
PowerINSPECT OMV Pro 2016

What's New



PowerINSPECT OMV Pro

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New features

PowerINSPECT OMV Pro 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 OMV Pro 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, probe paths are no longer supported by On-the-fly Surface Points groups.
- **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 create a Probed Ellipse item 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.

- **Label enhancements** (see page 34) — Improvements have been made to the formatting of labels, and you can now display labels for GD&T items.
- **PowerINSPECT OMV Pro settings** (see page 40) — The **Options** dialog contains several new features, including, new colour options, and the ability to specify whether PowerINSPECT OMV Pro 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 OMV Pro by dragging and dropping files into the main graphics window.

To load a model into PowerINSPECT OMV Pro:

- 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 OMV Pro graphics window.

- 4 When the cursor changes to , release the mouse button. If no documents are open, PowerINSPECT OMV Pro creates a document and adds the model to it.

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

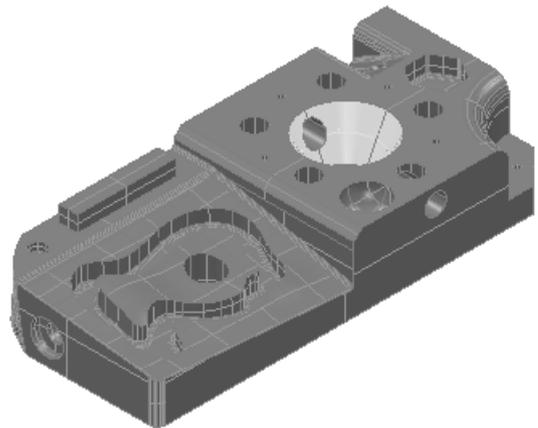
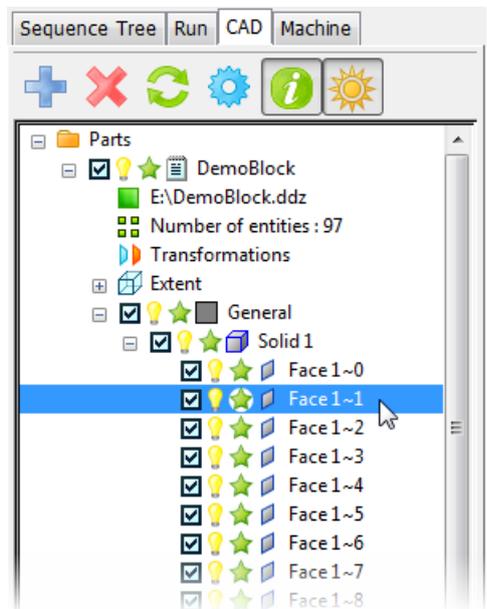
Highlighting objects in CAD models

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 Pro 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

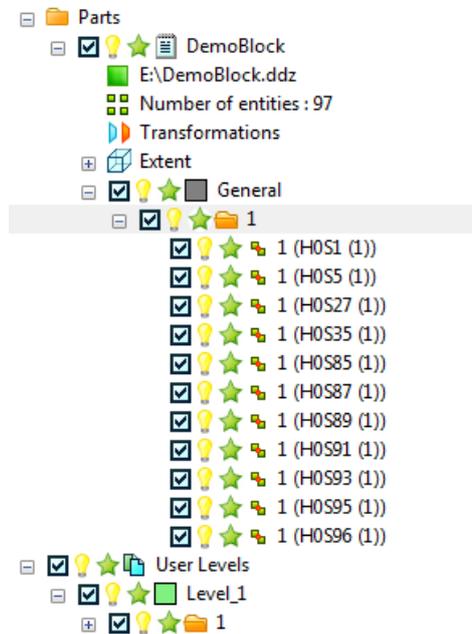
In PowerINSPECT OMV Pro 2016, the Detailed View of the **CAD** tab, the **CAD Object Visibility** dialog, the **CAD Object Selection** dialog, and the **CAD Object Verification** dialog display an icon next to each constituent part of a model. This makes it easier to identify the type of objects you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

