
PowerINSPECT Play Only 2016

Quick Start Guide



PowerINSPECT Play Only

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Introduction

PowerINSPECT Play Only enables you to open an existing inspection (created using a full version of PowerINSPECT), to manipulate the CAD model, to measure the part, and to generate a report.


This document describes how to:

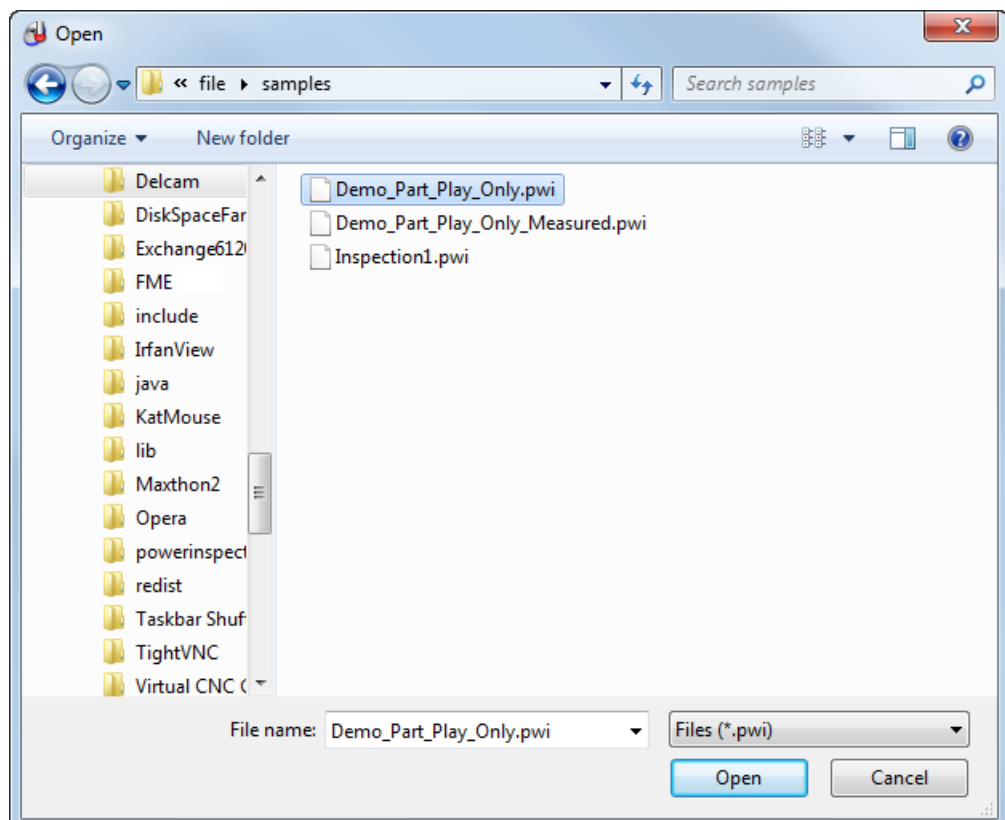
- open an inspection document (see page 2);
- automatic probe compensation (see page 4);
- measure a part (see page 6);
- view and print the report (see page 10);
- understand the screen layout (see page 14);
- manipulate the CAD model (see page 18).

Opening an inspection document

- 1 Double-click the PowerINSPECT Play Only icon on the desktop, or select **Start > All Programs > Delcam > PowerINSPECT > PowerINSPECT Play Only**.

