
PowerINSPECT OMV 2016

What's New



PowerINSPECT OMV

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New features

PowerINSPECT OMV 2016 contains the following new features and enhancements:

- **Working with CAD models** (see page 2) — Several improvements have been made to the way you load, manage, and view the details of models.
- **Changes to inspection groups** (see page 7) — To improve usability, a new type of inspection group has been created for measuring with probe paths, which replaces the On-the-fly Surface Points group.
- **Specifying nominals** (see page 10) — You can now specify the nominal vector for a Probed Parallel Plane item.
- **Displaying CAD nominals for ellipses** (see page 12) — Ellipses are now selectable using the Wireframe Checker. This enables you to view their nominal values in the **Geometry Explorer** tab.
- **Label enhancements** (see page 15) — Several improvements have been made to the formatting of labels.
- **Section enhancements** (see page 18) — Several improvements have been made to the way you display sections and view their measured results.
- **PowerINSPECT OMV settings** (see page 29) — The **Options** dialog contains several new features, including, new colour options, and the ability to specify whether PowerINSPECT OMV 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 3).
- A new button in the **CAD** tab enables you to highlight the selected entry in the CAD view (see page 4).
- New icons enable you to identify CAD objects by type (see page 5).

Loading CAD models

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

To load a model into PowerINSPECT OMV:

- 1 In Windows Explorer, locate the model file.
- 2 Left-click and hold the file to select it.
- 3 Drag the file into the main PowerINSPECT OMV graphics window.
- 4 When the cursor changes to , release the mouse button. If no documents are open, PowerINSPECT OMV creates a document and adds the model to it.


In addition, when you load a model into an inspection document, PowerINSPECT OMV 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.*

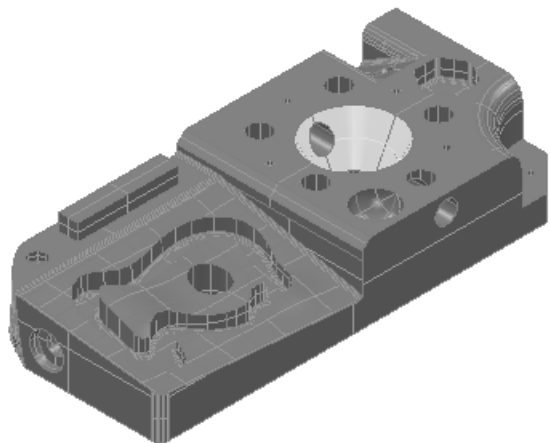
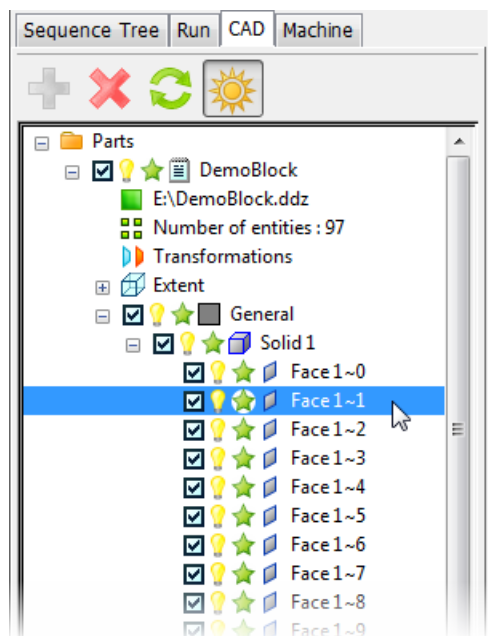
Highlighting objects in CAD models

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 OMV 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 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