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

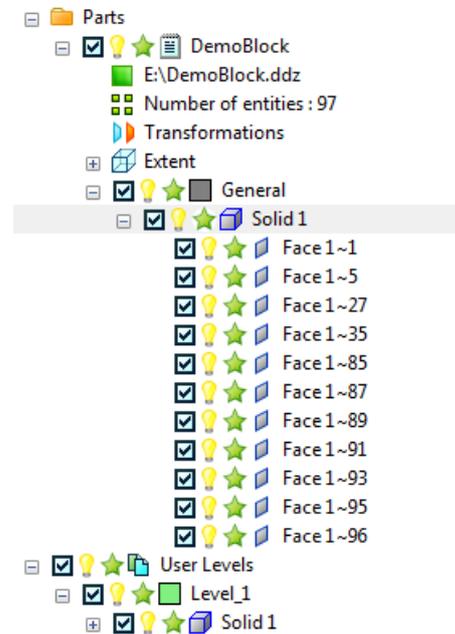
Similarly, parent objects are now distinguished as Groups  or Solids , instead of being marked by folder  icons.

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

PowerINSPECT OMV Pro 2015 R2



PowerINSPECT OMV Pro 2016



Automatic collision avoidance mode

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

To enable PowerINSPECT OMV Pro 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 OMV Pro:
 - 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 OMV Pro 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, Guided Edge Points groups are no longer available, and probe path support for On-the-fly Surface Points groups is no longer supported; however, you can continue to create On-the-fly Surface Points groups and use point sources to measure them.

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.

The CNC Edge Points group replaces the Guided Edge Points group and works in the same way. Use the new group when you want to automate the inspection of surface edges by specifying the probe path.



When you open a document in PowerINSPECT OMV Pro 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.

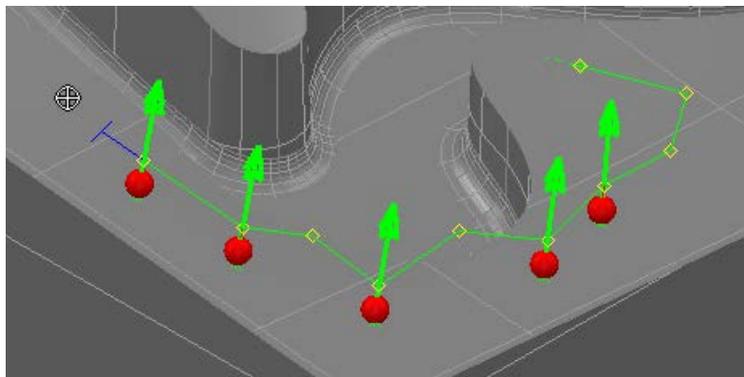
Creating inspection points from dynamic points

As in previous versions of PowerINSPECT OMV Pro, you can continue to create guided surface and guided edge points from dynamic points. Guided surface points are added to CNC Surface Points groups, and guided edge points are added to CNC Edge Points groups.

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 OMV Pro 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.

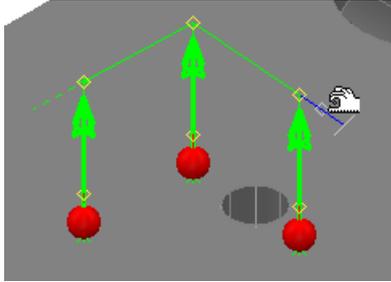


To create the probe path for a CNC Surface Points group using guided surface points:

- 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 OMV Pro 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.

