
PowerINSPECT CNC 2016

What's New



PowerINSPECT CNC

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New features

PowerINSPECT CNC 2016 contains the following new features and enhancements:

- **Working with CAD models** (see page 3) — Several improvements have been made to the way you load, manage, and view the details of models.
- **Automatic collision avoidance mode** (see page 8) — A new **Tools** menu option enables you to allow PowerINSPECT CNC to automatically manage all collision-free probe paths between features.
- **Changes to inspection groups** (see page 9) — To improve usability, two new types of inspection group have been created for measuring with probe paths. In addition, On-the-fly Surface Points and Guided Edge Points groups no longer support probe paths.
- **Specifying nominals** (see page 13) — You can now specify the nominal vector for Probed Parallel Plane, Plane: Parallel + Distance, Plane: Parallel Through Point, and Plane: Point + Vector items.
- **Selecting ellipses** (see page 15) — Ellipses are now selectable using the Wireframe Checker. This enables you to view their details, and to create items with the Geometry Explorer.
- **Report Note items** (see page 17) — The **Miscellaneous** toolbar now includes a Report Note item, which enables you to add customized text and images to the **Report** tab.
- **Section enhancements** (see page 21) — Several improvements have been made to the way you create and view sections.
- **Rack support enhancements** (see page 37) — The **Rack Locate** dialog now displays more information about the rack's position when you locate it on the measuring device.
- **Editing probe databases** (see page 41) — You can now edit the probe tool directly as you probe geometric items manually.

- **Label enhancements** (see page 42) — Improvements have been made to the formatting of labels, and you can now display labels for GD&T items.
- **Point cloud enhancements** (see page 48) — Several improvements have been made to PowerINSPECT CNC's point cloud support.
- **Displaying dual-column probe details** (see page 56) — If you have the dual-column cost option, you can now switch the probe displayed during the manual measurement of items.
- **PowerINSPECT CNC settings** (see page 58) — The **Options** dialog contains several new features, including, new colour options, and the ability to specify whether PowerINSPECT CNC uses exact or rounded error values to determine if measured items are within tolerance.

Working with CAD models

Several improvements have been made to the way you load, manage, and view the details of models:

- You can now load models by dragging and dropping files into the main graphics window (see page 4).
- A new button in the **CAD** tab enables you to highlight the selected entry in the CAD view (see page 5).
- New icons enable you to identify CAD objects by type (see page 6).

Loading CAD models


You can now load models into PowerINSPECT CNC by dragging and dropping files into the main graphics window.

To load a model into PowerINSPECT CNC:

- 1 In Windows Explorer, locate the model file.
- 2 Left-click and hold the file to select it.



*To select multiple models, press and hold the **Ctrl** key while searching the files.*

- 3 Drag the file into the main PowerINSPECT CNC graphics window.
- 4 When the cursor changes to , release the mouse button. If no documents are open, PowerINSPECT CNC creates a document and adds the model to it.


In addition, when you load a model into an inspection document that is not connected to the measuring device, PowerINSPECT CNC automatically resolves any overlap between the simulated probe and the model by repositioning the probe.








*To hide the probe, press the **F11** key; to redisplay the probe, press the key again.*

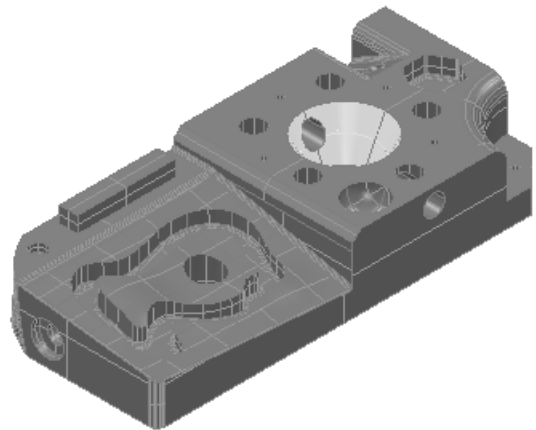
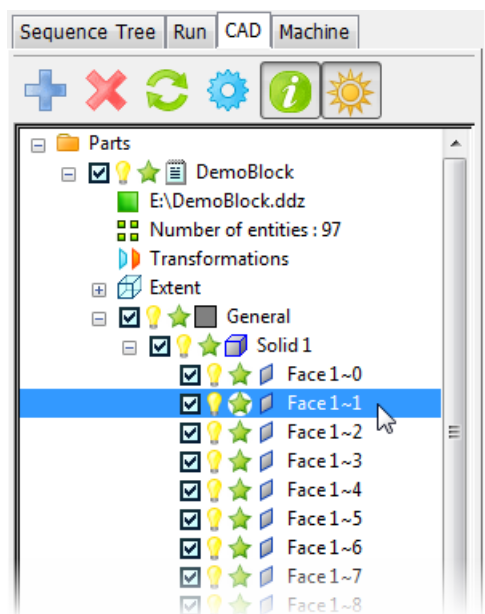
Highlighting objects in CAD models

Previously, when you selected a file, level, or object in the **CAD** tab, it was automatically highlighted in the CAD view to enable you to identify it. The colour of the highlighting was the same as that used for the Surface Selector. In PowerINSPECT CNC 2016, a new

Highlight CAD  button enables you to choose whether the highlighting occurs when you select an entry.

To highlight objects in CAD models:










- 1 Select the **CAD** tab.
- 2 Select the **Highlight CAD**  button.
- 3 Select the **Summary/Detailed View**  button.
- 4 Click  to expand the **Parts**  and **User Levels**  entries, and then select a file, object, or level. The entry is highlighted in the CAD view using a lighter shade of its original colour, for example:





If the original colour is white or a colour near white, the selected entry is highlighted in pink.

Identifying CAD objects by type

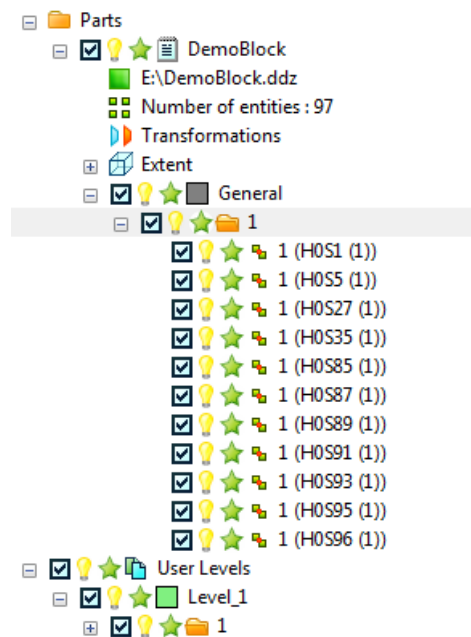
In PowerINSPECT CNC 2016, the Detailed View of the **CAD** tab, the **CAD Object Visibility** dialog, the **CAD Object Selection** dialog, and the **CAD Object Verification** dialog display an icon next to each constituent part of a model. This makes it easier to identify the type of objects you select and work with.

Icon	Object type
	Face
	Surface
	Wireframe
	Point
	Triangle Block
	GD&T Datum
	GD&T Tolerance
	Text
	Annotation Groups

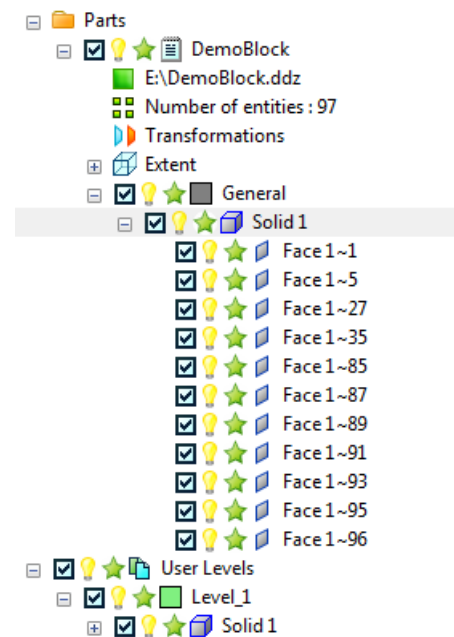
Similarly, parent objects are now distinguished as Groups  or Solids , instead of being marked by folder  icons.

The following example shows how CAD objects were displayed in the **CAD** tab in the previous version of PowerINSPECT CNC, and how the change has been implemented in the latest release:

PowerINSPECT CNC 2015 R2





PowerINSPECT CNC 2016



Automatic collision avoidance mode

An **Automatic Collision Avoidance Mode** option has been added to the **Tools** menu. Select it when you want PowerINSPECT CNC to automatically manage collision-free probe paths between features.

To enable PowerINSPECT CNC to manage collision-free probe paths automatically:

- 1 Select the **Tools > Automatic Collision Avoidance Mode** menu option.
- 2 In the confirmation message, click **Yes**. PowerINSPECT CNC:
 - creates any required Collision Free Link Path items and deletes any unnecessary ones as you create, delete, or move sequence items.
 - creates a Collision Free Link Path item if you delete one that is required.
 - prevents you from creating Collision Free Link Path items or moving existing ones.
 - creates an invalid  Collision Free Link Path item without creating a probe path, and inserts it immediately before a sequence item that uses a change of probe tool or probe angle. PowerINSPECT CNC does this as a warning to enable you to identify potential problems in the probe paths.
 - creates an invalid  Collision Free Link Path item when the probe path between two features contains an unavoidable collision with the part.



To disable the mode, deselect the **Tools > Automatic Collision Avoidance Mode** menu option, and then click **Yes** in the confirmation message.



This menu option has no effect on Intermediate Path items.

Changes to inspection groups

To improve usability, two new types of inspection group have been added to the **Inspection Groups** toolbar for measuring with probe paths:

-  **CNC Surface Points** groups (see page 10).
-  **CNC Edge Points** groups.

As a result, probe path support for On-the-fly Surface Points groups and Guided Edge Points groups is no longer supported; however, you can continue to create these types of group in manual mode.

The CNC Surface Points group combines the ability to specify probe paths with the ability to create guided surface points. This enables you to automate the inspection of surfaces on a part that can be repeated across Measures. You can also use the guided surface points as constraining features in RPS alignments.

For consistency, the ability to specify the probe path for a Guided Edge Points group has been transferred to the new CNC Edge Points group, which works in the same way. Use the new group when you want to automate the inspection of surface edges.



When you open a document in PowerINSPECT CNC 2016 that contains a Guided Edge Points group with a specified probe path, the group is automatically converted to the new CNC Edge Points group.

Creating inspection points from dynamic points

As in previous versions of PowerINSPECT CNC, you can continue to create guided surface and guided edge points from dynamic points. In PowerINSPECT CNC 2016, the type of group in which the points are added depends on whether probe path generation in the **Features** tab is enabled or disabled.

When PowerINSPECT CNC creates guided surface points from dynamic points, it adds them to a new:

- CNC Surface Points group if probe path generation is enabled.
- Guided Surface Points group if probe path generation is disabled.

