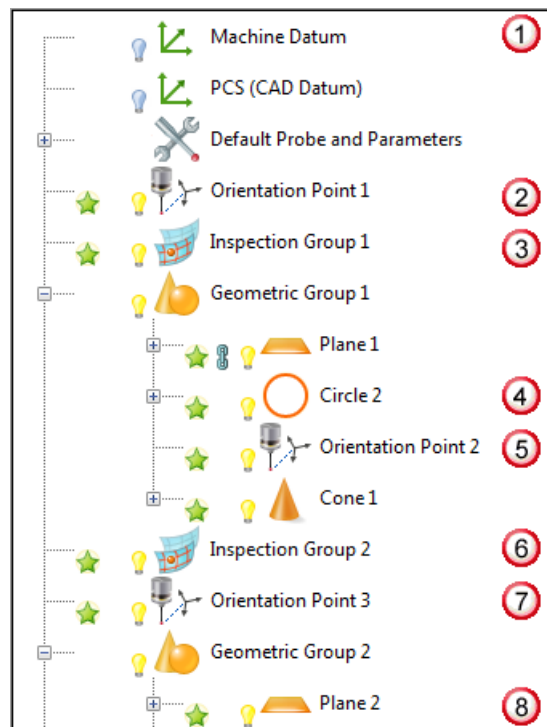



Simulated moves may not match the moves of the machine tool exactly.

Using orientation points

Orientation point items can be placed anywhere in the Definition level of the inspection sequence. They can also be used within a geometric group to orient the probe before or after any item.

The following example illustrates the use of Orientation Points in the inspection sequence:



- 1 The inspection sequence starts by using the default (Machine Datum) to position and orient the probe.
- 2 The probe is transformed by Orientation Point 1.
- 3 In Inspection Group 1, **Auto-Orientation** is selected for the group's UserDefined probing method, so the probe is transformed before probing begins.
 *If **Auto-Orientation** is not selected, the probe remains at the rotation specified in Orientation Point 1.*
- 4 In Circle 2, **Auto-Orientation** is not selected, so, before the first point on the Circle 2 probe path is played, the probe is transformed back to Orientation Point 1 because it is the preceding orientation point.
- 5 Orientation Point 2 transforms the probe for Cone 1.
- 6 Inspection Group 2 does not use **Auto-Orientation**, so the probe uses the orientation of Orientation Point 1 because it is the preceding orientation point item *at that level* in the inspection sequence.
- 7 When the probing of Inspection group 2 is complete, the probe is transformed by Orientation Point 3.

- 8 If Geometric Group 2 contains any item in which **Auto-Orientation** is selected, the probe is transformed by the associated probing method. Otherwise, the probe remains at the rotation specified in Orientation Point 3.

Measuring an inspection sequence

This section describes how to:

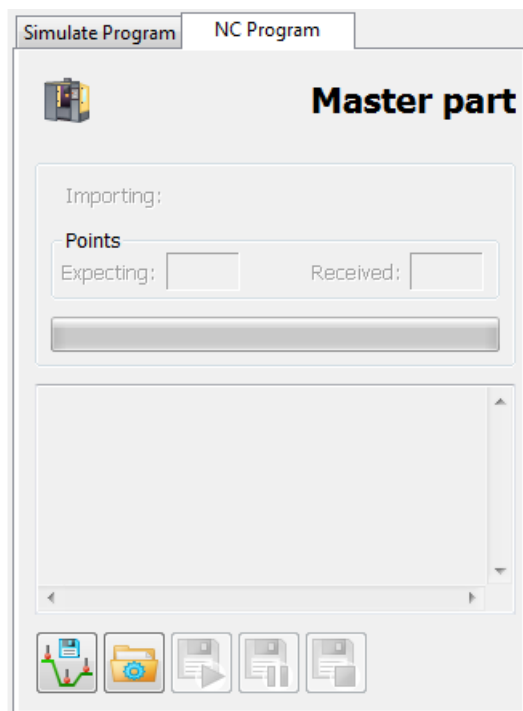
- export the inspection sequence to the machine tool (see page 39).
- run the NC program (see page 41).
- import data from the measured NC program (see page 42).