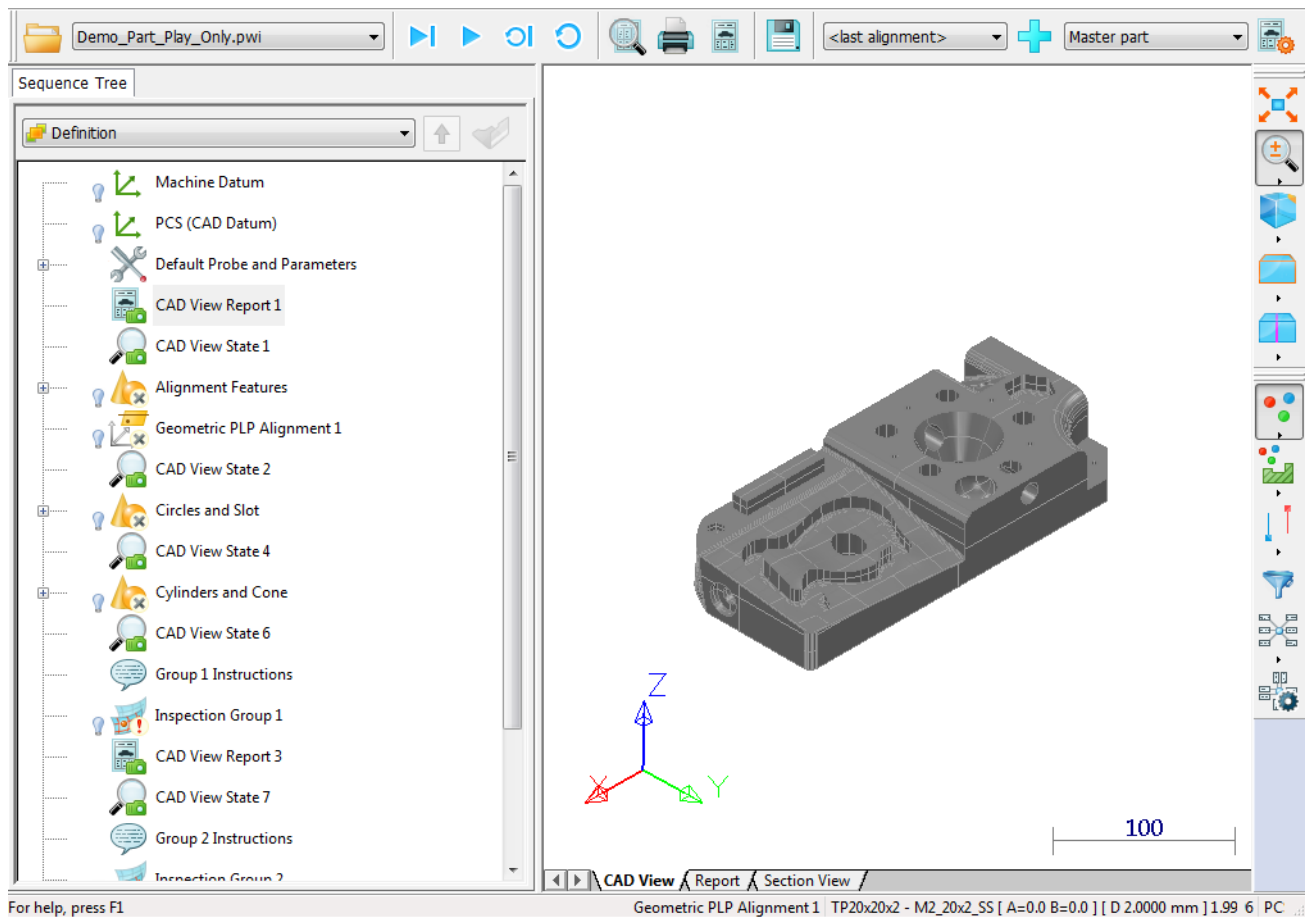


- 2 Click  on the **Main** toolbar.
- 3 In the **Open** dialog, navigate to the **Samples** folder in your PowerINSPECT Play Only installation.



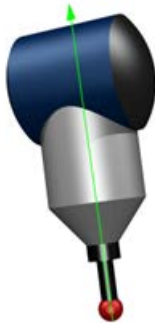
- 4 Select **Demo Part Play Only.pwi**, and click **Open**.

The CAD model is displayed in the **CAD View** tab and the features to be measured are displayed in the Sequence Tree.



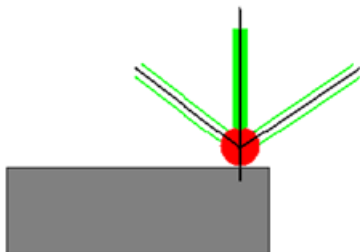
Automatic probe compensation

By default, PowerINSPECT Play Only automatically applies probe compensation in the direction of the vector running from the stylus tip to the last knuckle joint of the arm.