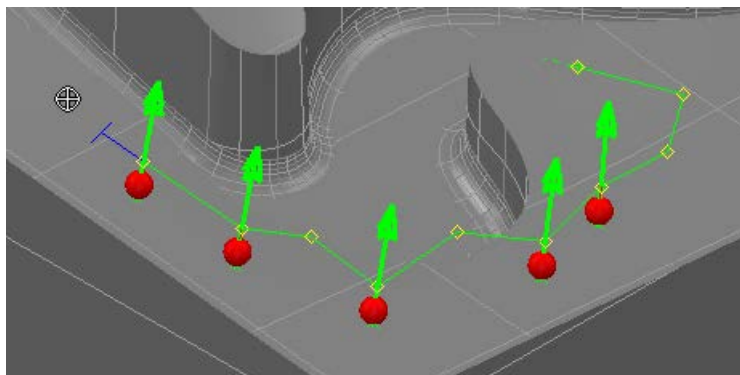
When PowerINSPECT CNC creates guided edge points from dynamic points, it adds them to a new:

- CNC Edge Points group if probe path generation is enabled.
- Guided Edge Points group if probe path generation is disabled.






Creating CNC surface points groups

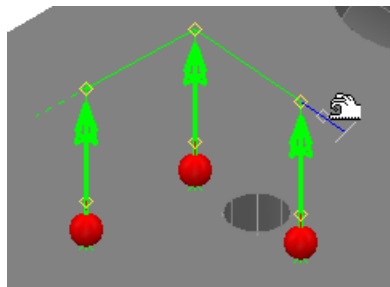
The new CNC Surface Points group enables you to specify the probe path for the free-form inspection of surfaces on a part using guided surface points. To create guided surface points, you must specify a **UserDefined** probing method for the probe path.

When you select a **UserDefined** probing method, PowerINSPECT CNC activates the Probe Path Editor, which enables you to specify the points to be measured. For surface inspection groups, the Probe Path Editor has been enhanced to provide greater flexibility for creating points and modifying the probe path.



To create the probe path for a CNC Surface Points group using guided surface points:

- 1 In the **Features** tab, deselect the **Generate Probe Path**  button to enable the creation of probe paths. The button changes to .
- 2 In the **Inspection Groups** toolbar, click the **CNC Surface Points**  button. An inspection group is added to the inspection sequence and the **Surface Inspect** dialog is displayed in the **Features** tab.
- 3 In the **Surface Inspect** dialog, select a  **UserDefined** probing method from the Probing method list. In the CAD view, the cursor changes to  to indicate the Probe Path Editor is active.
- 4 Create the points and positions in the path:
 - To add a guided surface point, double-click the model. A red sphere is displayed at the probing position, and the approach point is shown as a yellow diamond.
 - To add an intermediate point, move the cursor over the path or over the extensions from the first and last points in the path. When a grey diamond is displayed, click and hold the diamond, drag the point to a position, then release the mouse button. The diamond is displayed in yellow.



By default, the extension from the last specified point is highlighted blue.

- To insert a guided surface point within the probe path, click the path, then double-click the model.
- To insert an intermediate point within the probe path, right-click the path, and then select **Split** from the context menu.

- To move a guided surface point, click and hold the red sphere, drag the point to its new position, then release the mouse button. Alternatively, right-click the sphere, select **Edit** from the context menu, then enter the position in the **Point Coordinates** dialog.
- To move an intermediate point, click and hold the yellow diamond, drag the point to its new position, then release the mouse button. Alternatively, right-click the diamond, select **Edit** from the context menu, then enter the position in the **Point Coordinates** dialog.
- To use the current probe-tip position for an intermediate point, right-click the yellow diamond, and then select **Move to CMM Position** from the context menu.
- To delete a guided surface point or intermediate point, click and hold, then drag the cursor to draw a rectangle over the points you want to delete. When the points are selected, release the mouse button, then right-click the CAD view and select **Delete** from the context menu.




When you modify or delete a point, PowerINSPECT CNC automatically updates the probe path. If a guided surface point is referenced by an RPS alignment, the point cannot be modified or deleted.

- 5 When you have finished, click  in the **Surface Inspect** dialog to save the path and close the Probe Path Editor. You can now run the item as part of the inspection sequence.

To view and update the group settings in the **Inspection Group: Surface Inspect** dialog, right-click the group in the inspection sequence, and then select **Modify Item** from the context menu.



To edit the probe path created using another method, select

*the group in the inspection sequence, click  in the group's **Surface Inspect** dialog, and then select a **UserDefined** method.*

Specifying nominals

A **Normal Vector** area has been added to the dialogs for Probed Parallel Plane, Plane: Parallel + Distance, Plane: Parallel Through Point, and Plane: Point + Vector items. Use it to specify the nominal vector of the planes' perpendiculars, as with other plane items.

For example, the **Normal Vector** area in the **Probed Parallel Plane** dialog:

Probed Parallel Plane

Name: Plane 2

☒ Use nominals

Visible ☒
Output in report ☒

Page 1 Point Sources(0)

Coordinate System: <Active Alignment>

Material Condition: Mean

Hide Sign Information: ☐

Reference Plane: Plane 1

Normal Vector

Coordinate Type: Cartesian

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






Distance

Nominal	Low Tol	High Tol
-10.000	-0.100	0.100

Comment:

OK Apply Cancel Help


To specify the nominals for one of these plane items:

- 1 In the inspection sequence, double-click the item's entry to display its definition dialog. Alternatively, right-click the entry, and then select **Modify Item** from the context menu.
- 2 Select the **Use nominals** check box.
- 3 In the **Normal vector** area, choose an entry from the **Coordinate type** list to specify the nominal format. Select:
 - **Cartesian** to enter the nominal in **IJK** format.
 - **Apparent angles** to enter the nominal in **ABC** degree format.
 - **Spherical** to enter the nominal as an azimuth (**A**) and elevation (**E**).
- 4 Enter the nominals of the item in the **Nominal** boxes. Alternatively, click , and then select **from CAD Entity** to load the nominals from the CAD model, or select **from Active Measure** to load the nominals from the current Measure.
- 5 Enter the tolerance range for the item in the **Low Tol** and **High Tol** boxes. Alternatively, click the  buttons to create a tolerance band to use across items.
- 6 To display the details of each nominal in the report, click , and then select the check box to the right of each nominal value.
- 7 To invert the direction of the vector, click .
- 8 Click **OK** to save your changes and close the dialog. The normal vector details are displayed in the **Info** tab, the **Report** tab, and in the inspection sequence when you click  next to the item's entry.

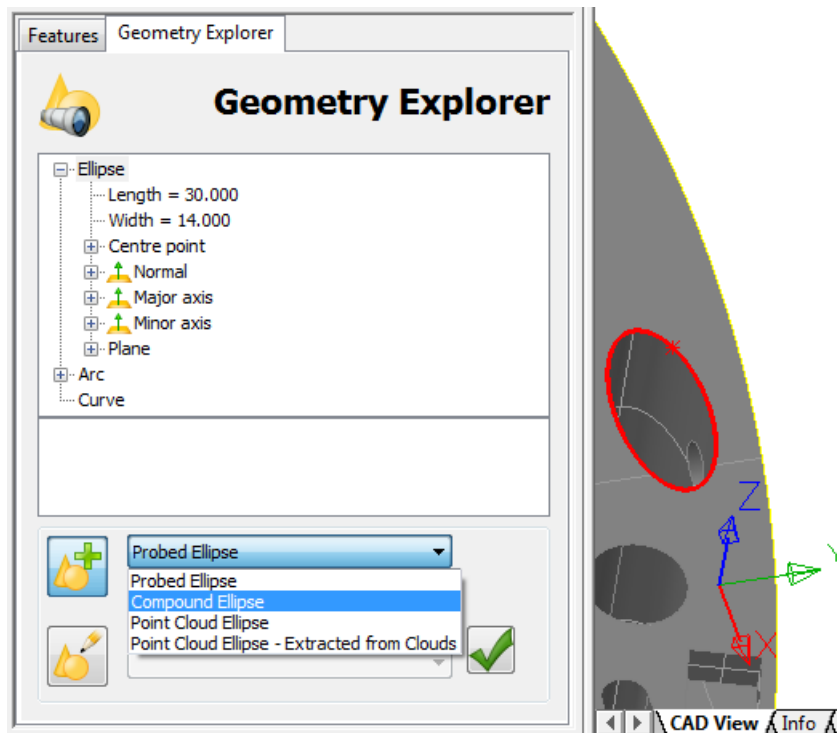
Selecting ellipses


You can now select ellipse features from CAD models using the Wireframe Checker's Wireframe and Points picking modes. This enables you to use the **Geometry Explorer** tab to extract and display the nominals for the feature, and to create a geometric item in the inspection sequence.

To create an ellipse item using the Wireframe picking mode:

- 1 Display the model in the CAD view.
- 2 Select the **Geometry Explorer** tab.
- 3 In the **Mouse Context** toolbar, select the **Wireframe Checker**  button.
- 4 Right-click the CAD view, and then select **Wireframe** from the context menu.
- 5 Move the cursor over the model until the ellipse is highlighted yellow.
- 6 Click the wireframe to select the highlighted feature. The ellipse is displayed in red and its nominals are listed in the **Geometry Explorer** tab.

- 7 Check that the item displayed in the Item list is correct. When you select an ellipse, you can create a Probed Ellipse, Compound Ellipse, Point Cloud Ellipse, or Point Cloud Ellipse - Extracted from Clouds item.




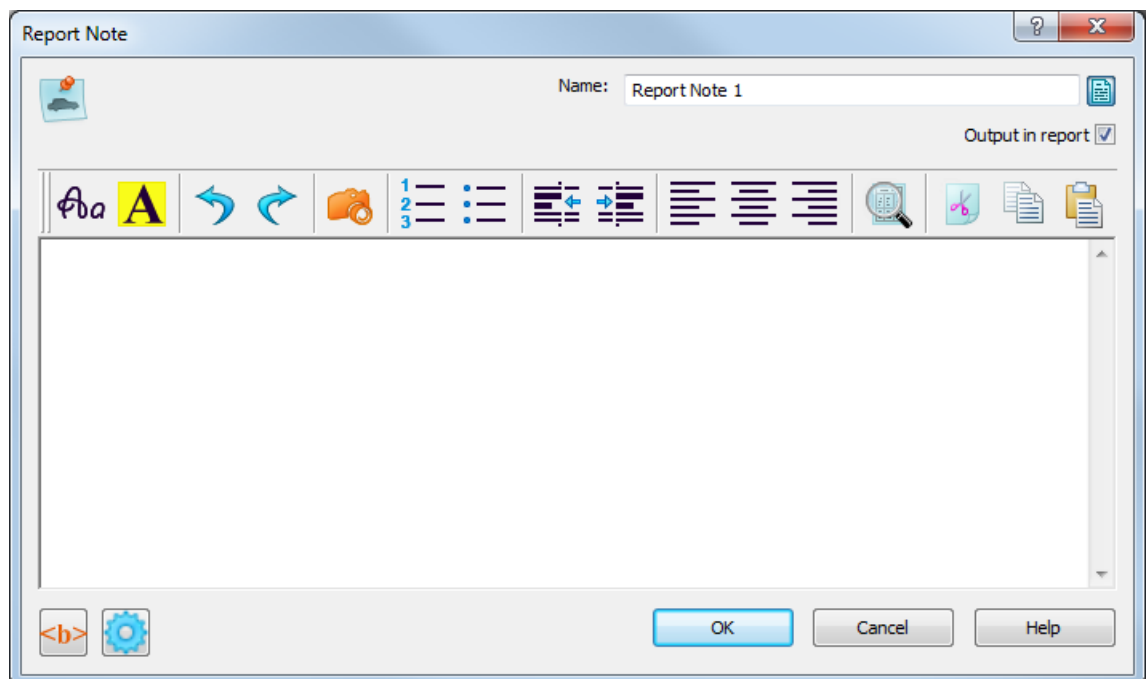
- 8 If you are creating a Probed Ellipse item, select a reference plane in the **Linked to** list.
- 9 Click . PowerINSPECT CNC adds the new item to the inspection sequence.