Exporting the inspection sequence to the machine tool

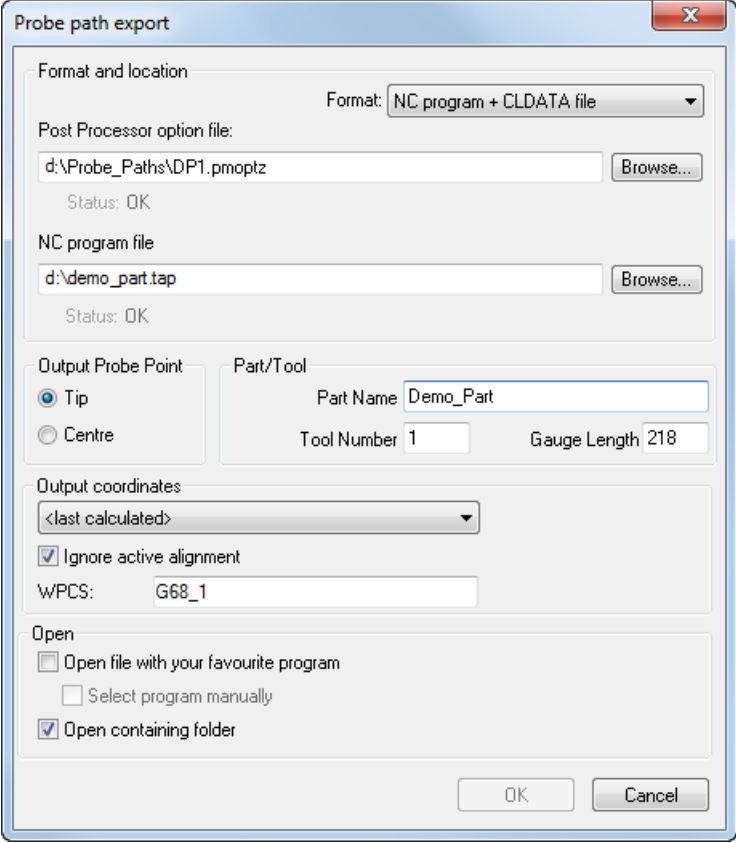
When you have checked the inspection sequence and resolved any faults in the probe path, you can convert it to an NC Program file and export the file to the machine tool.

To export the probe path:

- 1 Select the **NC Program** tab.



- 2 Click  to export the alignment features. PowerINSPECT OMV verifies the probe paths, and displays the **Probe Path Export** dialog.



The **Probe path export** dialog box contains the following fields and options:

- Format and location:**
 - Format: *NC program + CLDATA file* (dropdown)
 - Post Processor option file: *d:\Probe_Paths\DP1.pmoptz* (text box) with a **Browse...** button. Status: OK
 - NC program file: *d:\demo_part.tap* (text box) with a **Browse...** button. Status: OK
- Output Probe Point:** ☒ Tip, ☐ Centre
- Part/Tool:**
 - Part Name: *Demo_Part* (text box)
 - Tool Number: *1* (text box)
 - Gauge Length: *218* (text box)
- Output coordinates:** *<last calculated>* (dropdown)
- ☒ Ignore active alignment
- WPCS: *G68_1* (text box)
- Open:**
 - ☐ Open file with your favourite program
 - ☐ Select program manually
 - ☒ Open containing folder

Buttons: **OK**, **Cancel**

- 3 In the **Format** list, select *NC program + CLDATA file*.
- 4 Select the **Post Processor option file** for the machine tool on which you want to run the probe path.
- 5 In the **NC Program file** box, enter the location and name of the file in which you want to save the inspection sequence.
- 6 In the **Part Name** box, enter the name of the part to be measured.
- 7 In the **Tool Number** box, enter the tool number to be used when running the probe path. (This sets the tool number and the tool offset number.)
- 8 If your machine has rotary axes and no RTCP support, enter the distance from the probe tip to the gauge face of the spindle in the **Gauge Length** box.
- 9 In the **WPCS** box, enter the work piece coordinate system number that you want the machine tool to use when measuring the part.



On some machine tools, the WPCS is referred to as the work offset.

10 Click **OK** to create the NC program file.

You can now run the OMV inspection sequence on a machine tool (see page 38). When you have imported the point data into PowerINSPECT OMV, you can continue to add further details to the sequence.

Running the NC program

The PowerINSPECT OMV NC program (tape file) contains the probing moves specified in the inspection sequence. When you run it on a machine tool, it outputs the measurement results to a serial port or file, depending on the machine tool and the post-processor.

Before running the NC program on the machine tool, check:

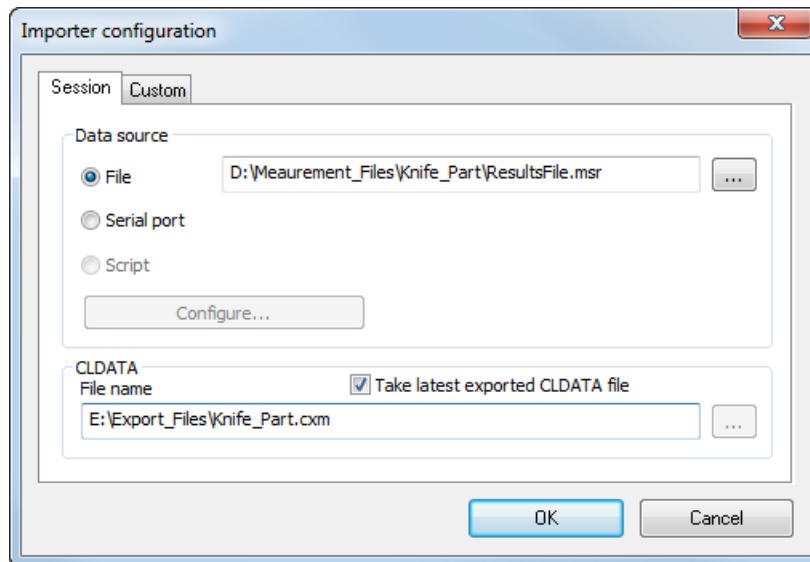
- the tool number in the NC program matches the number of the probe in the machine tool's magazine.
- the work offset on the machine tool matches the WPCS (workpiece coordinate system) in the NC program.
- import details have been specified in PowerINSPECT OMV (see page 42) if you are importing the data using a serial connection.