To use this method, you must ensure the probe is pointing at the surface being measured when you take a point. The following diagrams show how to position the probe:

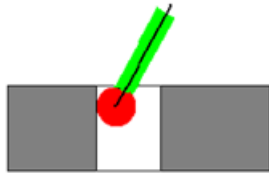
- Correct orientation of the probe relative to the surface.



- Incorrect orientation of the probe relative to the surface.



- When probing inside features, the probe must point towards the surface of the hole as near normal to the surface of the feature as possible. For example:



Start measuring

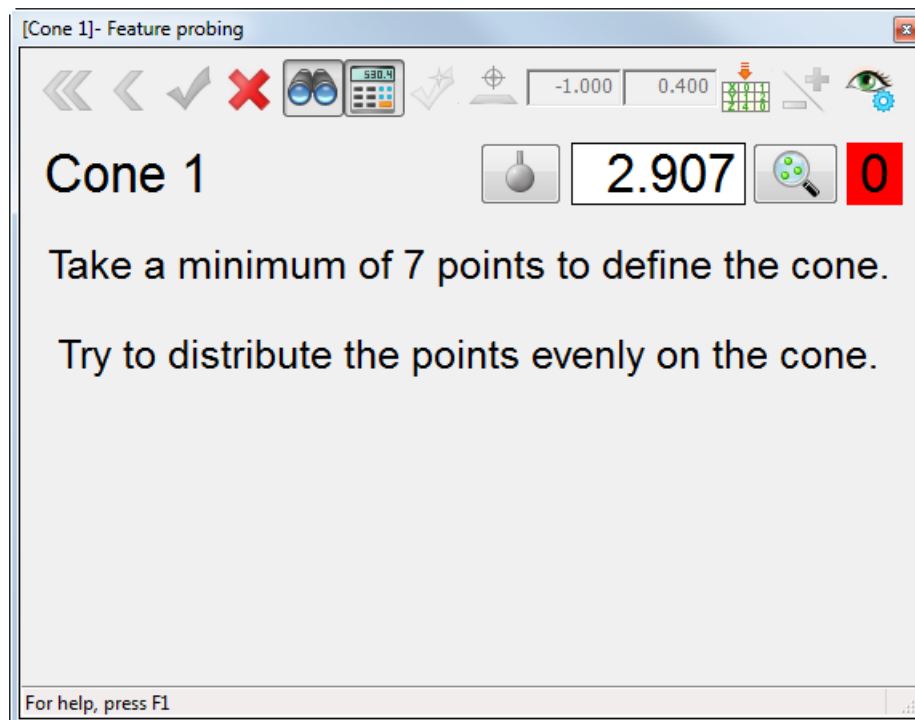
The inspection sequence specifies the features to be measured on the selected part and the order in which the features are to be measured. It can also contain CAD View State items, which change the CAD view during the inspection, and Comment items, which display information and instructions.

To inspect a part:

- 1 Select the Measure in which you want to save the results.

- 2 Click the **Play all**  button.

PowerINSPECT Play Only connects to the measuring device and the CAD view changes to a full screen mode. The **Feature Probing** dialog is displayed. For example:




- 3 Probe each item in the inspection sequence. The number of points you have currently taken is displayed in the box on the right of the **Feature Probing** dialog. The box changes to green when you have probed the required number of points for an item.



To delete a point, click



- 4 When you have probed all the points for an item, click  to save your measurements. In the inspection sequence, an icon is displayed on the item to indicate the results:



— the item is unmeasured.

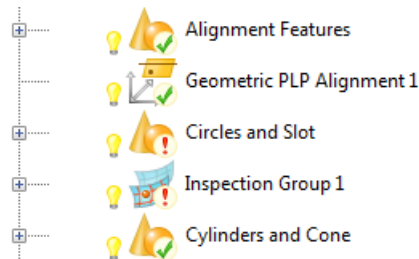



— the item is been measured and the results are within the tolerance band.



— the item is measured and some or all of the results are outside the tolerance band.