Report Note items

A new Report Note item has been added to the **Miscellaneous** toolbar. It works in a similar way to the Comment item, except that it is used to insert extra information and comments in the **Report** tab.

To create a Report Note item:

- 1 In the **Miscellaneous** toolbar, click the **Report Note**  button. The **Report Note** dialog is displayed.



- 2 Enter a **Name** for the item.



To hide the name in the report, click . Click the button again to reinstate the name.

- 3 To include the item in the **Report** tab, select the **Output in report** check box.

- 4 In the entry area of the window, enter the text and images you want to display.
- 5 To format the text, use the buttons in the **Report Note** toolbar. Click:



to select a typeface for the current selection. The **Font** dialog is displayed.



to specify the background colour of the text.



to undo the last action.



to redo the last undo action.



to insert an image file. The **Picture** dialog is displayed.



to create a numbered paragraph.



to create a bulleted paragraph.



to outdent the selected paragraphs.



to indent the selected paragraphs.



to left-justify the selected paragraphs.



to centre the text of the selected paragraphs.



to right-justify the selected paragraphs.



to display the comment in the **Print Preview** window.






to remove the selected text or image from the window and copy it to the Windows clipboard.



to copy the selected text or image to the Windows clipboard.




to paste the contents of the Windows clipboard into the dialog.

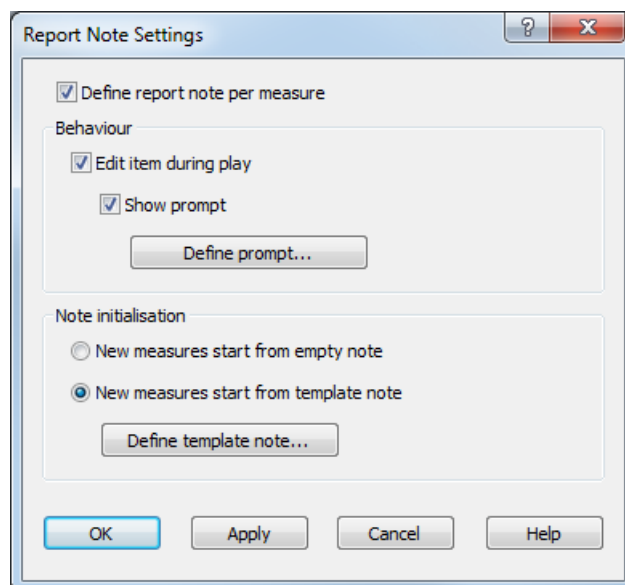
- 6 To edit the HTML source, click . The button changes to .
- 7 By default, the content of the Report Note item is applied to all Measures. To customize the Report Note item for each Measure, and to specify the action of the item when it is played or run as part of the inspection sequence, click the **Settings**  button to open the **Report Note Settings** dialog (see page 19).
- 8 In the **Report Note** dialog, click **OK** to close the dialog and add the item to the inspection sequence. The contents of the note are displayed in the **Report** tab.

Customizing Report Note items

Use the **Report Note Settings** dialog to specify how the Report Note item behaves, and to customize the note for each Measure in the inspection document.

To specify the settings for a Report Note item:

- 1 In the **Report Note** dialog, click the **Settings**  button. The **Report Note Settings** dialog is displayed.



- 2 Select the **Define report note per measure** check box to enable the Report Note item to be customized for each Measure.



If you deselect the check box for a customized Report Note item, it reverts to the contents you first specified.

- 3 Choose the initial contents of the **Report Note** dialog in each new Measure. Select:
 - **New measures start from empty note** to leave the note empty.
 - **New measures start from template note** to display the original note. To change the content, click **Define Template Note** and edit the note.
- 4 To display the **Report Note** dialog and edit the note when you run or play the item, select the **Edit item during play** check box.
Select the **Show prompt** check box to also display a prompt that cannot be edited. Click **Define Prompt** to specify the non-editable prompt.
- 5 Click **OK** to save your changes and close the dialog.

To customize the Report Note item in a Measure, select the Measure from the **Active Measures** list, and then edit the note.



Section enhancements

Several changes have been made to how you create sections and view their measured results:

- The **Cut Plane** area of the **Section** dialog now includes a slider when you select a major plane. Use it to change the **Coordinate** of the point through which the major plane must pass, and see the cross-section update in real time in the CAD view.
- To improve usability, a new type of section group has been created for measuring with probe paths, and another has been created to enable you to specify exact positions on the section to be probed (see page 22).
- A new shading mode option enables you to control how models with sections are displayed in the CAD view (see page 27).
- To improve the display of sections, the Section Group view and its toolbars have been redesigned (see page 29).
- A new **Section Measurement Display** pull-out toolbar combines the deviation display options that previously were split across the CAD view and the Section Group view (see page 32).
- Previously, the display of probed points in section groups was controlled using the settings for surface inspection points. Now, a new tab in the **Filter Display** dialog enables you to control their display independently (see page 34).

Changes to section groups

To improve usability, two new types of section group have been created:


-  **Guided Section** group (see page 22).
-  **CNC Section** group (see page 25).

The Guided Section group works in a similar way to the Guided Surface Points group, and enables you to ensure that specific positions on the cross-section of a part are measured. By specifying guided surface points, you can ensure repeatability across Measures and use the points as constraining features in RPS alignments.

The CNC Section group combines the ability to specify probe paths with the ability to create guided surface points. This enables you to automate the inspection of sections on a part that can be repeated across Measures. As with Guided Section groups, you can also use the guided surface points as constraining features in RPS alignments. As a result, probe path support for Section groups is no longer supported; however, you can continue to create this type of group in manual mode.


In addition, section groups are now created using the **Section** toolbar.

To create a section group:

- 1 In the **Item** toolbar, select the **Section**  button. The **Section** pull-out toolbar is displayed.
- 2 Choose the type of section group you want to create. Click:

 to create a Section group.


 to create a Guided Section group.

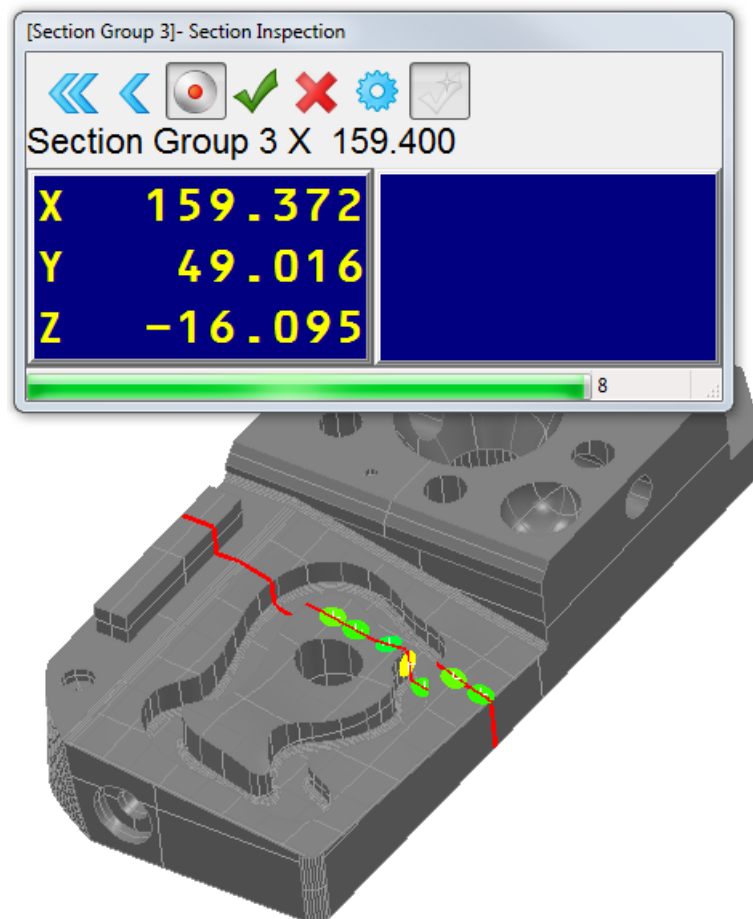
 to create a CNC Section group.

Creating guided section groups


Guided Section groups enable you to ensure that specific positions on the cross-section of a part are measured.

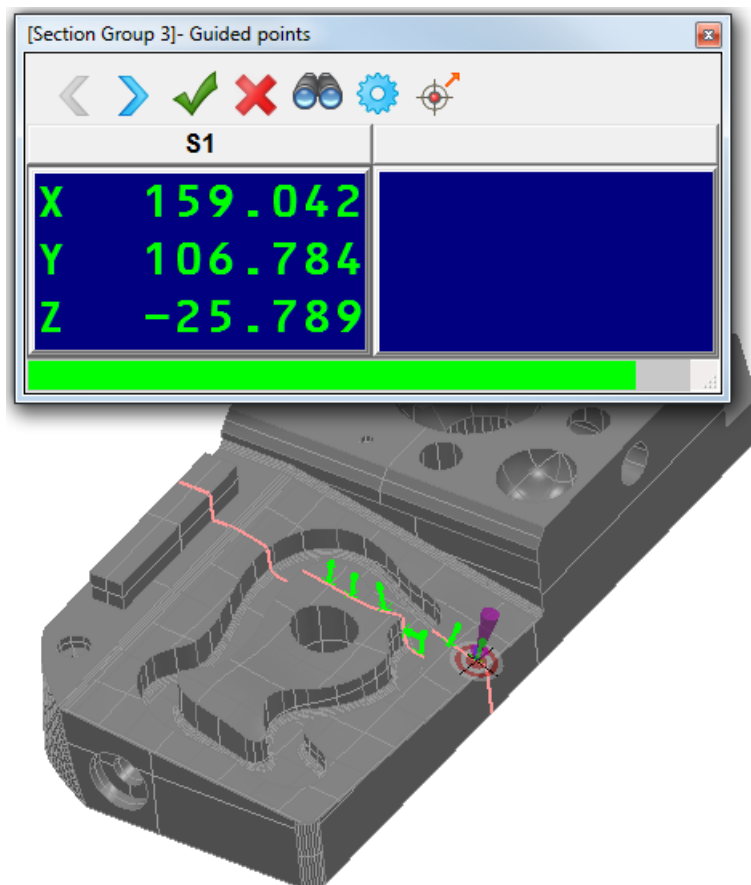
To create a Guided Section group:

- 1 In the **Section** toolbar, click the **Guided Section**  button. The **Section** dialog is displayed.
- 2 Use the **Cut Plane** area of the dialog to specify the location of the cross-section on the part, and use the other settings to specify naming rules for the probed points, and the tolerances and offsets for the section.
- 3 Click **OK** to close the dialog and add the group to the inspection sequence.
- 4 Play the group. PowerINSPECT CNC displays a confirmation message.
- 5 Click **Yes**. PowerINSPECT CNC switches to full-screen mode and displays the **Section Inspection** dialog.
- 6 Probe the points on the section you want to inspect.