Importing measurements into PowerINSPECT OMV

To import measurement results into PowerINSPECT OMV:

- 1 Select the **Run** tab.
- 2 Select the **NC Program** tab.

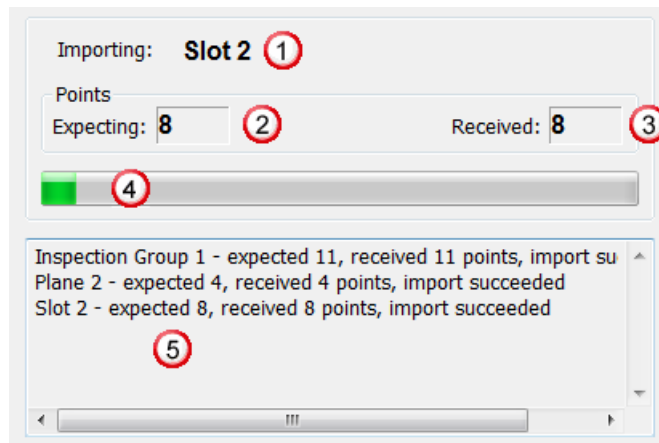
- 3 Click  to display the **Importer Configuration** dialog.



- 4 Select the **Custom** tab and enter the location and name of the default parser file, **reader_psfixture.rsfl**, in the **Script file name** box. The file is located in the **\OMVReaders** folder of your installation.
- 5 Select the **Session** tab.
- 6 Choose an import method. Select:
 - **File** to import the point data from the measurement file created by the machine-tool controller. Enter the location and name of the file you want to import in the adjacent box. Alternatively, click  to select the file.
 - **Serial port** to import the point data directly from the serial port connection of the machine-tool controller. When you first use this option, click **Configure** and ensure PowerINSPECT OMV's serial configuration settings match those required by the controller. For more information on the serial configuration settings required, refer to your controller's documentation.
- 7 Click **OK** to save your changes and close the dialog.

- 8 In the **NC Program** tab, click  to import the measurement results from the machine tool.

The **NC Program** tab displays the details of the import as it progresses.



Importing: **Slot 2** ①

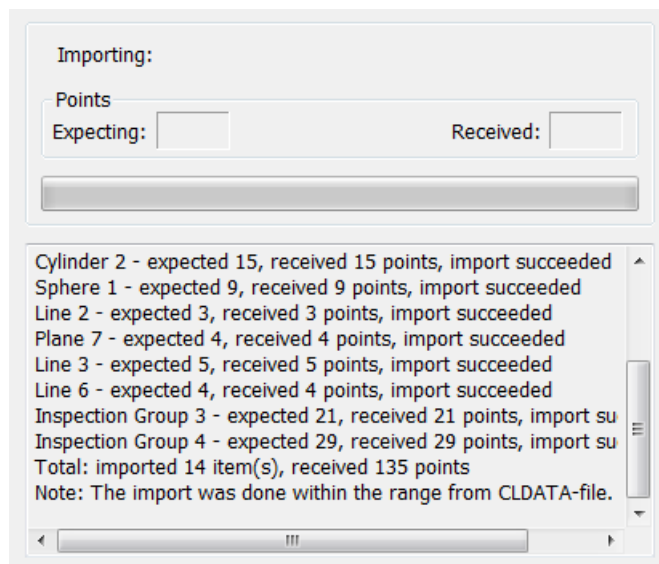
Points
Expecting: 8 ② Received: 8 ③

④

Inspection Group 1 - expected 11, received 11 points, import su
Plane 2 - expected 4, received 4 points, import succeeded
Slot 2 - expected 8, received 8 points, import succeeded
⑤

Where:

- ① shows the name of the inspection sequence item being imported.
- ② shows the minimum number of points required to measure the item.
- ③ shows the number of points received for the item.
- ④ shows the progress for the current item.
- ⑤ lists the details of the imported items. When the import is complete, a summary is displayed, for example:



Importing:

Points
Expecting: Received:

Cylinder 2 - expected 15, received 15 points, import succeeded
Sphere 1 - expected 9, received 9 points, import succeeded
Line 2 - expected 3, received 3 points, import succeeded
Plane 7 - expected 4, received 4 points, import succeeded
Line 3 - expected 5, received 5 points, import succeeded
Line 6 - expected 4, received 4 points, import succeeded
Inspection Group 3 - expected 21, received 21 points, import su
Inspection Group 4 - expected 29, received 29 points, import su
Total: imported 14 item(s), received 135 points
Note: The import was done within the range from CLDATA-file.