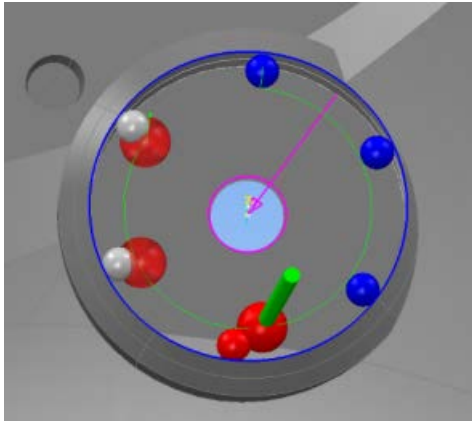
For example:



- 5 If you want to remeasure any item, select its entry in the inspection sequence and click the **Play item**  button to retake the points.

The Bouncing Ball

The Bouncing Ball provides an on-screen, point-by-point guide to probing features on the part. When you play an item that has a probe path, the points required to measure the feature are shown in the CAD view and the first point to be taken is shown in red. When you take the point, its colour changes to grey and the colour of the next point to be measured changes to red. This continues for each point until you have completed the measurement of the item. For example:



The locations of the probed points are shown as red transparent spheres for geometric items, and as confetti for surface inspection points.


Applying temperature compensation

Part compensation enables you to adjust measurements for the effects of thermal expansion on the part.



PowerINSPECT Play Only does not compensate for thermal expansion of the probe head and stylus. To compensate for these changes, calibrate your measuring device at the temperature at which the measurements will be made.

To apply thermal compensation to the current Measure:

- 1 In the **Main** toolbar, click . The **Part Compensation** dialog is displayed.



*The **Part Compensation** dialog is also displayed when you click **Play All** if all items in the inspection sequence are unmeasured; you are not using the temperature recorded by the measuring device; and you are using a standard or custom thermal expansion coefficient greater than zero.*

- 2 In the **Temperature** box, enter the temperature of the part in degrees Celsius and press the **Tab** key. PowerINSPECT Play Only calculates the measurement compensation factor from the **Thermal expansion coefficient** and displays it in the **Scaling factor** box.



*The **Thermal expansion coefficient** of the material must be specified during the creation of the inspection sequence.*




- 3 Click **OK** to save your changes and close the dialog.