The left of the **Section Inspection** dialog displays the coordinate of the point on the section plane nearest the probed point. The progress bar at the bottom of the dialog shows how close the probed point is to the point on the section. When the probed point is in-tolerance, a coloured disc is displayed on the model, and the right of the dialog shows the distances between the probed point and the point on the section.

- 7 When you have specified the points, click  to close the dialog and return to the CAD view.
- 8 In the inspection sequence, right-click the group, and then select **Reset Item** from the context menu.
- 9 Play the group again. PowerINSPECT CNC switches to full-screen mode and displays the **Guided Points** dialog. The positions of the points to be probed are shown as arrows on the model.
- 10 Probe the points indicated by the arrows.









The left of the **Guided Points** dialog displays the coordinate of the current guided point. The progress bar at the bottom of the dialog shows how close the probed point is to the guided point. When the probed point is in-tolerance, a coloured disc is displayed on the model, and the right of the dialog shows the distances between the probed point and the guided point.

- 11 When you have finished probing, click  to close the dialog.

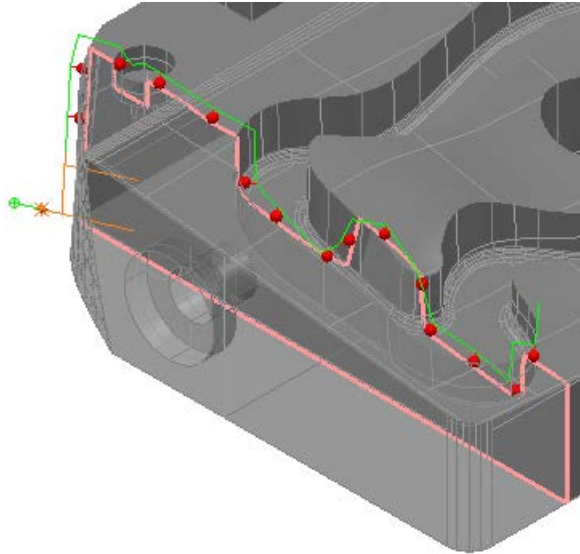
Creating CNC section groups

The new CNC Section group enables you to specify the probe path for the inspection of sections on a part using guided surface points. To create guided surface points, you must first specify an **AutoTouchTrigger** probing method for the probe path, and then edit the probe path to use a **UserDefined** probing method.



To create the probe path for a CNC Section group:

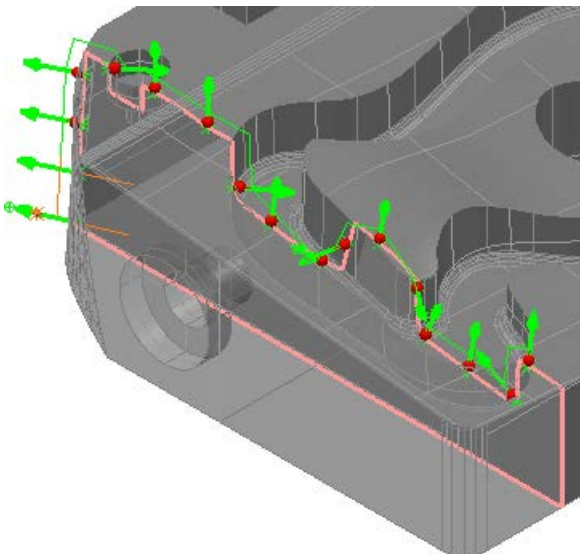
- 1 In the **Features** tab, deselect the **Generate Probe Path**  button to enable the creation of probe paths. The button changes to .
- 2 In the **Section** toolbar, click the **CNC Section**  button. A section group is added to the inspection sequence and the **Section Inspect** dialog is displayed in the **Features** tab.
- 3 In the inspection sequence, double-click the group's name to display the **Section** dialog.
- 4 In the **Section** dialog:
 - a Use the **Cut Plane** area to specify the location of the section on the part.
 - b To use CAD levels to control the area of the part to be probed, select the **Use custom levels** check box, and then click **Levels** to specify the levels.
 - c Click **OK** to save your changes and close the dialog.
- 5 In the **Section Inspect** dialog:
 - a Select the  **AutoTouchTrigger** probing method of **Raster** from the Probing method list.
 - b Click the **Parameters**  button. The **Raster Parameters** dialog is displayed.
- 6 Complete the parameters in the **Raster Parameters** dialog, and then click  to save your changes and close the dialog.

- 7 In the **Section Inspect** dialog, click  to save your changes. The probe path of the section is displayed in the CAD view, for example:



To edit the probe path of a CNC Section group:

- 1 In the inspection sequence, select the group.
- 2 In the **Section Inspect** dialog, click .
- 3 In the Probing method list, select a  **UserDefined** probing method to activate the Probe Path Editor. PowerINSPECT CNC converts the points to guided surface points, for example:





- 4 Click .

Section shading mode

A new shading mode option has been added to the **CAD View** toolbar, which enables you to control how models with sections are displayed in the CAD view. Use it to help you identify selected section groups more easily.

To choose a shading mode:

- 1 In the inspection sequence, click 💡 next to the section group you want to work with. The icon changes to 💡.
- 2 Select the section group in the inspection sequence.
- 3 In the **CAD View** toolbar, click  below the **Section Shading**

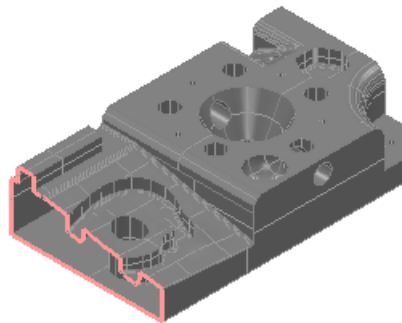
Mode  button. The pull-out toolbar is displayed.

- 4 Click the shading mode you want to use. Click:



to hide the part of the model that appears nearest.

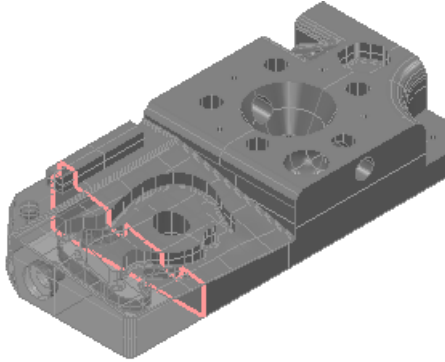
For example:





to display a semi-transparent representation of the part of the model that appears nearest.

For example:



*This only works when you select the **Show Shaded and Wireframe** or **Shaded View** shading mode options for the model.*



to disable the shading mode.

Section view

To improve the display of sections, the Section Group view has been redesigned. Previously, PowerINSPECT CNC displayed a separate **Section Group** tab at the bottom of the main graphics window for each section group you created. In PowerINSPECT CNC 2016, a single **Section View** tab is displayed, which shows the details of the currently selected section group in the inspection sequence.

When you have probed a section, you can use the updated **Section View** toolbars (see page 30) to choose how the deviations of measured points are displayed, and to manipulate the view.

In addition, you can now use the CAD View Report item to take a snapshot of the Section view.















The background colour of the **Section View** tab is controlled using the settings for the **CAD View** tab. To change the colour, use the **Colours > General** page of the **Options** dialog (see page 58).

Section view toolbars

The **Section View** toolbars are displayed when you select the **Section View** tab. They now include several new buttons, plus others that were previously available only in the **CAD View** and **View Options** toolbars in the **CAD View** tab. These additional buttons provide greater flexibility to manipulate the view of the cross-section through the part, and to control what information is displayed.

The buttons added to the **Section View** toolbars include:




Button	Action
	<p>Click  below the View Mode button to display the pull-out toolbar. Use the buttons in the toolbar to select the way in which the left mouse button manipulates the view.</p> <p> <i>The button is the same as that available in the CAD View toolbar, and replaces the previous Zoom In and Zoom Out buttons.</i></p>
	<p>Click  below the Rotate View Around View Axis button to display the pull-out toolbar.</p> <p>Click:</p> <p> to rotate the view around the normal of the section plane in a clockwise direction.</p> <p> to rotate the view around the normal of the section plane in a counter-clockwise direction.</p>
	<p>Click the Switch Section Front/Back View button to reverse the viewing direction of the section plane.</p>


Button	Action
	<p>Click  below the Section Measurement Display button (see page 32) to display the pull-out toolbar. Use the buttons in the toolbar to control the display of the deviation of probed section points.</p>
	<p>Click the Filter Display button to display the Filter Display dialog. Use the new Section Inspection tab (see page 34) in the dialog to specify which probed points are displayed.</p> <p> <i>The button is the same as that available in the View Options toolbar.</i></p>

Displaying probed section points

A new **Section Measurement Display** button with pull-out toolbar has been added to the **View Options** toolbar in the CAD view, and to the **Section View Options** toolbar in the Section view. It combines the deviation display options for probed section points that previously were split across the **CAD View** and **Section Group** tabs.

To select the way in which probed section points are displayed:

- 1 In the inspection sequence, click  next to the measured section groups whose probed points you want to display. The icon changes to .
- 2 In the **View Options** toolbar (in the **CAD View** tab) or the **Section View Options** toolbar (in the **Section View** tab), click  below

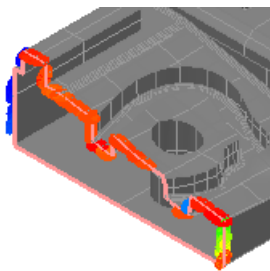
the **Section Measurement Display**  button. The pull-out toolbar is displayed.

- 3 Choose the display mode you want to use. Click:



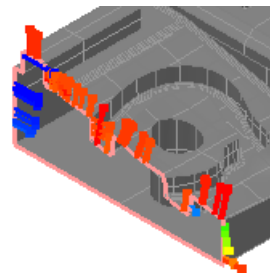
to display the deviation of each point as **Spot Confetti**.

For example, in the CAD view:



to display the deviation of each point as **Deviation Lines**.

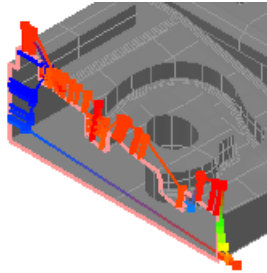
For example, in the CAD view:





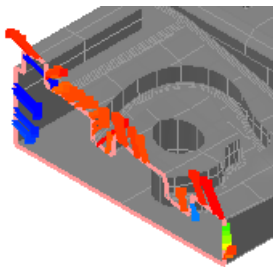
to display the deviation of each point as **Connected Deviation Lines**.

For example, in the CAD view:



to display the deviation of each point as **Vector Lines**.

For example, in the CAD view:



PowerINSPECT CNC compares the measurement results to the CAD data and displays the deviation using colour:

- Green points are within tolerance.
- Red points are above the tolerance band.
- Blue points are below the tolerance band.

To hide the measured points, deselect the **Section Measurement Display** button.