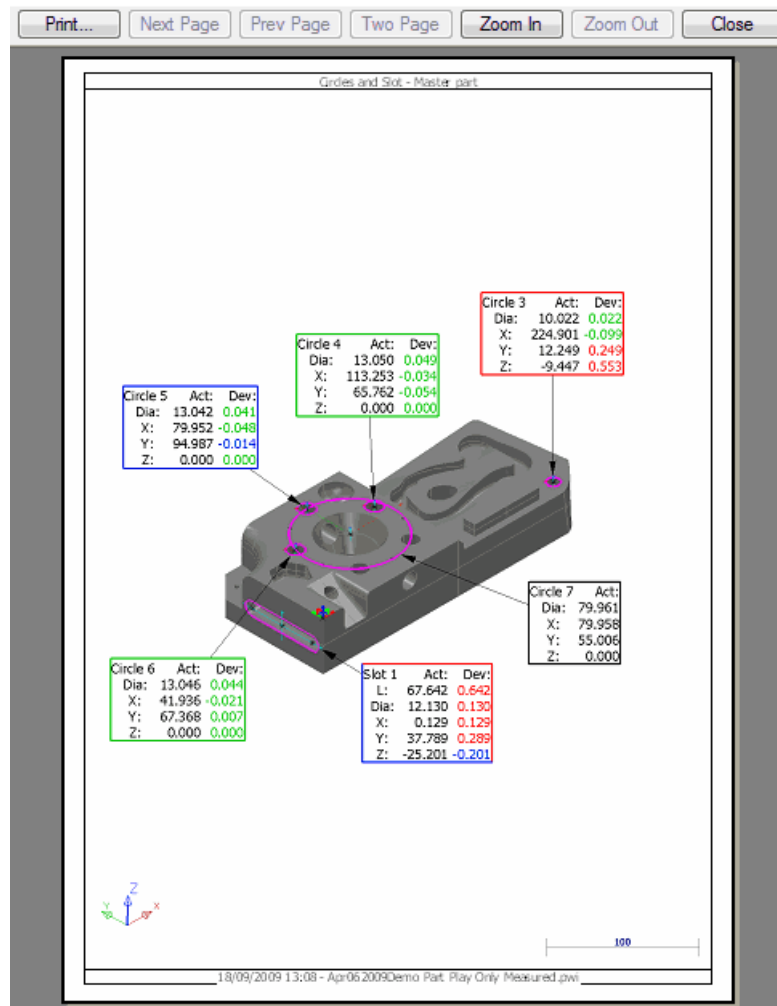
Displaying the measurement results

There are several ways to view measurement results in PowerINSPECT OMV. This section describes how to display them in:

- An on-screen report (see page 45).
- An HTML report (see page 46).


Creating an on-screen report


The Print Preview option enables you to quickly print the information displayed in the current view.




To create an on-screen report of the CAD view:

- 1 Select the **CAD view** tab.
- 2 Position and size the model in the CAD view to display the features you want to include in the report.
- 3 Select the information you want to display. For example, to

display the feature information in labels, click  in the **View Options** toolbar.

- 4 If you want to hide the labels of any of the items, click the show  icon on the item's entry in the inspection sequence. If you want to change the information displayed in each label, click