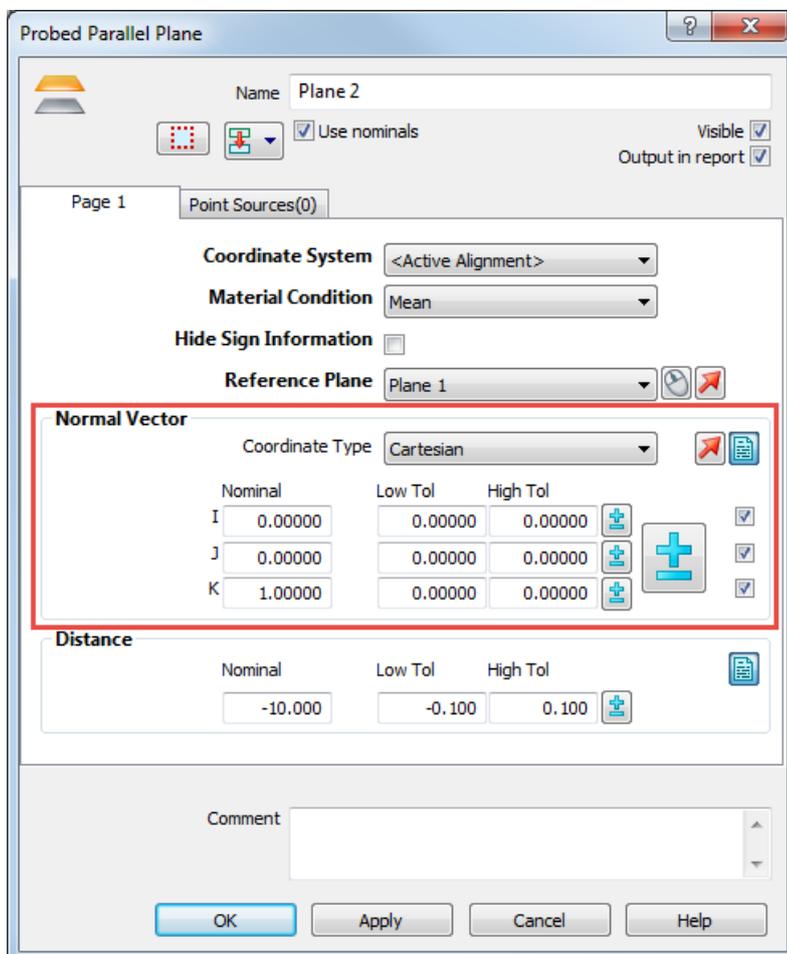


*To edit the probe path created using another method, select the group in the inspection sequence, click  in the group's **Surface Inspect** dialog, and then select a **UserDefined** method.*

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:



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.

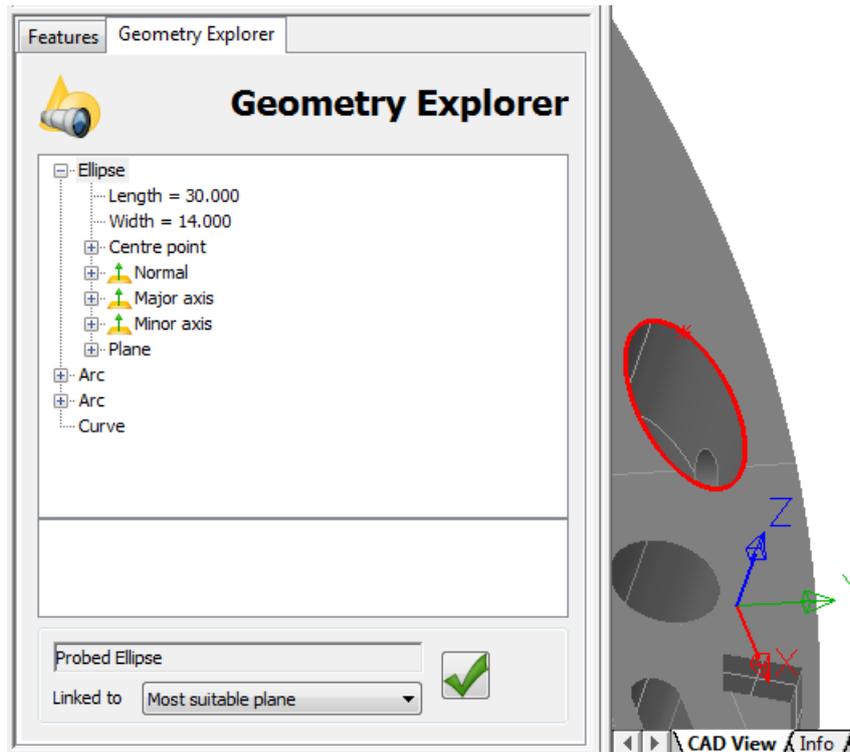
Selecting ellipses

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 Probed Ellipse item in the inspection sequence.