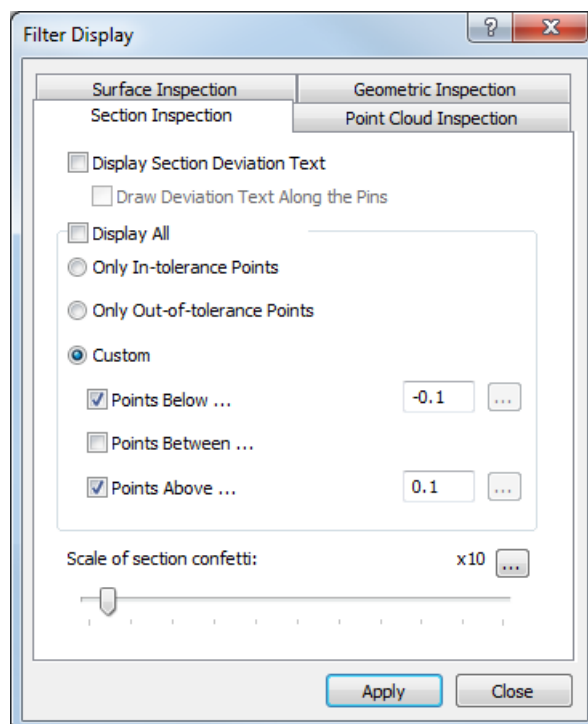
*To show more details for each measurement, select the **Display section deviation text** check box in the **Section Inspection** tab of the **Filter Display** dialog (see page 34).*

Display settings for probed section points

Previously, the settings in the **Surface Inspection** tab of the **Filter Display** dialog also controlled the display of probed section points. In PowerINSPECT CNC 2016, the dialog has been extended to include a **Section Inspection** tab. Use the new tab to specify which measured points in section groups are displayed in the CAD view and in the Section view when you select a **Section Measurement Display** option.

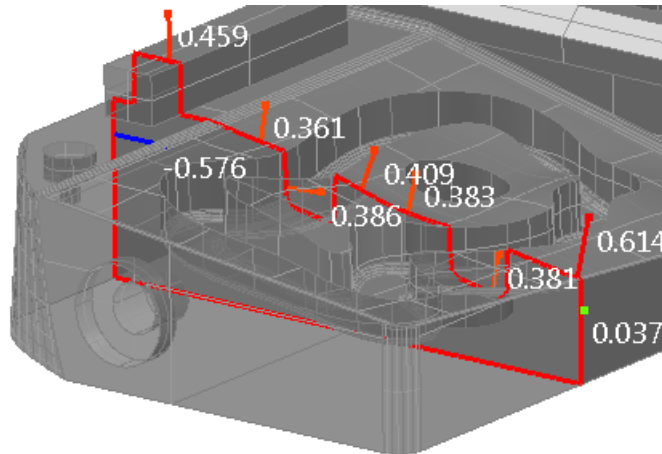
To specify the settings for the display of probed section points:

- 1 In the inspection sequence, click  next to the measured section groups whose probed points you want to display. The icon changes to .
- 2 Select an option in the **Section Measurement Display** toolbar (see page 32).
- 3 In the **View Options** toolbar (in the **CAD View** tab), or the **Section View Options** toolbar (in the **Section View** tab), click the **Filter Display**  button. The **Filter Display** dialog is displayed.
- 4 Select the **Section Inspection** tab.



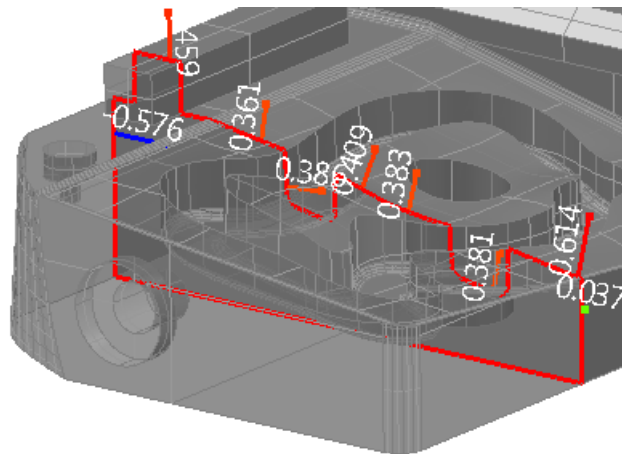
- 5 To show the calculated deviation of each probed point alongside its confetti, deviation line, or vector, select the **Display section deviation text** check box.



For example, in the CAD view:



- 6 If the deviation of probed points are displayed as lines or vectors, select the **Draw deviation text along the pins** check box to display the calculated deviation along the length of the line or vector.

For example, in the CAD view:



- 7 By default, PowerINSPECT CNC displays all probed points in the section group. To display only some of the points, deselect the **Display all** check box, and then select:
- **Only in-tolerance points** to display only points that are within tolerance.
 - **Only out-of-tolerance points** to display only points that are out-of-tolerance.
 - **Custom** to specify the lower and upper tolerance limits of the points to be displayed. Select the **Points below** check box to display points below the specified lower tolerance limit; select the **Points above** check box to display points above the specified upper tolerance limit; select **Points between** to display points between the specified upper and lower tolerance limits.
- 8 To change the magnification of the deviation indicators, move the slider. To change the magnification limit, click , and then enter a new maximum value.
-  *This has no effect if the deviation is displayed as **Spot Confetti**.*
- 9 Click **Apply** to save your changes and display them in the CAD view and in the Section view.
- 10 Click **Close** to close the dialog.

Rack support enhancements

Several improvements have been made to the rack support introduced in PowerINSPECT CNC 2015 R2. When you locate a rack using the **Rack Locate** dialog, you can now:

- view the origin of the rack coordinate system in machine coordinates.
- view the rack's orientation in **IJK** vectors.
- switch the display of the **Safe Point** position between rack and machine coordinates.

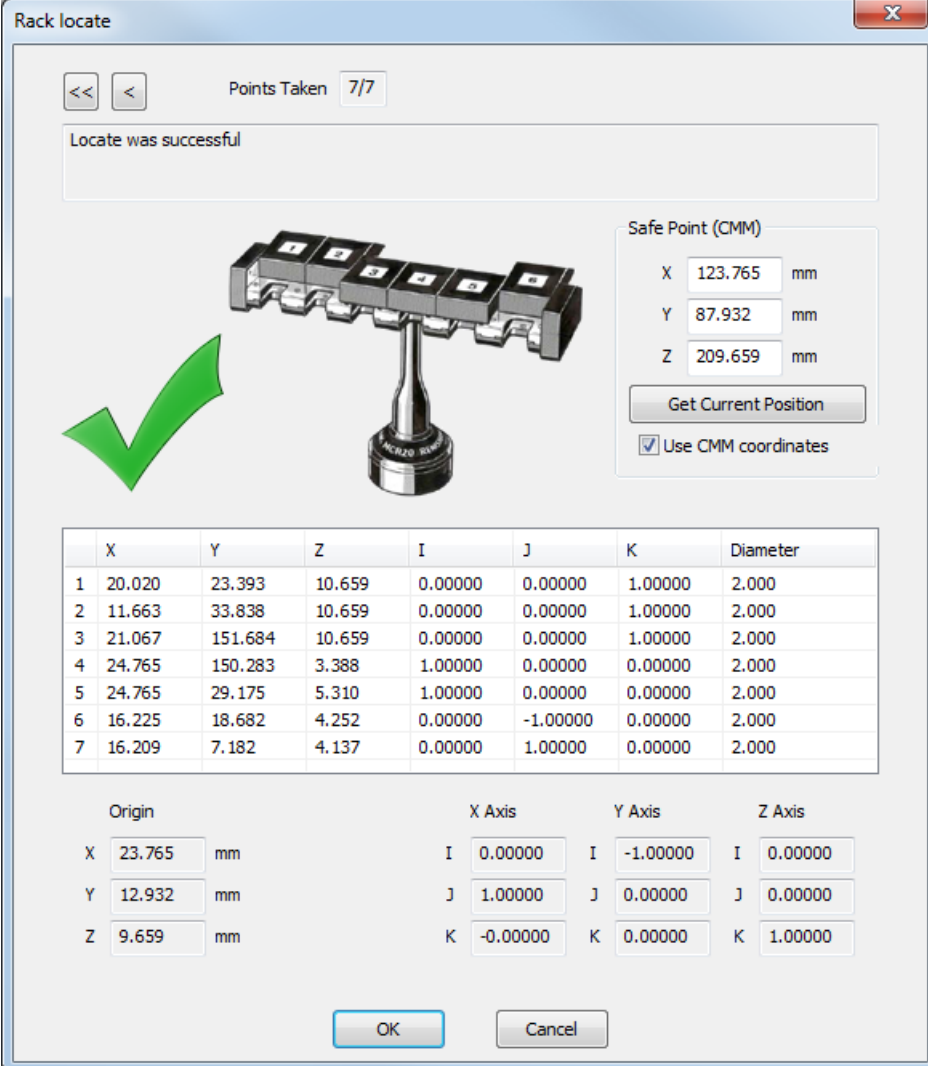
In addition, when you locate a port or insert, the **Port Locate** dialog now displays the updated **Delta Offset**.

To locate a rack:

- 1 Calibrate the active probe tool.
- 2 In the **CNC racks** area of the **Machine** tab, select the rack you want to locate, and then click **Locate**. The **Rack Locate** dialog is displayed.
- 3 Follow the diagram and instructions in the dialog to probe the points.

As you take each point, the dialog displays its **XYZ** machine coordinates, **IJK** vectors, the **Diameter** of the probe, and the number of points remaining.

When you have taken all seven points, the dialog displays *Locate was successful* to indicate the rack has been located on the measuring device. It also shows the **Origin** of the rack coordinate system in machine coordinates, the rack's orientation in **IJK** vectors, and the position of the **Safe Point**. For example:



Points Taken 7/7

Locate was successful

Safe Point (CMM)

X 123.765 mm

Y 87.932 mm

Z 209.659 mm

Get Current Position

☒ Use CMM coordinates

	X	Y	Z	I	J	K	Diameter
1	20.020	23.393	10.659	0.00000	0.00000	1.00000	2.000
2	11.663	33.838	10.659	0.00000	0.00000	1.00000	2.000
3	21.067	151.684	10.659	0.00000	0.00000	1.00000	2.000
4	24.765	150.283	3.388	1.00000	0.00000	0.00000	2.000
5	24.765	29.175	5.310	1.00000	0.00000	0.00000	2.000
6	16.225	18.682	4.252	0.00000	-1.00000	0.00000	2.000
7	16.209	7.182	4.137	0.00000	1.00000	0.00000	2.000

Origin

X 23.765 mm

Y 12.932 mm

Z 9.659 mm

X Axis

I 0.00000

J 1.00000

K -0.00000

Y Axis

I -1.00000

J 0.00000

K 0.00000




Z Axis

I 0.00000

J 0.00000

K 1.00000

OK Cancel

 To delete the last point taken, click . Alternatively, click  to delete all points.