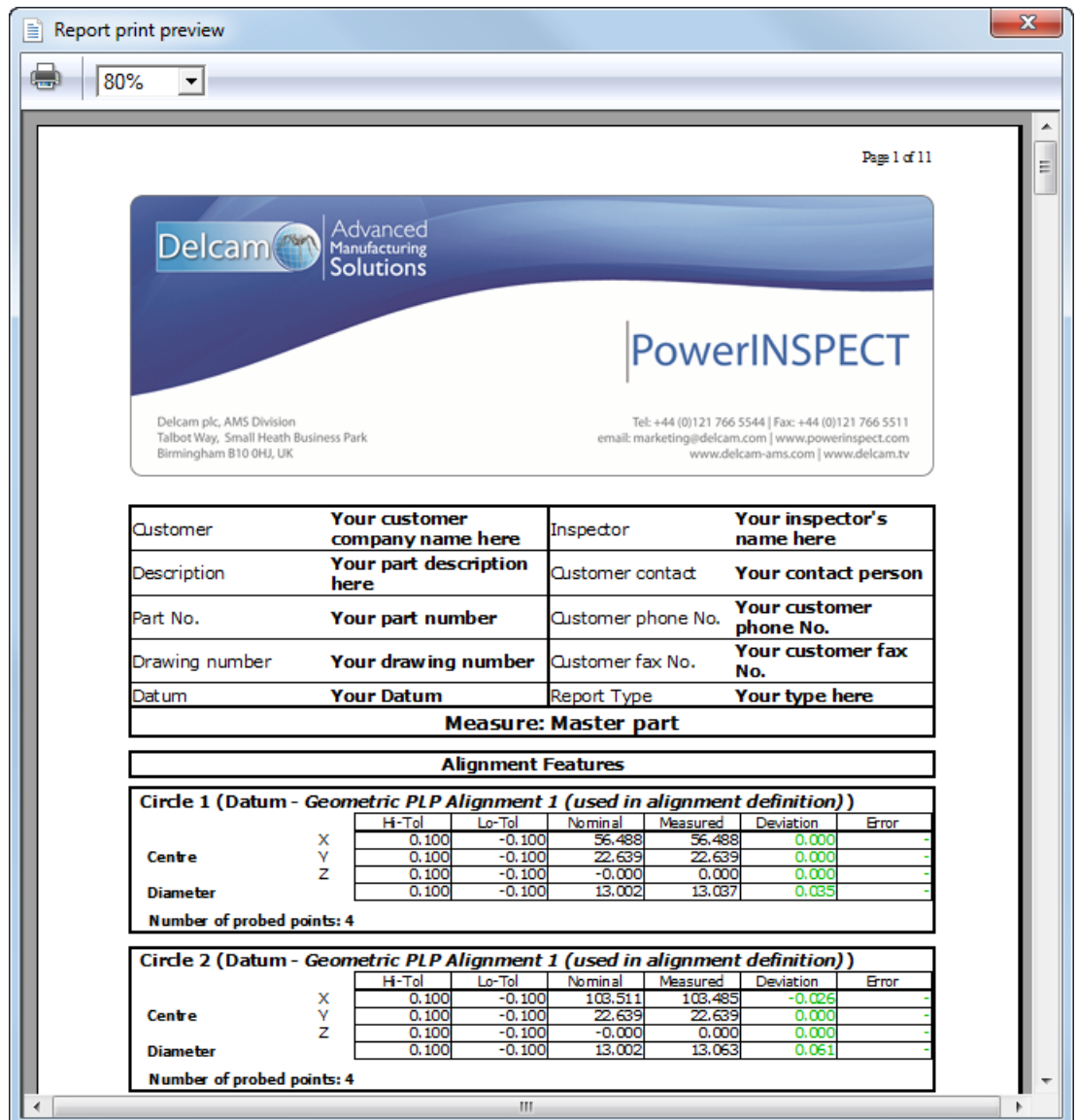
- 5 With a display format selected, click  to preview the report.
- 6 If you want to change the details shown in the report, click **Close** and adjust the CAD view, otherwise, click **Print** to print the on-screen report.

Creating an HTML report

HTML reports provide you with detailed numerical and graphical information on the measurements in the current document. You can customize reports for specific jobs or specific customers.

To create an HTML report:

- 1 Click the **Report** tab. PowerINSPECT OMV generates the report.
- 2 Click  to preview the report.



The screenshot shows a 'Report print preview' window with a zoom level of 80%. The report is titled 'PowerINSPECT' and includes the Delcam logo and 'Advanced Manufacturing Solutions' text. It contains contact information for Delcam plc, AMS Division, and a table for customer and inspector details. The report also includes two sections for alignment features, each with a table of measurements and deviations.

Page 1 of 11

Delcam Advanced Manufacturing Solutions

PowerINSPECT

Delcam plc, AMS Division
Talbot Way, Small Heath Business Park
Birmingham B10 0HJ, UK

Tel: +44 (0)121 766 5544 | Fax: +44 (0)121 766 5511
email: marketing@delcam.com | www.powerinspect.com
www.delcam-ams.com | www.delcam.tv

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

Measure: Master part

Alignment Features

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



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

Number of probed points: 4

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

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

Number of probed points: 4

- 3 Click  to print the report.
- 4 If you want to export the report for viewing in a browser, click the **Export Report**  button to save the report in .mht format.

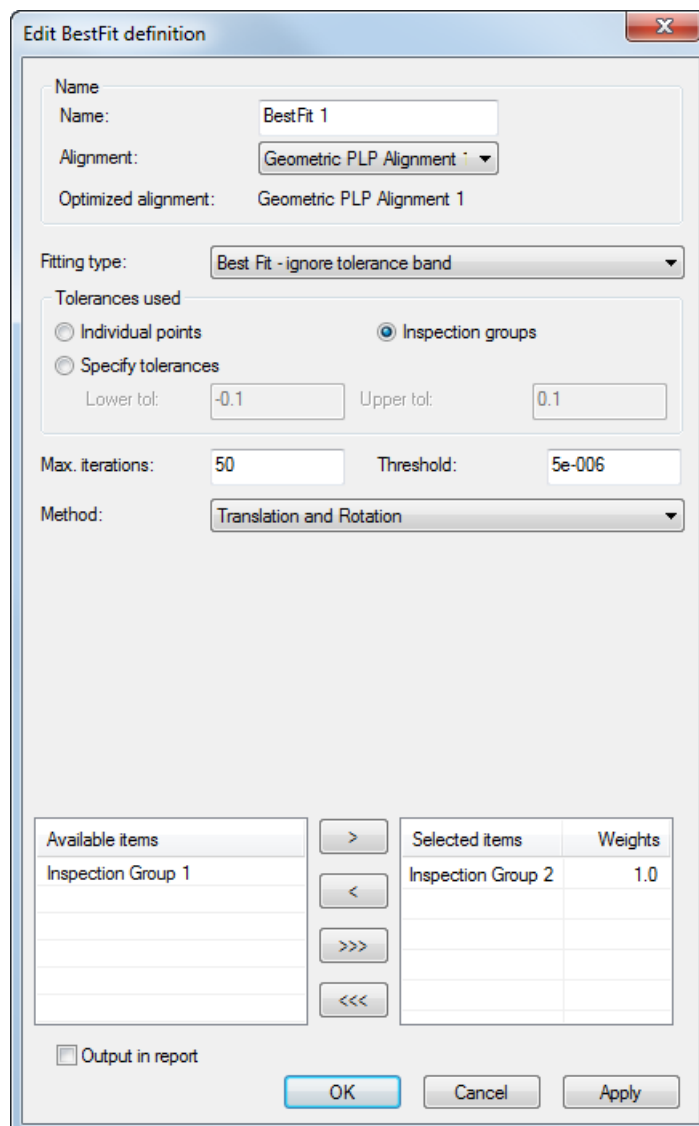
Optimizing the alignment (Best Fit)