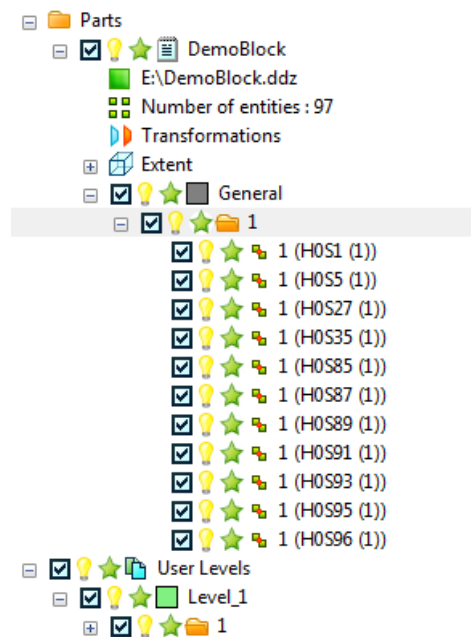
In PowerINSPECT OMV 2016, the **CAD** tab displays an icon next to each constituent part of a model. This makes it easier to identify the type of objects you select and work with.

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

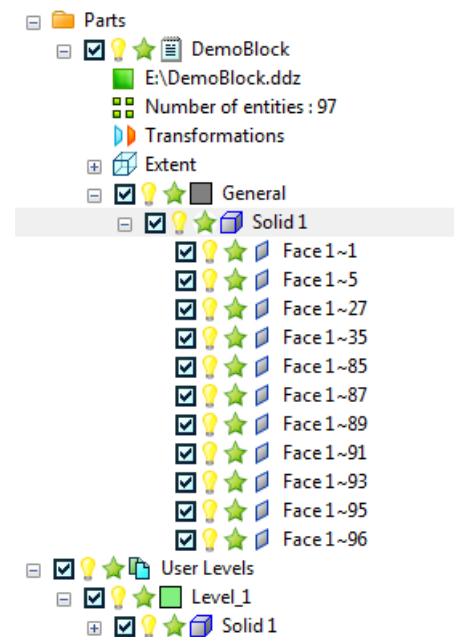
Similarly, parent objects are now distinguished as Groups  or Solids , instead of being marked by folder  icons.

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

PowerINSPECT OMV 2015 R2



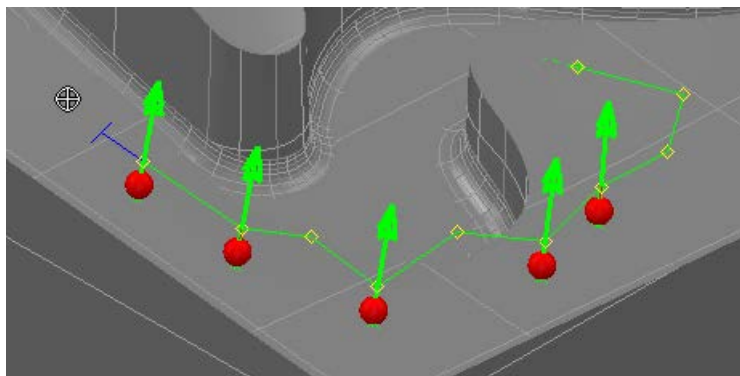
PowerINSPECT OMV 2016








Changes to inspection groups

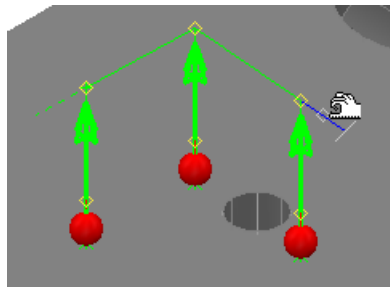
To improve usability, a CNC Surface Points group has been added to the **OMV Item** toolbar. It replaces the On-the-fly Surface Points group. As with the previous group, you can specify the probe path, however, you can now create guided surface points. This enables you to automate the inspection of surfaces on a part that can be repeated across Measures. You can also use the guided surface points as constraining features in RPS alignments.

When you select a **UserDefined** probing method, PowerINSPECT OMV 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 **OMV Item** 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 OMV 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.
- 6 To view and update the group settings in the **Inspection Group: Surface Points** dialog, right-click the group in the inspection sequence, and then select **Modify Item** from the context menu.