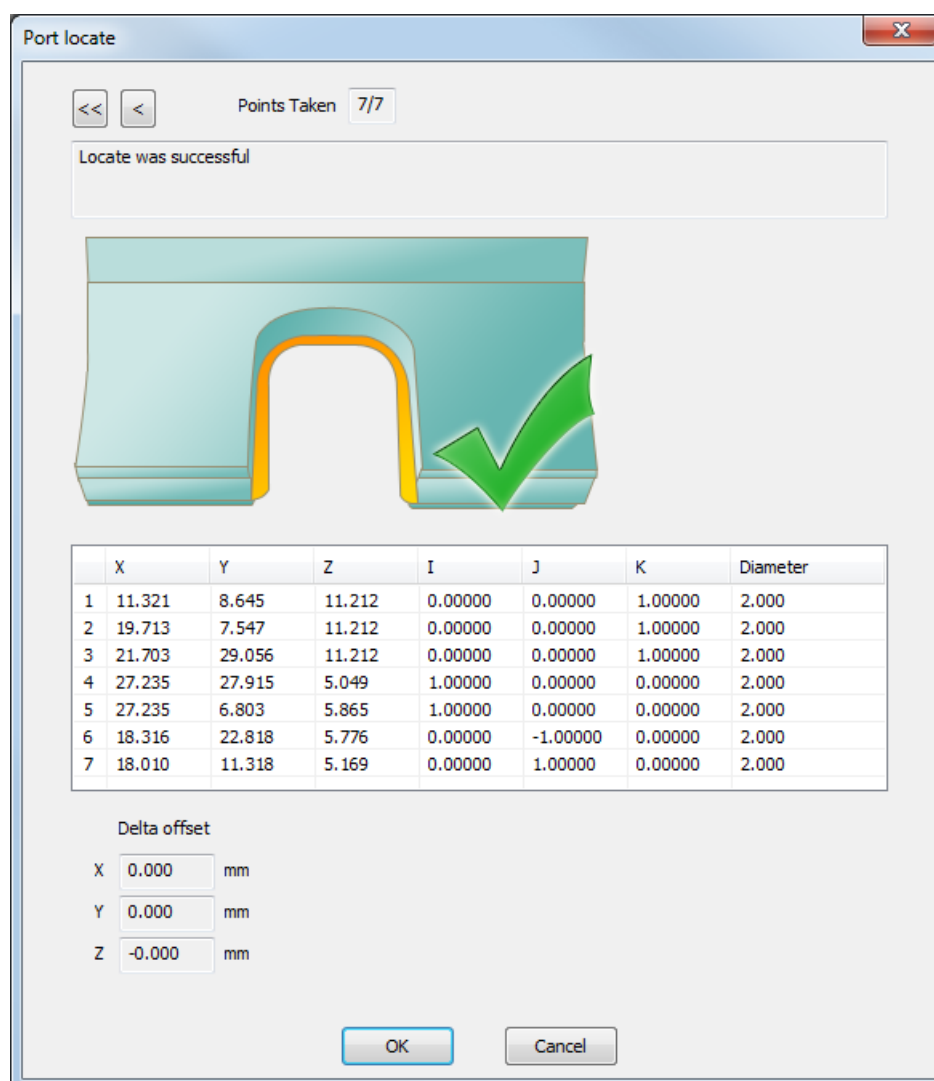
- 4 To display the **Safe Point** in machine coordinates, select the **Use CMM coordinates** check box. Deselect the check box to display the position in rack coordinates.
- 5 To enter a new **Safe Point**, overtype the values in the **X**, **Y**, and **Z** boxes, or click **Get Current Position** to use the current quill mounting location.
- 6 Click **OK** to save the location and close the dialog.

To locate a port or insert:

- 1 Calibrate the active probe tool.
- 2 In the **CNC racks** area of the **Machine** tab, select the rack whose ports or inserts you want to locate, and then click **Edit**. The **Rack** dialog is displayed.
- 3 In the **Ports** area, select the port you want to adjust from the Port list, and then click **Locate Insert**. The **Port Locate** dialog is displayed.
- 4 Follow the diagram and instructions in the dialog to probe the points.

As you take each point, the dialog displays its **XYZ** machine coordinates, **IJK** vectors, the **Diameter** of the probe, and the number of points remaining.


When you have taken all seven points, the dialog displays *Locate was successful* to indicate the port, or insert, has been located within the rack. It also shows the **Delta Offset** from the original calculated position. For example:



Port locate

Points Taken 7/7

Locate was successful



	X	Y	Z	I	J	K	Diameter
1	11.321	8.645	11.212	0.00000	0.00000	1.00000	2.000
2	19.713	7.547	11.212	0.00000	0.00000	1.00000	2.000
3	21.703	29.056	11.212	0.00000	0.00000	1.00000	2.000
4	27.235	27.915	5.049	1.00000	0.00000	0.00000	2.000
5	27.235	6.803	5.865	1.00000	0.00000	0.00000	2.000
6	18.316	22.818	5.776	0.00000	-1.00000	0.00000	2.000
7	18.010	11.318	5.169	0.00000	1.00000	0.00000	2.000

Delta offset



X 0.000 mm

Y 0.000 mm

Z -0.000 mm

OK Cancel



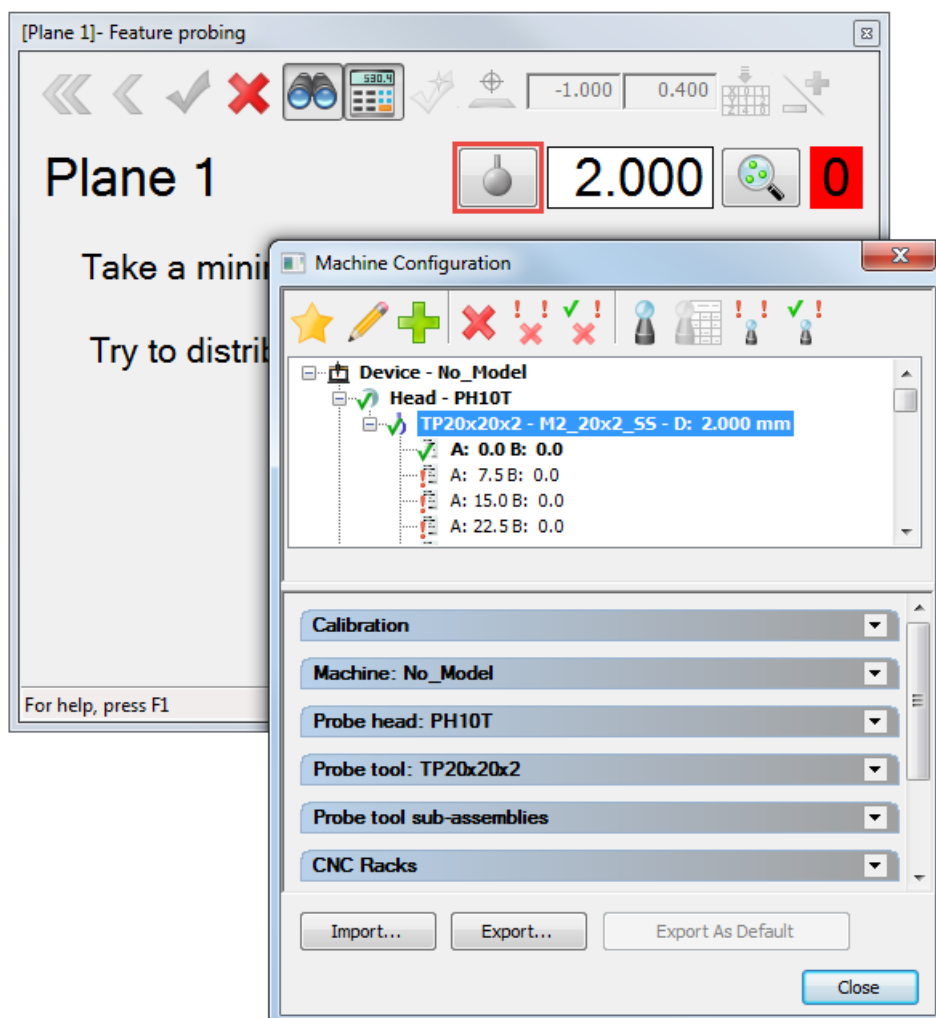
To delete the last point taken, click . Alternatively, click  to delete all points.

- 5** Click **OK** to save the location, close the dialog, and return to the **Rack** dialog.
- 6** Repeat steps 3 to 5 to continue locating ports or inserts.
- 7** Click **Save** to save your changes and close the dialog.

Editing probe databases



A new **Edit Database** button has been added to the **Feature Probing** dialog, which is displayed when measuring items manually. Use it to view and change the settings of the measuring device in the **Machine Configuration** dialog.



Label enhancements

Several improvements have been made to PowerINSPECT CNC's label feature:


- Labels can now be displayed in the CAD view for GD&T items (see page 43).
- The **Session Label Settings** dialog includes a new option that enables you to display grid lines on labels (see page 45).
- The form tolerance requirements for a Geometric item can now be displayed on the item's label (see page 46).

Displaying labels for GD&T items

You can now display the details of GD&T items in the CAD view using labels. In addition, new options have been added to the **Session Label Settings** dialog to enable you to further customize what information is included in the label by default.

To display labels for GD&T items:

- 1 In the inspection sequence, click  next to the GD&T item entries for which you want to display a label. The icon changes to .

- 2 In the **View Options** toolbar, click  below the **Show/Hide**



Labels button. The pull-out toolbar is displayed.

- 3 Select a label mode. A label is displayed in the CAD view for each visible GD&T item.

- 4 In the **View Options** toolbar, click the **Show Global Label Settings**



button to specify label settings. The **Session Label Settings** dialog is displayed.

- 5 Select the **General** tab.

- 6 Select the settings you want to use. In the **Rows** area, select the:

- **Control frame** check box to display a summary of the tolerancing requirements for a GD&T item in a control frame at the top of the label. For GD&T Datum items, PowerINSPECT CNC displays the **Datum feature symbol**.



A control frame is not displayed in the label for a GD&T Linear Dimension item.

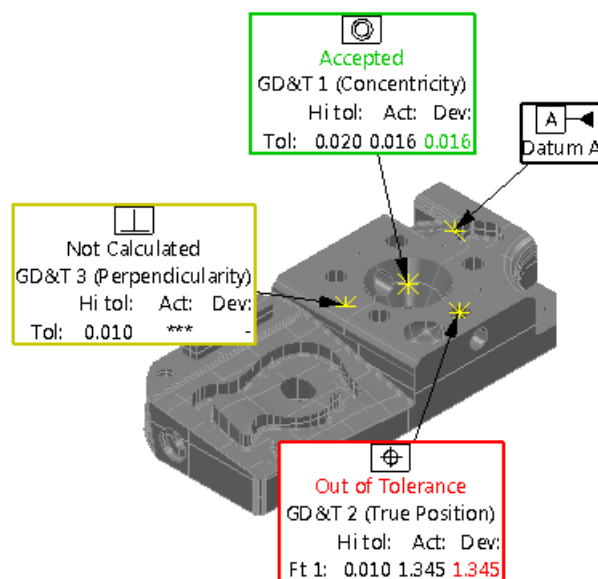
- **GD&T status** check box to display the measurement status for a GD&T item.

- 7 Complete the dialog, and then click **OK** to save your changes and close the dialog.

In the CAD view, PowerINSPECT CNC displays a label, and an arrow terminated by an asterisk marker. The arrow links the label to the feature on the model used to specify the GD&T item. If you specify more than one feature for a GD&T True Position item, the marker is displayed on the first specified feature.