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

To optimize an alignment:



- 1 Open the file [BestFitExample.pwi](#) from the [Samples](#) folder.

- 2 Click  to activate the **Best Fit** option.



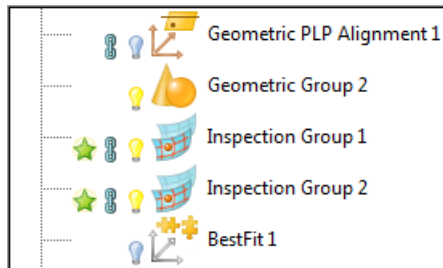
Available items	>	<	>>>	<<<	Selected items	Weights
Inspection Group 1					Inspection Group 2	1.0

- 3 In the **Fitting type** list, select **Best Fit - ignore tolerance band** to minimize the deviations of all the points.

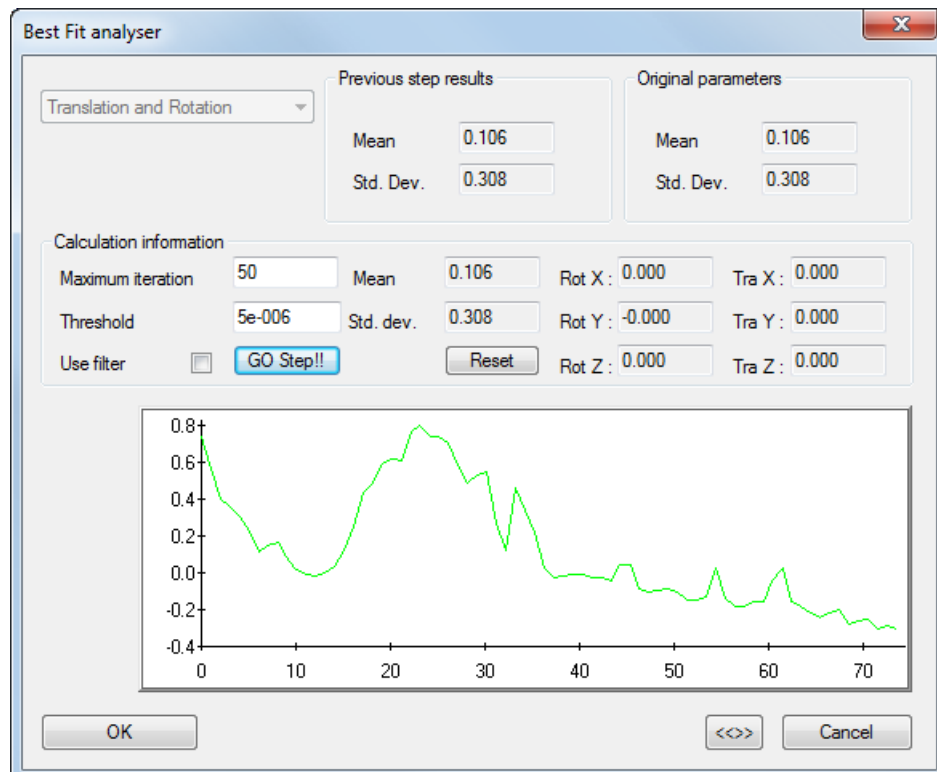
- 4 In the **Available items** list, select the item you want to use to optimize the alignment and click  to move it to the **Selected items** list. To move all the items to the **Selected items** list, click .

By default, all items in the **Selected items** list have an equal effect on the best fit calculation. If you want to assign greater emphasis to the measurements in specific items, click the **Weights** cell for the item, then type a multiplier in the entry box.