To create a Probed 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.

- Click the wireframe to select the highlighted feature. The ellipse is displayed in red and its nominals are listed in the **Geometry Explorer** tab.



- In the **Linked** to list, select a reference plane.



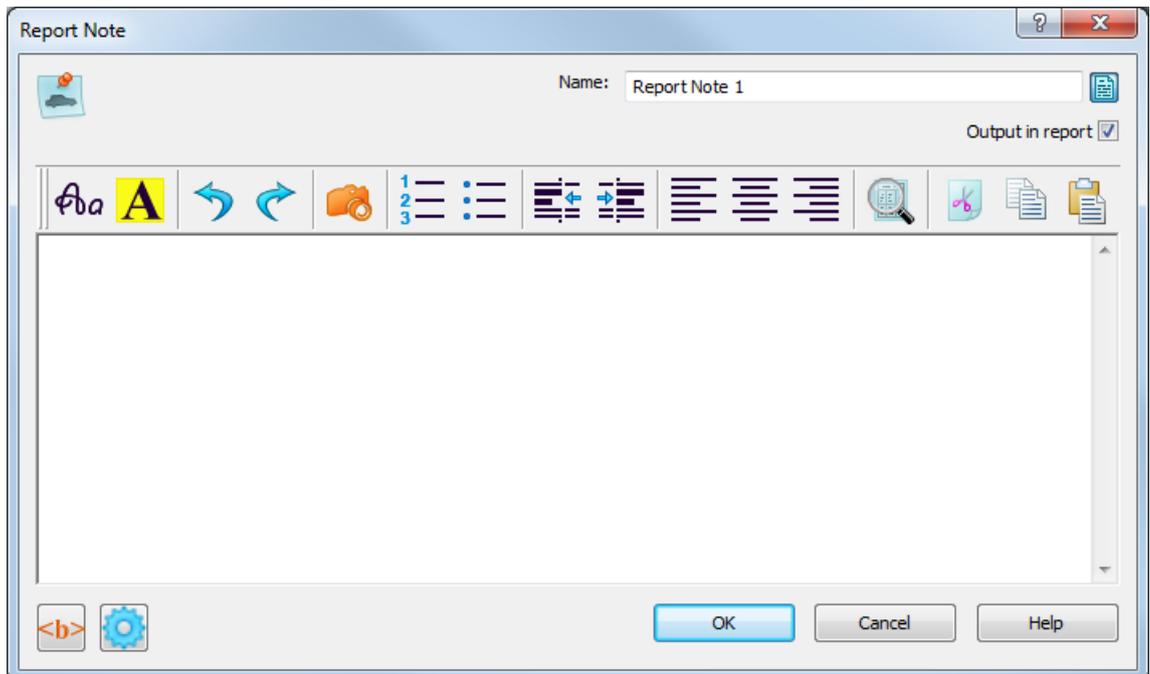
- Click . PowerINSPECT OMV Pro adds the Probed Ellipse item to the inspection sequence.

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.