Specifying nominals

In PowerINSPECT OMV 2016, a **Normal Vector** area has been added to the **Probed Parallel Plane** dialog. Use it to specify the nominal vector of the plane's perpendicular. This brings the Probed Parallel Plane item into line with the Probed Plane item.

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
J	0.00000	0.00000	0.00000
K	1.00000	0.00000	0.00000






Distance

	Nominal	Low Tol	High Tol
	-10.000	-0.100	0.100

Comment:

OK Apply Cancel Help


To specify the nominals for a Probed Parallel Plane item:

- 1 In the inspection sequence, double-click the item's name. The **Probed Parallel Plane** dialog is displayed.
- 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.

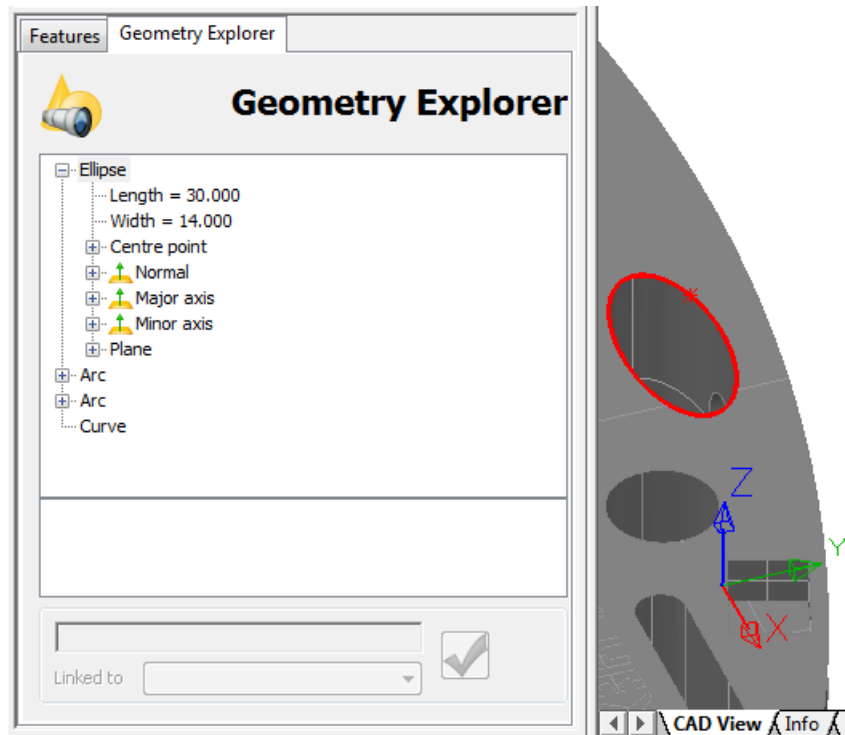
Displaying CAD nominals for ellipses

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.

To display the nominals of an ellipse 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. For example:



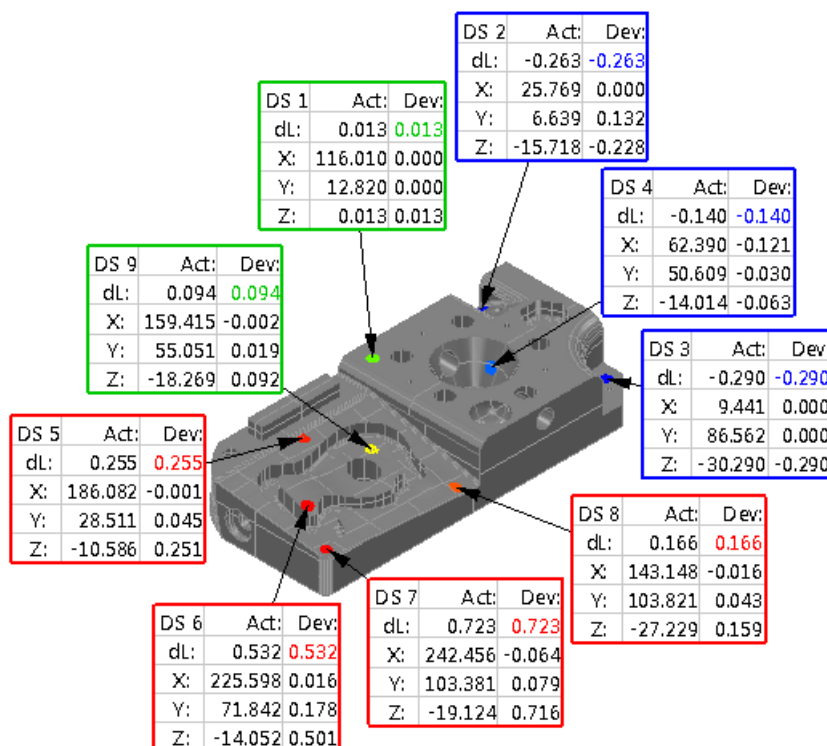
Label enhancements

New options have been added to the **Session Label Settings** dialog, which enable you to further customize the display of labels. Within labels, you can now display:


- grid lines to make it easier to view the displayed information (see page 15).
- a summary of the form tolerance requirements for Geometric items (see page 16).

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 29).

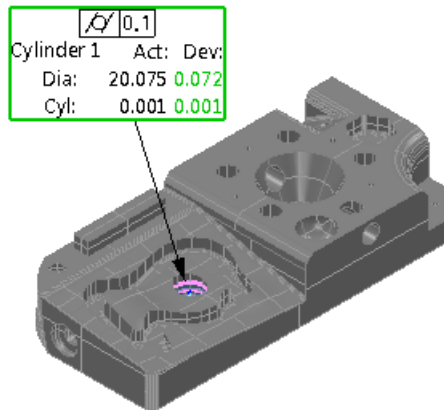
Displaying form tolerance requirements in labels

With the introduction of the **Control frame** option 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 OMV 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.

Section enhancements


PowerINSPECT OMV enables you to load documents that contain section groups. In PowerINSPECT OMV 2016, several changes have been made to how you display sections and their measured results:


- A new shading mode option enables you to control how models with sections are displayed in the CAD view (see page 19).
- To improve the display of sections, the Section Group view and its toolbars have been redesigned (see page 21).
- 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 24).
- 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 26).

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**

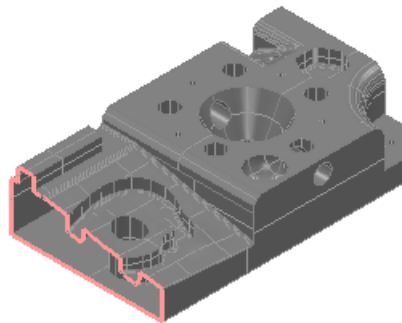
Mode  button. The pull-out toolbar is displayed.

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



to hide the part of the model that appears nearest.

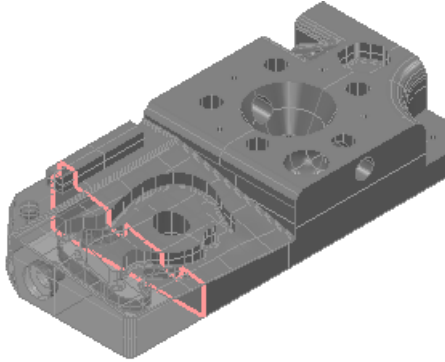
For example:





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.*



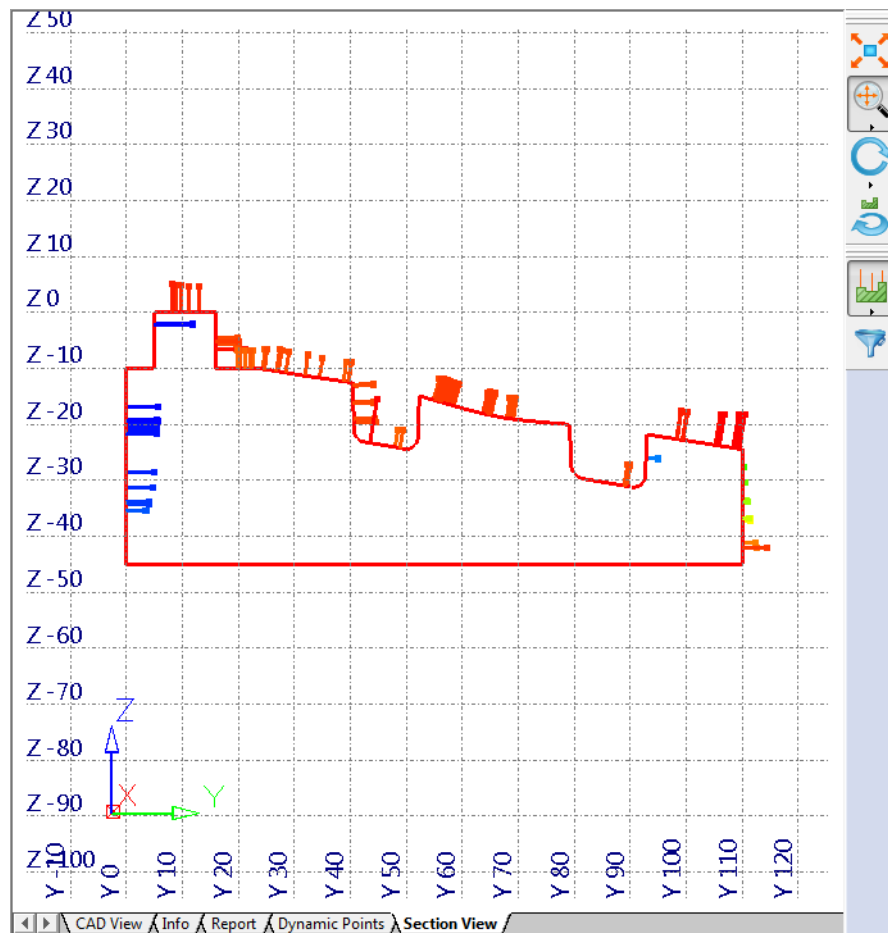
to disable the shading mode.

Section view

Previously, if your document contained section groups, a **Section Group** tab for each group was displayed at the bottom of the graphics window. In PowerINSPECT OMV 2016, a single **Section View** tab is displayed, and it shows the details of the currently selected section group in the inspection sequence.

You can use the updated **Section View** toolbars (see page 22) 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 29).

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 24) 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 26) 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

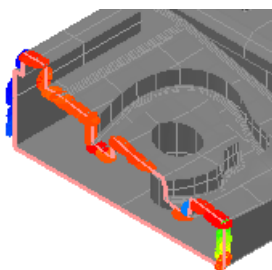
the **Section Measurement Display**  button. The pull-out toolbar is displayed.

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



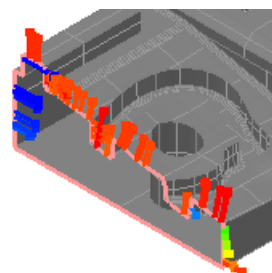
to display the deviation of each point as **Spot Confetti**.

For example, in the CAD view:



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

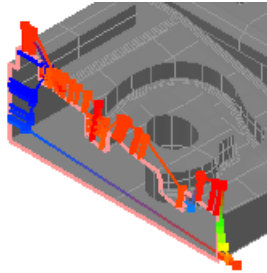
For example, in the CAD view:





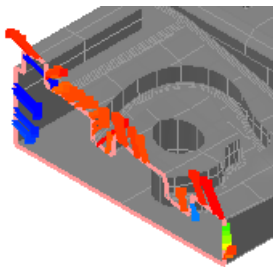
to display the deviation of each point as **Connected Deviation Lines**.