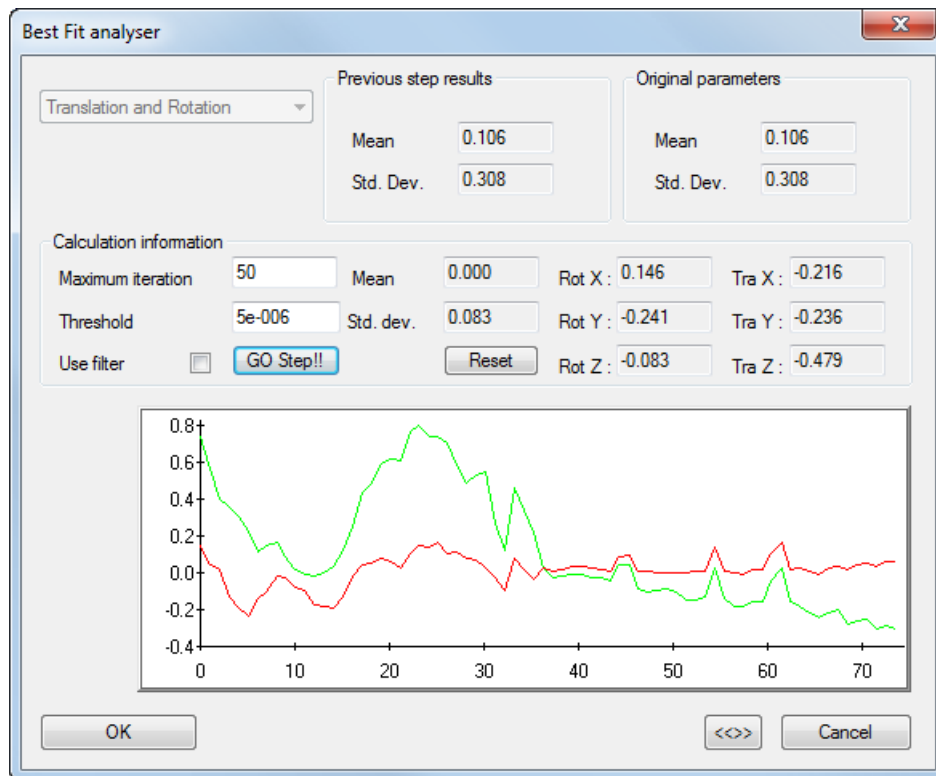
- 5 Click **Apply**. A Best Fit item is added to the inspection sequence.



- 6 Right-click the Best Fit item and select **Play**. The **Best Fit Analyser** dialog is displayed.



7 Click **Go Step** to activate the Best Fit.



A graphical representation of the deviations is shown: the green line shows the deviation prior to the Best Fit calculation, and the red line shows the deviation following the calculation. This updates all previously measured results.



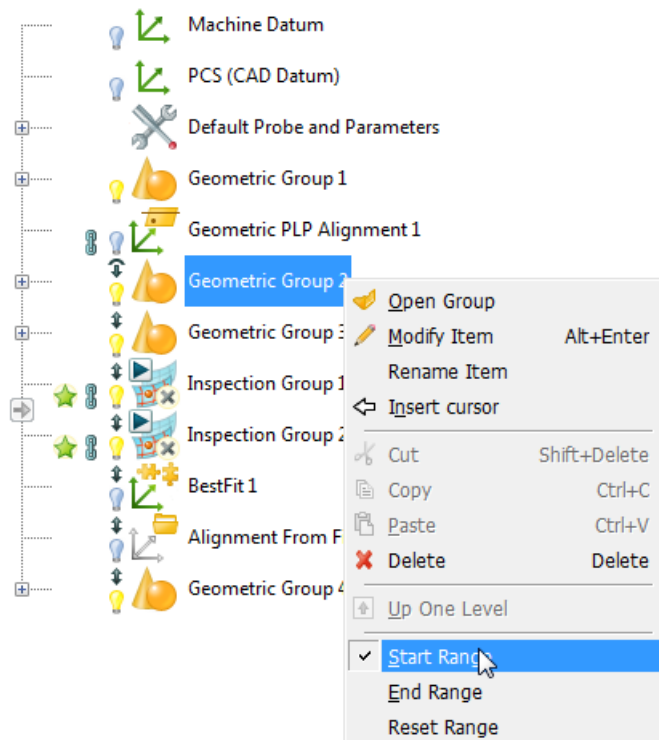
If a best fit is applied, reports should only be generated after this is complete.

Exporting a section of the inspection sequence

The range commands enable you to run only part of the inspection sequence. Ranges apply to the simulation and the NC program export, so you can use them when you only want to rerun changes made to previously measured sequence items.

To export only Geometric Group 2 and the following inspection group:

- 1 Select **Geometric Group 2** from the inspection sequence, right click and select **Start of Range** from the context menu.



Range indicators are displayed in the inspection sequence to show the items that are included. By default, the range extends to the last item in the sequence. If you do not want to run the inspection sequence to the end, you can specify the last item in the range using the **End of Range** menu option.

- 2 Export the NC program (see page 39) as you did earlier. This time, only the selected range is exported.