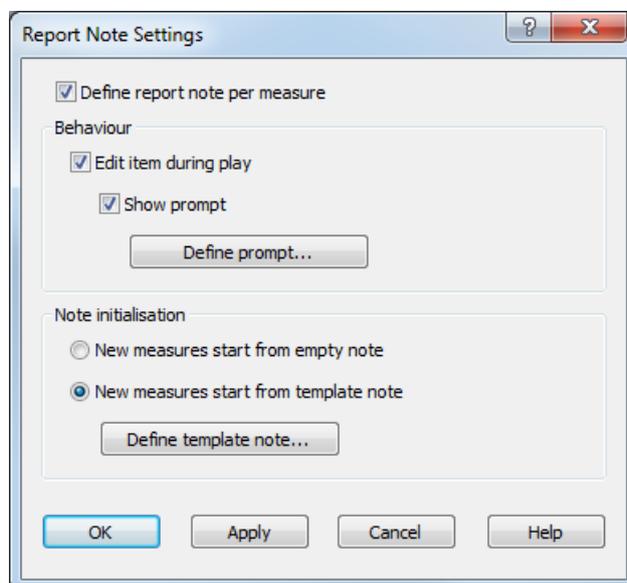
- 6 To edit the HTML source, click . The button changes to .
- 7 By default, the content of the Report Note item is applied to all Measures. To customize the Report Note item for each Measure, and to specify the action of the item when it is imported into PowerINSPECT OMV Pro, click the **Settings**  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 import 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.
- The previously available Section group has been replaced by a new CNC Section group for measuring sections with probe paths, which enables repeatability across different Measures (see page 22).
- A new shading mode option enables you to control how models with sections are displayed in the CAD view (see page 24).
- To improve the display of sections, the Section Group view and its toolbars have been redesigned (see page 26).
- 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 29).
- 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 31).

Changes to section groups

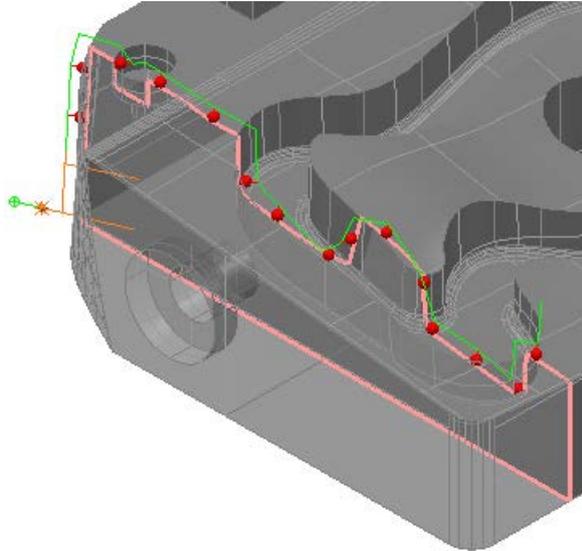
To improve usability, a CNC Section group has been created, which replaces the previously available Section group. The new group combines the ability to specify probe paths with the ability to create guided surface points. This enables you to automate the inspection of sections on a part that can be repeated across Measures. In addition, you can use the guided surface points as constraining features in RPS alignments.

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:

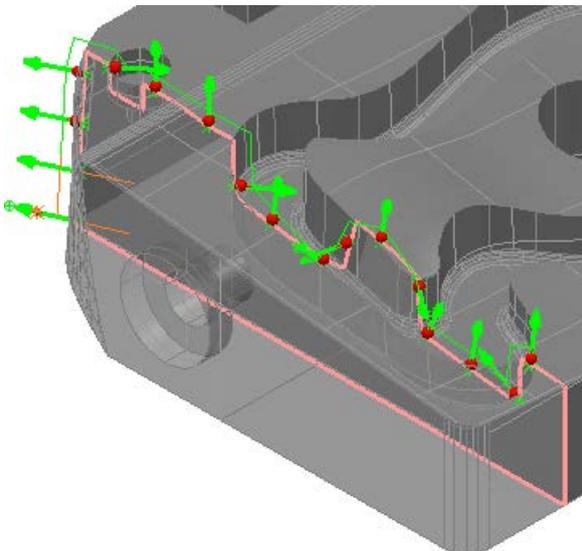
- 1 In the **Features** tab, deselect the **Generate Probe Path**  button to enable the creation of probe paths. The button changes to .
- 2 In the **Item** toolbar, click the **CNC Section**  button.
- 3 A section group is added to the inspection sequence and the **Section Inspect** dialog is displayed in the **Features** tab.
- 4 In the inspection sequence, double-click the group's name to display the **Section** dialog.
- 5 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.
- 6 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.
- 7 Complete the parameters in the **Raster Parameters** dialog, and then click  to save your changes and close the dialog.