The colour of the label's border indicates the result of the item:

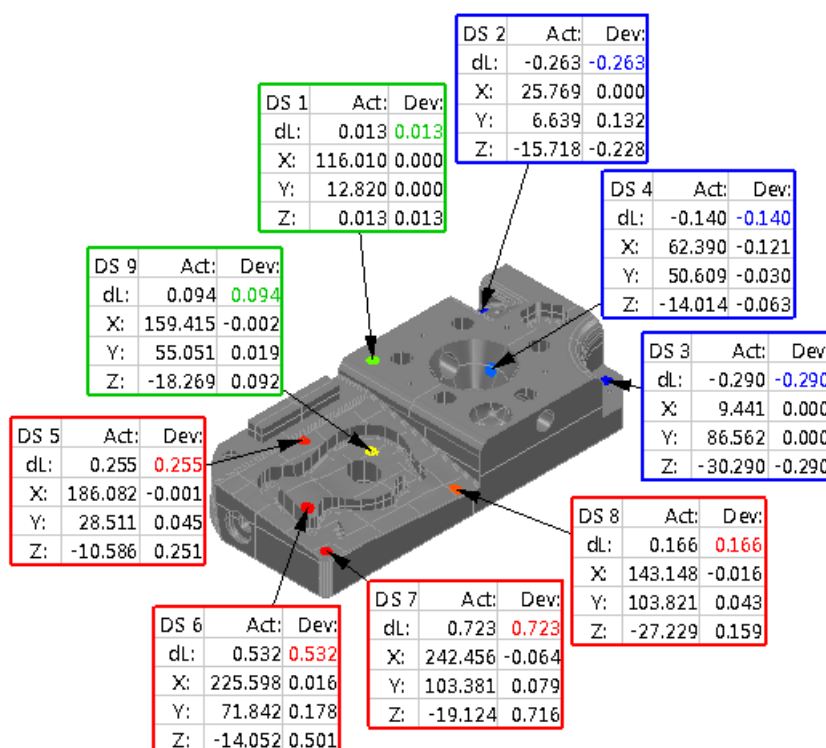
- Green indicates the measurement is in-tolerance.
- Red indicates the measurement is out-of-tolerance.
- Gold indicates the measurement has not been calculated.
- Black indicates the item is unmeasured, for example, a GD&T Datum item.




*To change the colour of the asterisk marker, use the **Colours > Entities** page of the **Options** dialog (see page 58).*

Displaying grid lines on labels

You can now display grid lines on labels. This makes it easier to view the information shown in the labels in the CAD view, and in the report. For example:



To display grid lines on labels:

- 1 Display the labels in the CAD view.
- 2 In the **View Options** toolbar, click the **Show Global Label Settings**  button. The **Session Label Settings** dialog is displayed.
- 3 Select the **General** tab.
- 4 In the **Appearance** area, select the **Display gridlines** check box.
- 5 Click **OK** to save your changes and close the dialog. The grid lines in the labels are displayed.






To change the grid line colour, use the **Colours > General** page of the **Options** dialog (see page 58).

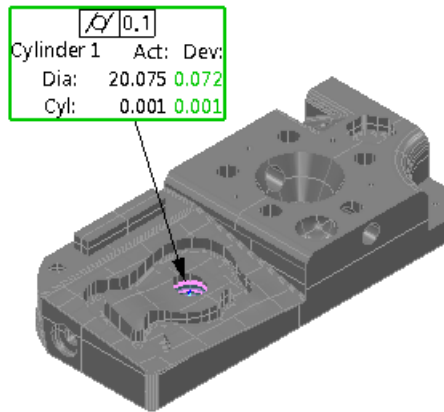
Displaying form tolerance requirements in labels

With the introduction of the **Control frame** option (see page 43) in the **Session Label Settings** dialog, you can now display a summary of the form tolerance requirements for a Geometric item at the top of its label. Form tolerances include, Circularity, Conicity, Cylindricity, Flatness, Profile, Sphericity, and Straightness.

To display a summary of the form tolerance requirements in the labels of Geometric items:

- 1 Display the labels in the CAD view.
- 2 In the **View Options** toolbar, click the **Show Global Label Settings**
 button. The **Session Label Settings** dialog is displayed.
- 3 Select the **General** tab.
- 4 In the **Rows** area, select the:
 - a **Control frame** check box.
 - b **Geom tolerance** check box.
- 5 Click **OK** to save your changes and close the dialog.
- 6 In the CAD view, right-click the label for a Geometric item, and then select **Edit Item** from the context menu. The item's definition dialog is displayed.
- 7 Select the **Output in report** check box.
- 8 Click  next to the form tolerance property. The button changes to .

- 9 Click **OK** to save your changes and close the dialog. PowerINSPECT CNC summarizes the form tolerance requirements for the item in a control frame at the top of the item's label. For example:



- 10 Repeat steps 6 to 9 to continue displaying form tolerance requirements in the labels for other Geometric items.

Point cloud enhancements

Several improvements have been made to PowerINSPECT CNC's point cloud support:

- The **Import Points Wizard** now enables you to import point clouds from Triangle STL and DMT Triangle formats.
- The **Point Cloud Tool** dialog includes a new option that allows you to create multiple point cloud items during a single scanning session (see page 49).
- The **Point Cloud Acquisition Options** dialog includes a new parameter that enables you to further control the full-screen mode's zoom function during scanning (see page 51).
- The **Point Cloud Features** toolbar now includes a Point Cloud Ellipse item, which enables you to create and measure an ellipse using point cloud data (see page 52).
- The inspection sequence now displays the number of points and scanlines used to measure a Point Cloud Feature item (see page 55).




Point clouds are available only as a cost option.

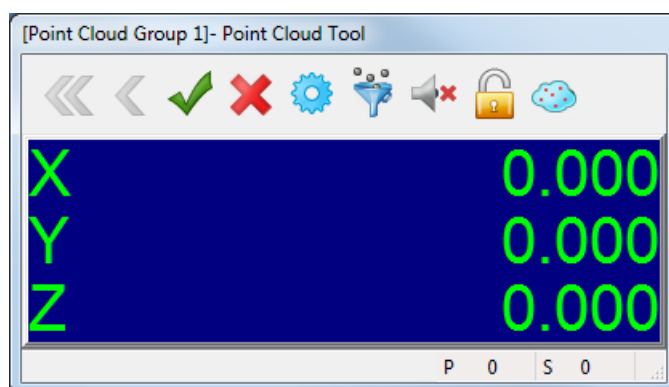
Scanning point clouds

Enhancements to the **Point Cloud Tool** dialog mean you can now save points in new point cloud items each time you pause and resume scanning. This allows you to create several point cloud items during one scanning session without having to leave the full-screen view.


To create multiple point cloud items during a single session:


- 1 Create a point cloud group.

- 2 Open the group, and then click the **Point Cloud**  button in the toolbar. The CAD view switches to full-screen mode and the **Point Cloud Tool** dialog is displayed.



- 3 Use the buttons in the toolbar to specify the **Parameters**, **Acquisition 3D Filter**, **Laser Proximity Sound**, and **Lock View** settings.

- 4 Select the **Turn Committing On Pause**  button to create a new point cloud item each time you pause and resume scanning. The

button changes to .




*To select the button by default, select the **Commit point clouds on pausing** check box in the **Full Screen Measurement** page of the **Options** dialog (see page 58).*

- 5 Hold the laser probe perpendicular to the feature, project the laser line onto the part, and then press the arm's trigger button.
- 6 Move the probe over the part to record points ensuring the laser line is broken by the feature.

- 7 When you have completed scanning the part, press the trigger button again. The scanned points are added to a point cloud item in the inspection sequence.



To delete the points in the last point cloud, click .

Alternatively, click  to delete all points taken.

- 8 Repeat steps 5 to 7 to create another point cloud item.



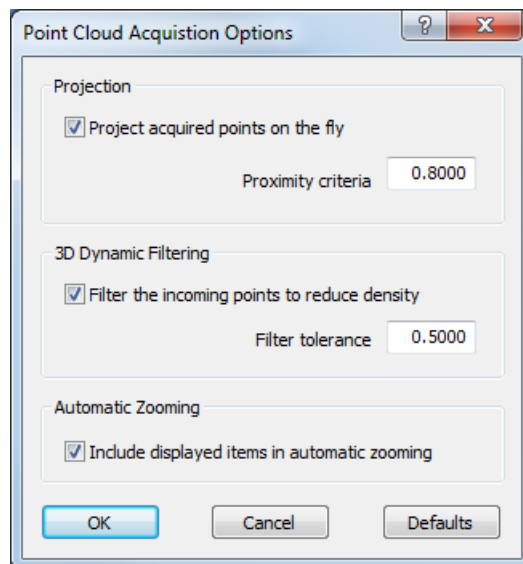
- 9 When you have finished, click  to save the points, close the dialog, and return to the CAD view.

Resizing point clouds

A new **Automatic Zooming** option in the **Point Cloud Acquisition Options** dialog enables you to control which points are displayed in the full-screen view as you scan a part.

To specify the zoom parameters:

- 1 In the **Point Cloud Tool** dialog, click the **Parameters**  button. The **Point Cloud Acquisition Options** dialog is displayed.



- 2 Specify the **Projection** and **3D Dynamic Filtering** parameters.
- 3 As you scan the part, the full-screen view automatically adjusts to display the acquired points. If you want the full-screen view to also display all visible point clouds, select the **Include displayed items in automatic zooming** check box.
- 4 Click **OK** to save your changes and close the dialog.

Point Cloud Ellipse items

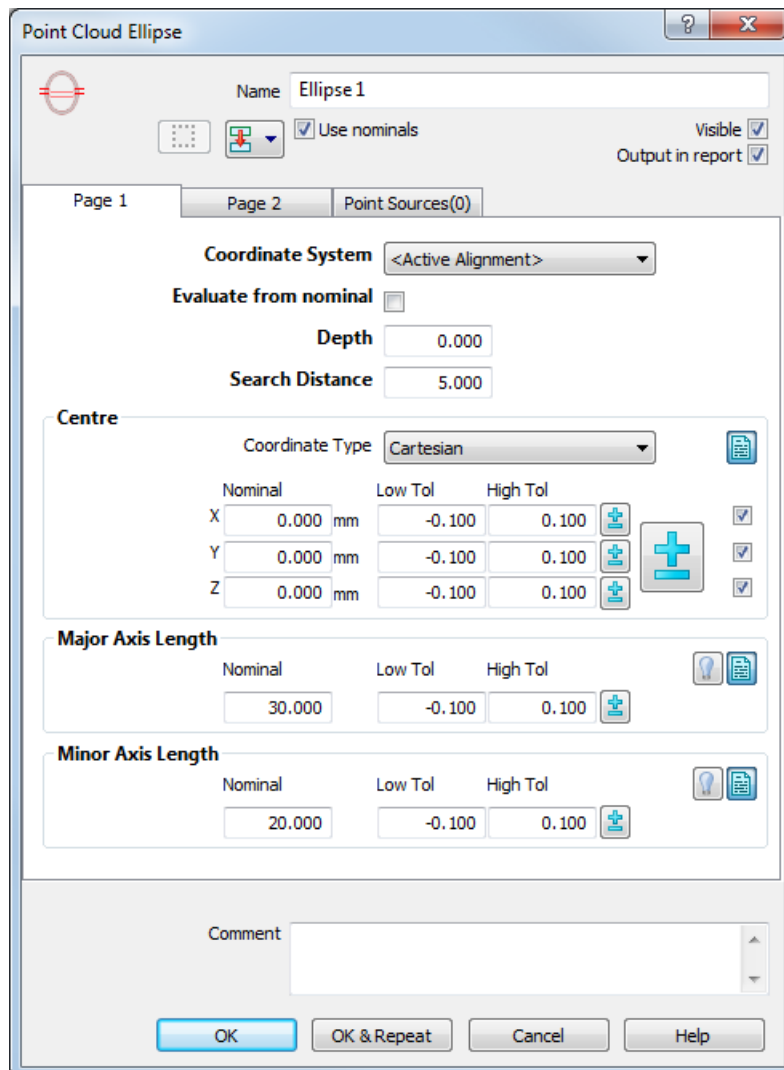
A new Point Cloud Ellipse item has been added to the **Point Cloud Features** toolbar. Use it to enable you to measure an ellipse within a point cloud.