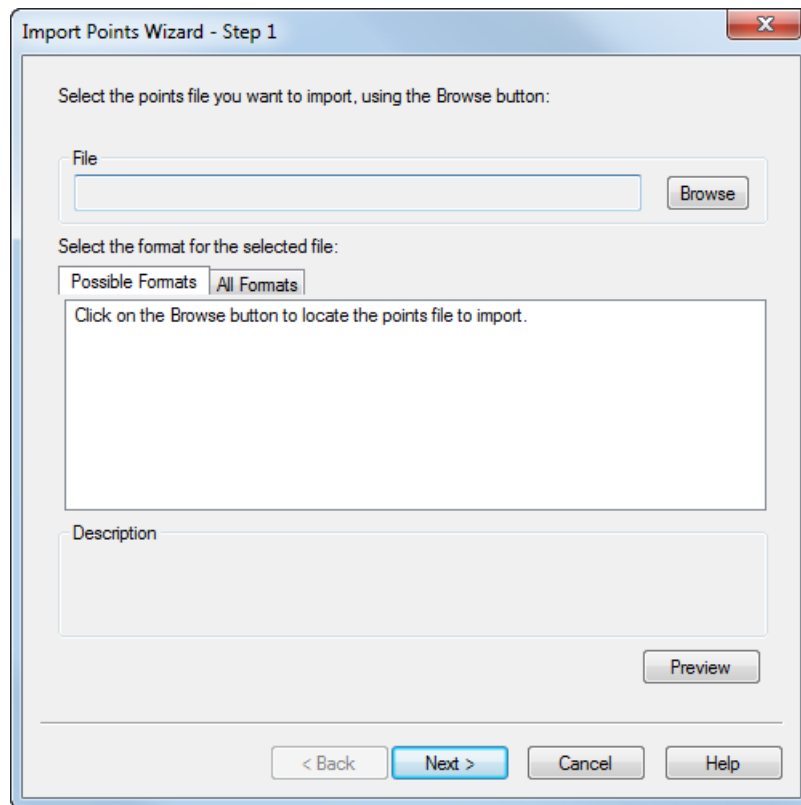
Creating a probe path from imported points

The **Tools > Import Points** menu options enable you to create a probe path from a text file containing coordinates of nominal points, such as a comma separated file of coordinates in an **XYZIJK** format.

To create a probe path from imported points:

- 1 Open a document that contains the model associated with the file to be imported,
- 2 Select:
 - **Tools > Import Points > Create Surface Inspection Group** to create a surface inspection group from the file's points.
 - **Tools > Import Points > Create Geometric Points** to create a geometric group of single points from the file's points.

The **Import Points** wizard is displayed.

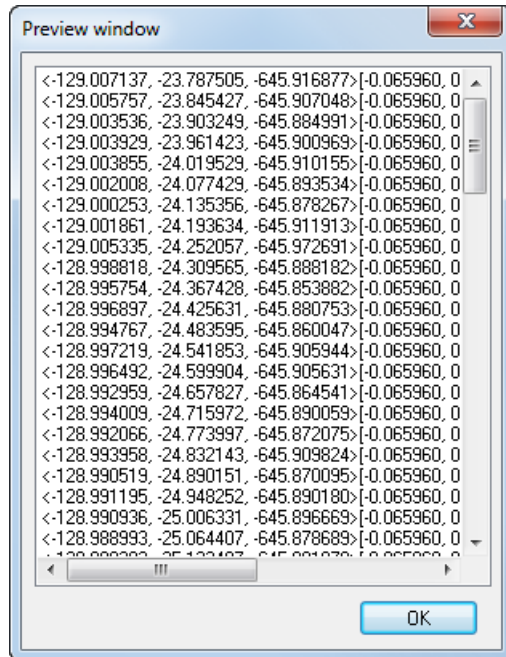


- 3 Click **Browse**. In the **Open** dialog, locate the points file you want to import.

If PowerINSPECT OMV recognizes the file extension, it displays descriptions of the file types under the **Possible Formats** tab.

- 4 Select the required format. If no format is displayed in the **Possible Formats** box, click the **All Formats** tab and select the format from the list displayed.

- 5 If you want to preview how the selected format interprets the point coordinates in the file, click **Preview**. For example:



Click **OK** to close the window.

- 6 When you have selected the file format, click **Next** to view a summary of the import options:
- 7 If you are satisfied with this summary, click **Next** to proceed. Otherwise, verify the options using the **Preview** button before proceeding. If you want to change the file format, click **Back** and proceed from step 4.

PowerINSPECT OMV starts importing the points into a new group.

When the import is complete, PowerINSPECT OMV displays the points in the CAD view and a message confirms the creation of the group in the inspection sequence, and the number of points used to create it. The probe path is displayed when you close the wizard.

For an example of the creation of a surface inspection group from imported points, refer to **Tools > Import Points > Create Surface Inspection Group** in PowerINSPECT OMV's online help. For an example of the creation of a geometric group, refer to **Tools > Import Points > Create Geometric Points**.