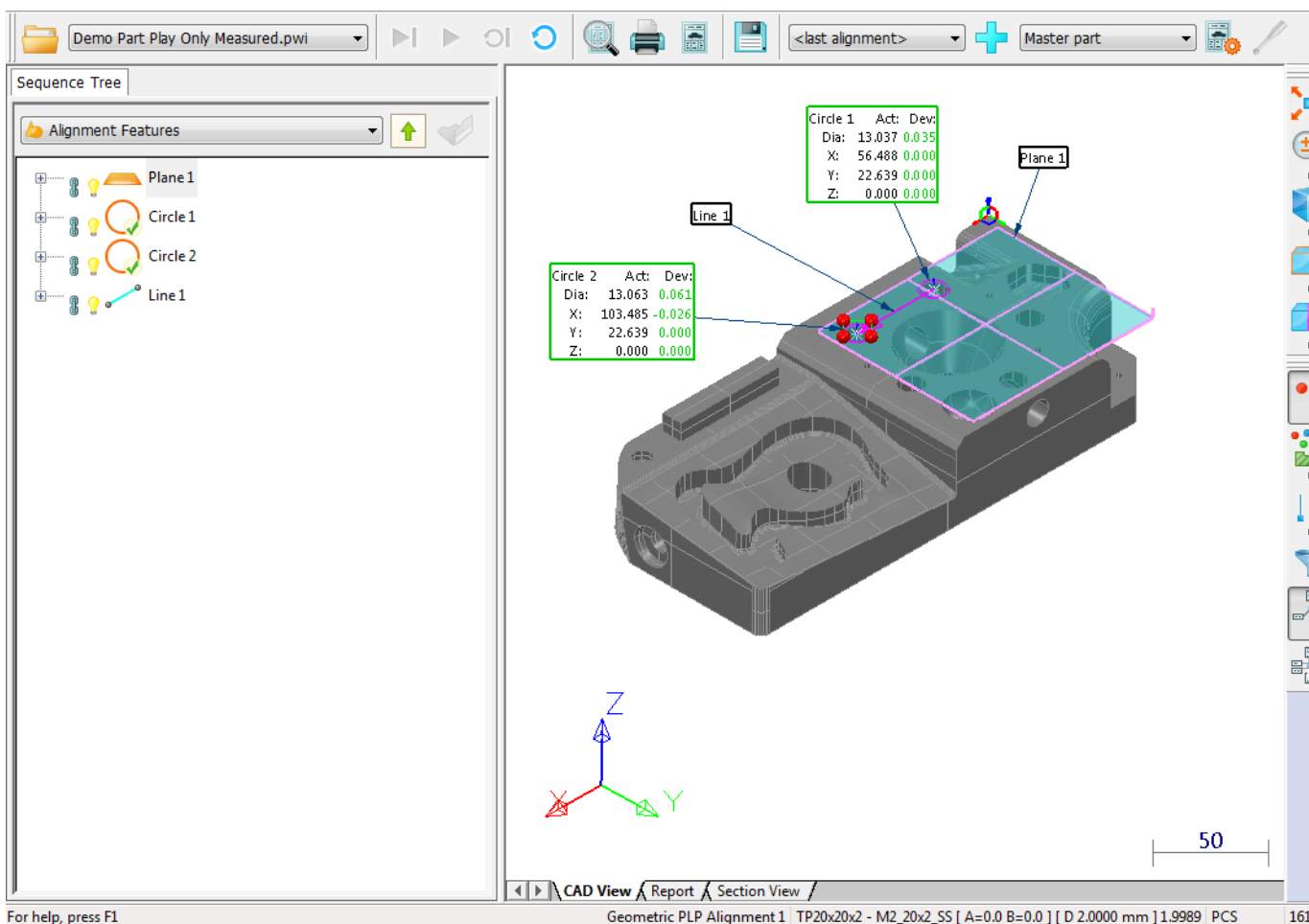



You must specify the part temperature for each Measure in the document.

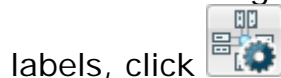
Viewing the measurement results

When you have measured a feature on the part, you can display the results as a label (see page 50) in the CAD view. For example, to display sample measurement results for the part:

- 1 Click  on the **Main** toolbar and open **Demo Part Play Only Measured.pwi**.
- 2 Select **Alignment Features** in the Sequence Tree, and click  to display the items it contains.
- 3 Click . The colour of the measurements and the box surrounding the label indicate whether the measurement is in or out of tolerance.