- 8 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 OMV Pro 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:

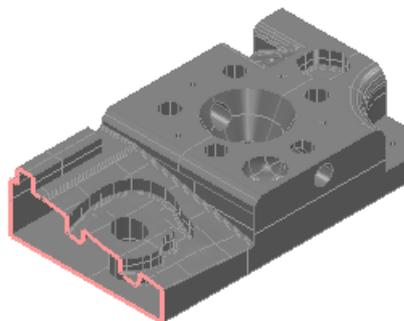
- 1 In the inspection sequence, click  next to the section group you want to work with. The icon changes to .
- 2 Select the section group in the inspection sequence.
- 3 In the **CAD View** toolbar, click  below the **Section Shading**

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.

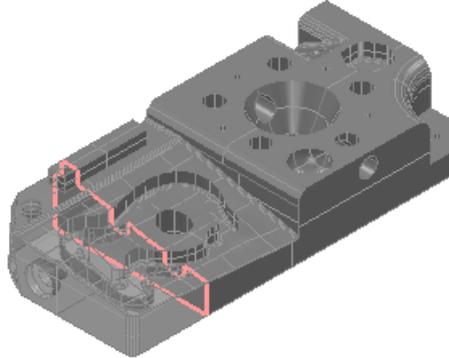
For example:





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 OMV Pro displayed a separate **Section Group** tab at the bottom of the main graphics window for each section group you created. In PowerINSPECT OMV Pro 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 27) 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 40).

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 29) 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 31) 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

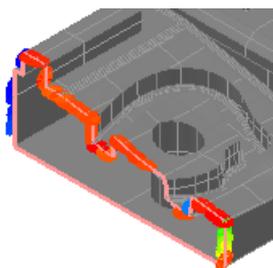
the **Section Measurement Display**  button. The pull-out toolbar is displayed.

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



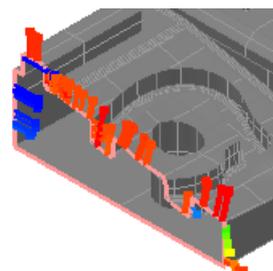
to display the deviation of each point as **Spot Confetti**.

For example, in the CAD view:



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

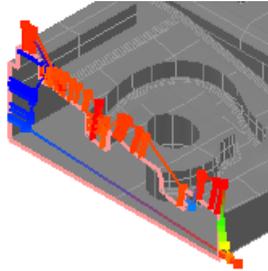
For example, in the CAD view:





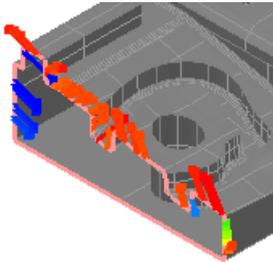
to display the deviation of each point as **Connected Deviation Lines**.

For example, in the CAD view:



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

For example, in the CAD view:



PowerINSPECT OMV Pro 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.