To create a Point Cloud Ellipse item:

- 1 In the **Point Cloud Features** toolbar, click the **Point Cloud Ellipse**



button. The **Point Cloud Ellipse** dialog is displayed.

The dialog box for creating a Point Cloud Ellipse item. It includes fields for Name, Coordinate System, Evaluate from nominal, Depth, Search Distance, Centre (with X, Y, Z coordinates and tolerances), Major Axis Length, and Minor Axis Length. It also has checkboxes for Visible and Output in report, and a Comment field at the bottom.

Point Cloud Ellipse

Name:

☐ ☒ Use nominals Visible ☒ Output in report ☒

Page 1 | Page 2 | Point Sources(0)

Coordinate System:

Evaluate from nominal: ☐

Depth:

Search Distance:

Centre

Coordinate Type:

	Nominal	Low Tol	High Tol
X	<input type="text" value="0.000 mm"/>	<input type="text" value="-0.100"/>	<input type="text" value="0.100"/>
Y	<input type="text" value="0.000 mm"/>	<input type="text" value="-0.100"/>	<input type="text" value="0.100"/>
Z	<input type="text" value="0.000 mm"/>	<input type="text" value="-0.100"/>	<input type="text" value="0.100"/>

Major Axis Length

Nominal	Low Tol	High Tol
<input type="text" value="30.000"/>	<input type="text" value="-0.100"/>	<input type="text" value="0.100"/>

Minor Axis Length

Nominal	Low Tol	High Tol
<input type="text" value="20.000"/>	<input type="text" value="-0.100"/>	<input type="text" value="0.100"/>

Comment:

OK OK & Repeat Cancel Help


- 2 Enter a **Name** for the item.
- 3 To display the item in the CAD view, select the **Visible** check box.
- 4 To include the item in the report, select the **Output in report** check box.
- 5 Select the **Coordinate system** relative to which the item's measurements are to be reported.

- 6 To compare the measurements to their nominal values:
 - a Select the **Use nominals** check box.
 - b In the **Centre** area, enter the nominal and tolerances for the position of the ellipse's centre.
 - c In the **Major axis length** area, enter the nominal length and tolerances of the ellipse's longest axis.
 - d In the **Minor axis length** area, enter the nominal length and tolerances of the ellipse's shortest axis.
 - e In the **Main axis** area, enter the nominal and tolerances for the orientation of the major axis.
 - f In the **Normal vector** area, enter the nominal vector and tolerances for the normal of the scanned surface.




*Unlike 2D features created using conventional probes, laser-scanned 2D features are not projected onto a reference plane. Instead, the reference plane is determined by the area around the feature; the **Normal vector** specifies the orientation of this plane.*



*Alternatively, click  and select **from CAD Entity** to load the nominals from the model, or select **from Active Measure** to load them from the current Measure.*

- 7 By default, PowerINSPECT CNC searches the entire point cloud for the feature. To limit the search for the ellipse to the region around its nominal position:
 - a Select the **Evaluate from nominal** check box.
 - b Enter a **Depth** to specify the distance from the feature's plane at which scanlines are considered.
 - c Enter a **Search distance** to specify the maximum distance from the feature's nominal position within which you want PowerINSPECT CNC to search.
- 8 In the **Profile** area, enter the **Maximum** distance from the nominal ellipse at which scanned points remain in-tolerance. If the deviation of a point exceeds this value, it is reported as out-of-tolerance.
- 9 To include any extra information about the item, enter a **Comment**.

- 10 To use the points from other items to measure this item, select the **Point Sources** tab, and then select the items that contain the points you want to use.
- 11 Click **OK** to close the dialog and create the item in the inspection sequence. The details of the feature are displayed in the **Info** tab, the **Report** tab, and in the inspection sequence when you click  next to the item's entry.


To detect the ellipse, you must pass the laser probe over the feature so that the scanlines cover the outline. The number of scanlines required depends on the size of the feature.

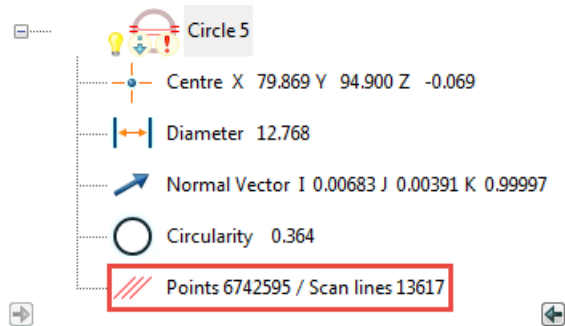


You can create a Point Cloud Ellipse item using the Geometry Explorer when you select an ellipse feature from the CAD model (see page 15).

Viewing points and scanlines

The number of points and scanlines used to measure a Point Cloud Feature item is now displayed in the inspection sequence.

To view the details of a Point Cloud Feature item, click  next to the item's entry in the inspection sequence. For example:



Displaying dual-column probe details

The **Machine Configuration** dialog now enables you to switch columns when using a dual-column device in manual mode. In addition, the **Status** display box and **Tracking Box** button have been added to the dialog, giving you the ability to see the connection status and probe position for the selected column.



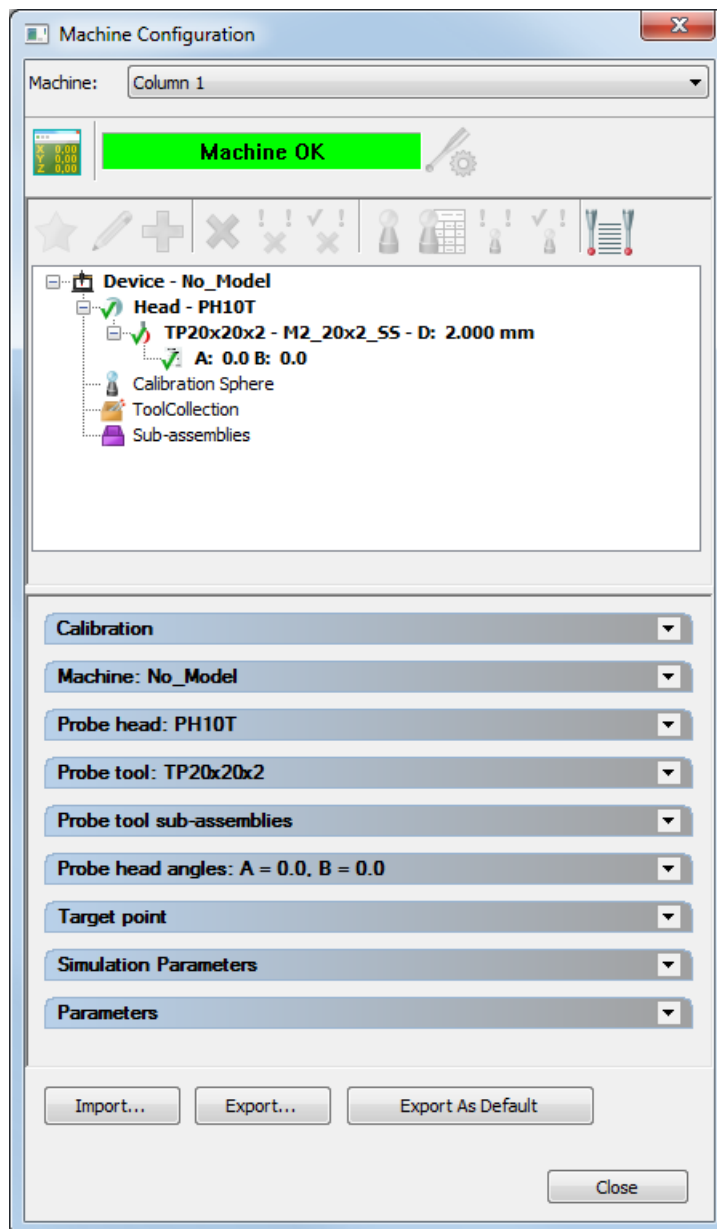
Dual-column is available only as a cost option.

To view the column details during manual probing:

- 1 Play the inspection sequence. PowerINSPECT CNC switches to full-screen mode, displays the **Feature Probing** dialog for the first item and displays the probe specified for that item.

- 2 In the **Full Screen Options** toolbar, click the **Edit Database** button. The **Machine Configuration** dialog is displayed, for example:





- 3 By default, the dialog displays the details of the probe on the last column you selected. Select the other column in the **Machine** list to switch to its details and to display its probe in the full-screen view.

- 4 To show the position of the probe as it moves, click the **Tracking**



Box button. The **Coordinate** dialog is displayed.



*When the **Coordinate** dialog is displayed and you switch columns, the dialog remains open and continues to display the position of the probe on the previously selected column.*

- 5 To return to the original column, select its entry in the **Machine** list.
- 6 When you have finished, click **Close** to close the dialog.

PowerINSPECT CNC settings

Several changes have been made to the settings in the **Options** dialog. To view and change the settings, select the **Tools > Options** menu option to display the dialog, and then select the page you want to work with.

Display options page

By default, PowerINSPECT CNC determines if a measured item is within tolerance using the item's exact calculated error value. The new **Use rounded error value to determine measure state** check box enables you to determine the tolerance status using the error value rounded to the specified **Number of decimal places**.



Using rounded error values is useful when the accuracy of the measuring device is limited to a small number of decimal places.

Colours > General page

A new option on the **Colours > General** page enables you to change the colour of the grid lines (see page 45) shown in labels.

To change the colour of the grid lines:

- 1 In the list, select **Label gridline colour**.
- 2 Click the **Main Colour** swatch.
- 3 In the **Color** dialog, select an option, and then click **OK** to save your changes and close the dialog.

Colours > Entities page

When you display labels for GD&T items (see page 43), an asterisk marker terminates the arrow that links the label to the feature. A new option on the **Colours > Entities** page enables you to change the colour of the marker.

To change the colour of the marker:

- 1 In the list, select **Label anchor**.
- 2 Click the **Main Colour** swatch to specify the colour of the marker.
- 3 In the **Color** dialog, select an option, and then click **OK** to save your changes and close the dialog.
- 4 Click the **Highlight Colour** swatch to specify the highlight colour of the marker when the GD&T item is selected in the inspection sequence, or the GD&T label is selected in the CAD view.
- 5 In the **Color** dialog, select an option, and then click **OK** to save your changes and close the dialog.



*To highlight the marker using the specified **Highlight Colour**, select the **Highlight entities using highlight colour** check box. Deselect the check box to highlight the marker by changing the luminosity of the specified **Main Colour**.*

Colours > Tolerance band page

A new option on the **Colours > Tolerance band** page enables you to change the colour of the text in the tolerance scale for inspection groups and point clouds when they are displayed in the CAD view and in the report.

To change the colour of the text:

- 1 Click the **Text Colour** swatch.
- 2 In the **Color** dialog, select an option, and then click **OK** to save your changes and close the dialog.



*To revert to the original colour, click **Reset**.*

Full screen measurement page

A new option on the **Full screen measurement** page enables you to select the **Turn Committing On Pause** button in the **Point Cloud Tool** dialog (see page 49) by default.

To select the button by default, select the **Commit point clouds on pausing** check box.

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