For example, in the CAD view:



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

For example, in the CAD view:



PowerINSPECT OMV 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.






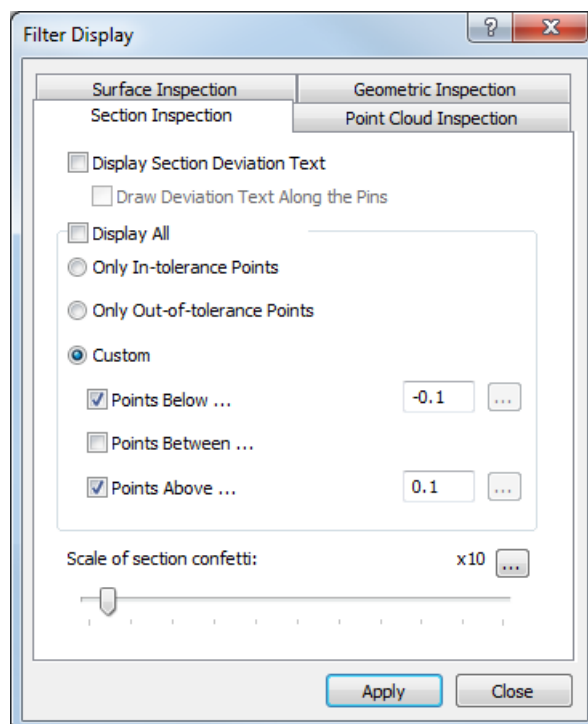
*To show more details for each measurement, select the **Display section deviation text** check box in the **Section Inspection** tab of the **Filter Display** dialog (see page 26).*

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 OMV 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.

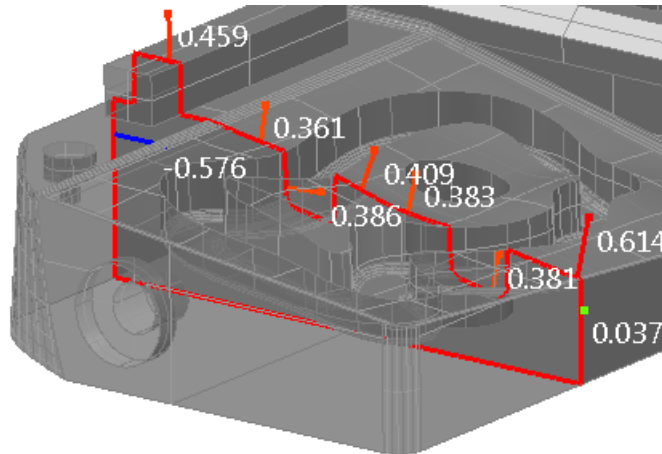
To specify the settings for the display of probed section points:

- 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 24).
- 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.



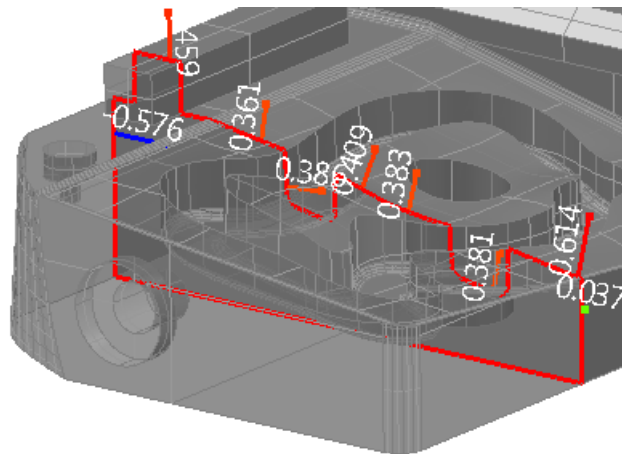
- 5 To show the calculated deviation of each probed point alongside its confetti, deviation line, or vector, select the **Display section deviation text** check box.



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 OMV 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.

PowerINSPECT OMV 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 OMV 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 15) 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, 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**.*

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