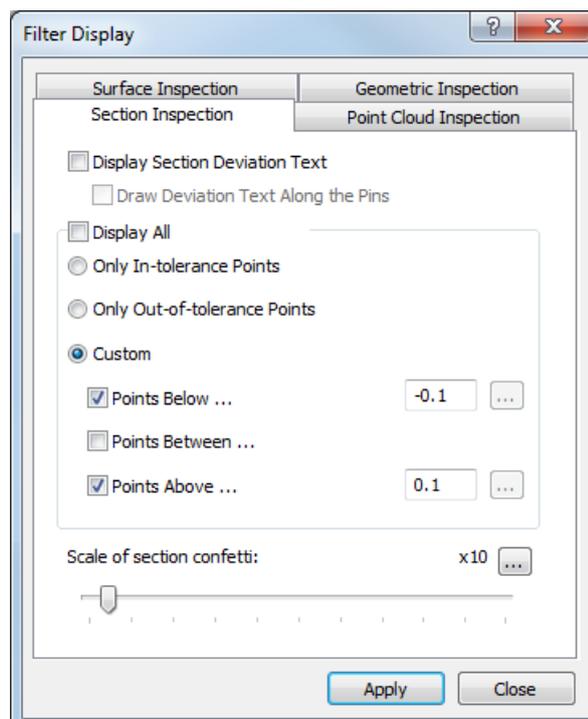
*To show more details for each measurement, select the **Display section deviation text** check box in the **Section Inspection** tab of the **Filter Display** dialog (see page 31).*

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 Pro 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.

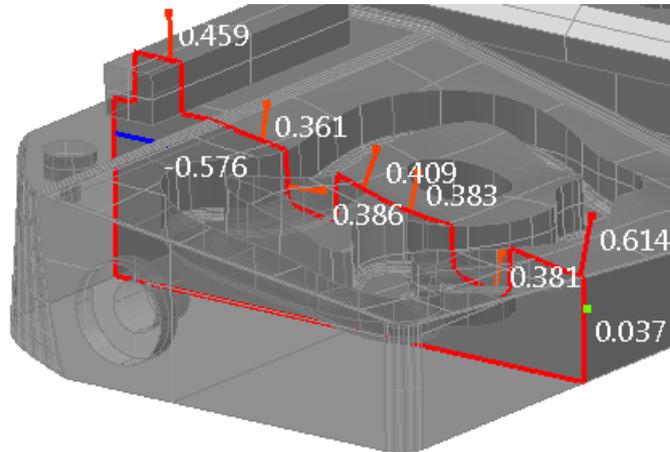
To specify the settings for the display of probed section points:

- 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 29).
- 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.



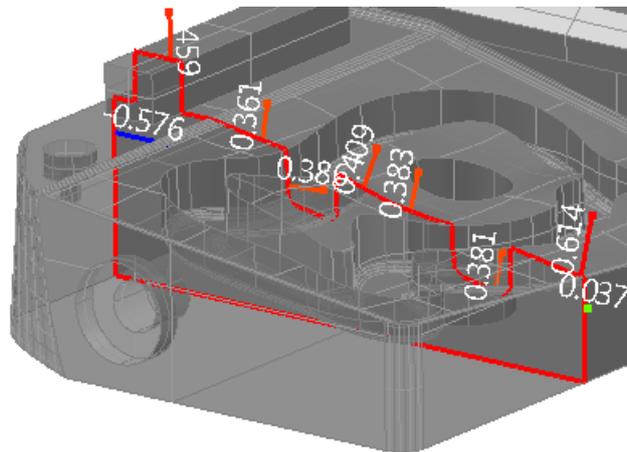
- 5 To show the calculated deviation of each probed point alongside its confetti, deviation line, or vector, select the **Display section deviation text** check box.

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 Pro 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.

Label enhancements

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

- Labels can now be displayed in the CAD view for GD&T items (see page 35).
- The **Session Label Settings** dialog includes a new option that enables you to display grid lines on labels (see page 37).
- The form tolerance requirements for a Geometric item can now be displayed on the item's label (see page 38).

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 OMV Pro displays the **Datum feature symbol**.



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

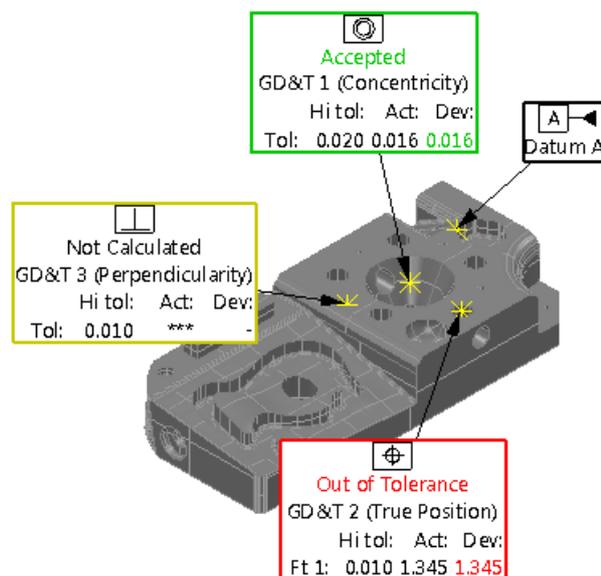
- **GD&T status** check box to display the measurement status for a GD&T item.

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

In the CAD view, PowerINSPECT OMV Pro 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:

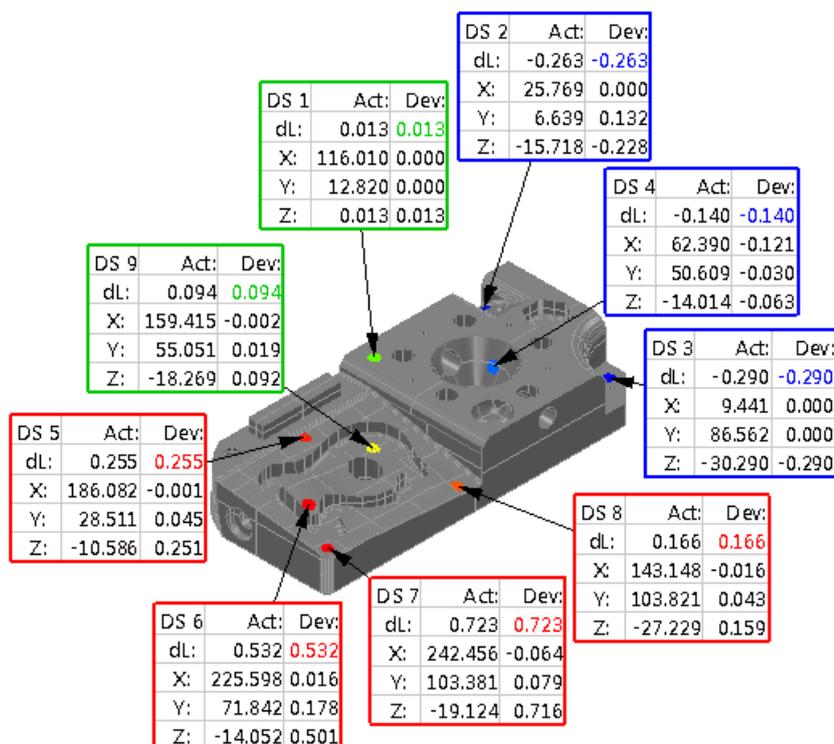
- Green indicates the measurement is in-tolerance.
- Red indicates the measurement is out-of-tolerance.
- Gold indicates the measurement has not been calculated.
- Black indicates the item is unmeasured, for example, a GD&T Datum item.



*To change the colour of the asterisk marker, use the **Colours > Entities** page of the **Options** dialog (see page 40).*

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 40).

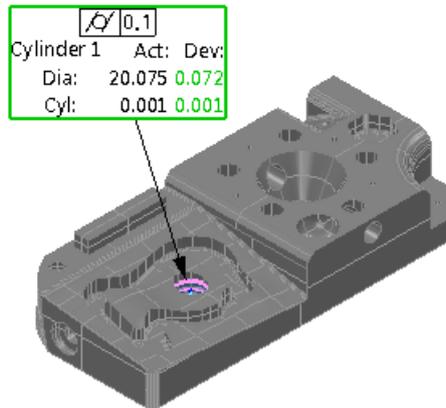
Displaying form tolerance requirements in labels

With the introduction of the **Control frame** option (see page 35) 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 Pro 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.

PowerINSPECT OMV Pro 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 Pro 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 37) 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 35), 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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