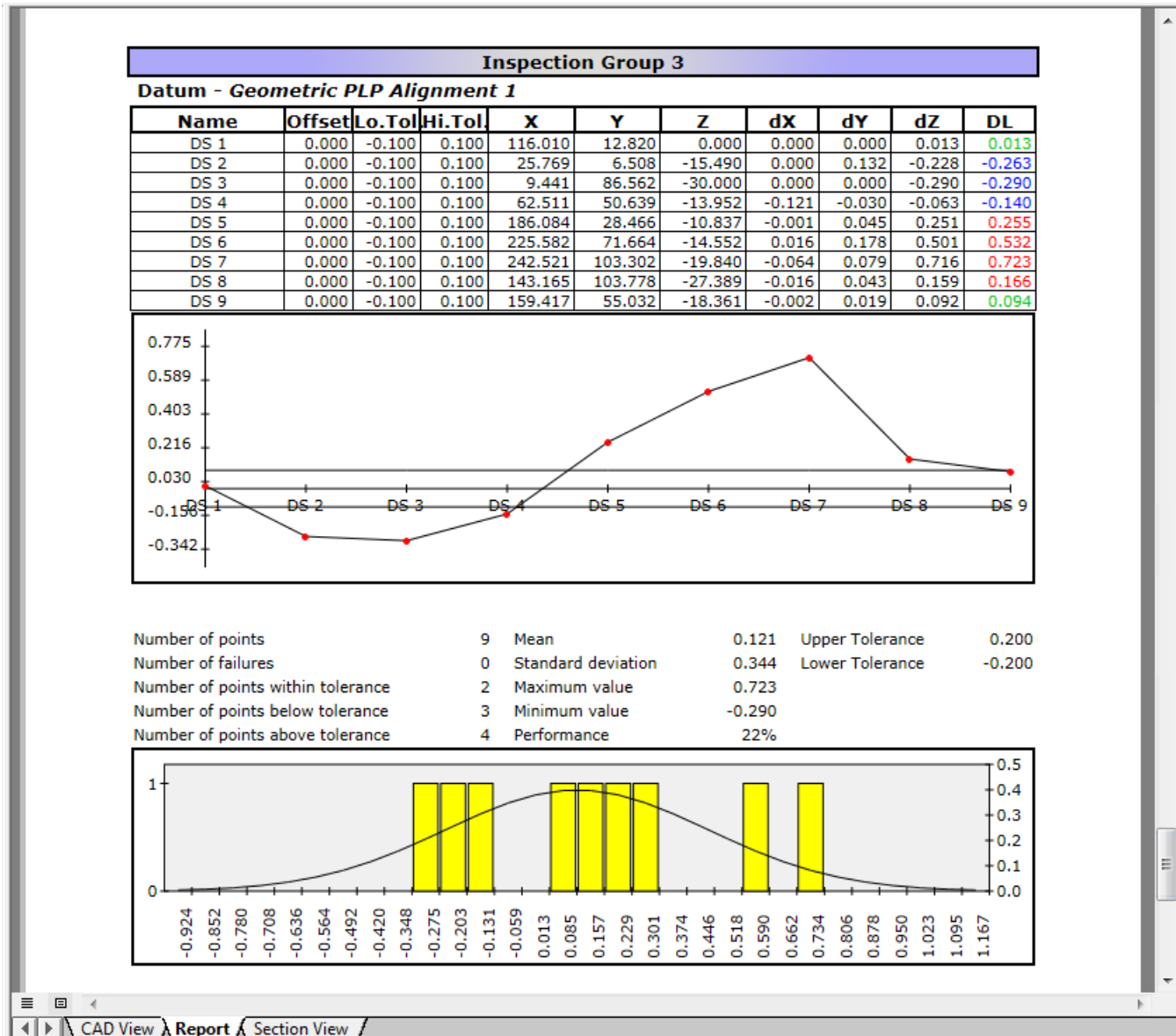


You can hide individual labels by clicking the light bulb  icon on the feature's entry in the Sequence Tree. To redisplay the result, click the icon again. To choose the information displayed in the



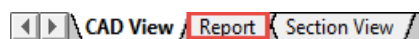
labels, click  to open the **Session Label Settings** dialog (see page 56).

PowerINSPECT Play Only also includes a customizable reporting function that provides a detailed breakdown of your measurements.



To display the measurement results in a report:

- 1 Click the **Report** tab at the bottom of the PowerINSPECT Play Only window:




The measurement results are displayed. Use the scroll bar on the right of the report to display all the details:

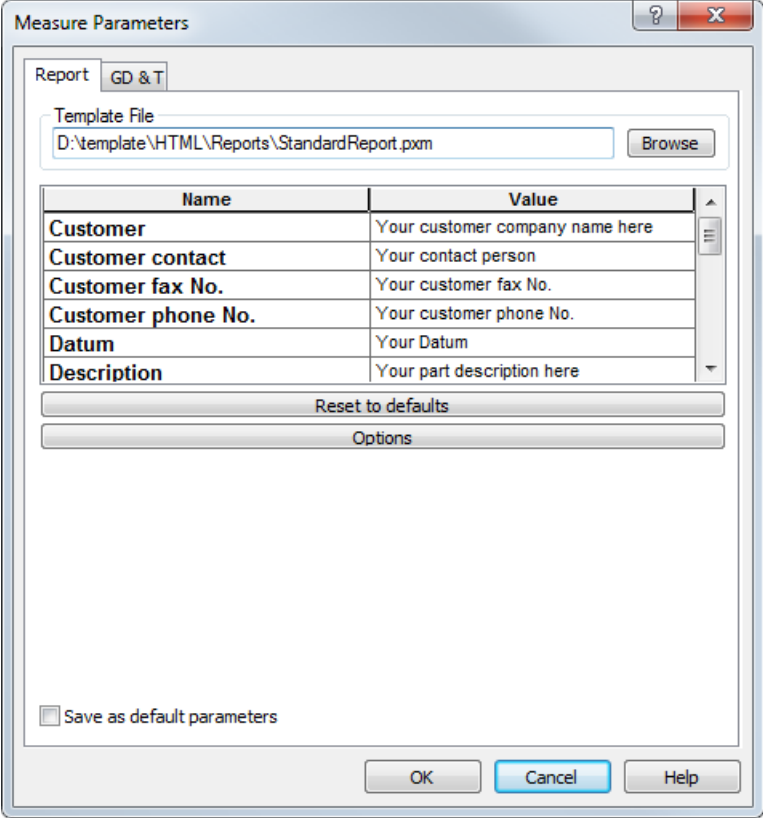
- The name of each geometric feature is followed by a string in parentheses indicating the reference items to which the results are relative. For example:

Angle 1 (References: <i>RPS Alignment 1::Plane X (Y02), RPS Alignment 1::Z Axis, BestFit 3::Z Axis</i>)						
Angle	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	0.100	-0.100	0.000	xxxxx	xxxxx	xxxxx

Angle 2 (References: <i>RPS Alignment 1::Plane Y (X02), Machine Datum::Plane X (Y02)</i>)						
Angle	Hi-Tol	Lo-Tol	Nominal	Measured	Deviation	Error
	0.100	-0.100	0.000	125.000	125.000	124.900

- Unmeasured surface inspection groups are indicated by the phrase *No points have been taken*.
- Unmeasured values are displayed as **xxxxxx**.

- 2 Click the **Set report template**  button in the **Main** toolbar (see page 15). The **Measure Parameters** dialog is displayed. It enables you to specify information to be included in the report:



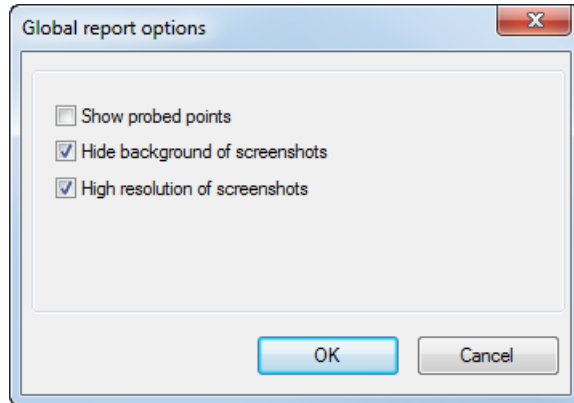
The **Measure Parameters** dialog box is shown. It has a title bar with a question mark and a close button. Inside, there's a 'Report' tab with a 'GD & T' sub-tab. Below this is a 'Template File' section with a text box containing 'D:\template\HTML\Reports\StandardReport.pxm' and a 'Browse' button. A table with two columns, 'Name' and 'Value', contains the following entries:

Name	Value
Customer	Your customer company name here
Customer contact	Your contact person
Customer fax No.	Your customer fax No.
Customer phone No.	Your customer phone No.
Datum	Your Datum
Description	Your part description here

Below the table are two buttons: 'Reset to defaults' and 'Options'. At the bottom left is a checkbox labeled 'Save as default parameters'. At the bottom right are three buttons: 'OK', 'Cancel', and 'Help'.

- 3 Click the **Browse** button, select **HTML** in the **Open File** dialog, followed by the folder in which the template is located, and then select the .pxm file for the required template. Click the **Open** button.
- 4 Click **Reset to defaults** to refresh the report screen and extract the variables into the report header.
- 5 Change the text in the **Value** column as required.

- 6 Select the **Save as default parameters** check box to set the currently selected template as the default.
- 7 If you want to display probed points in the CAD view, click **Global report options**, select the **Show probed points** check box in the **Global Report Options** dialog, and then click **OK**.




- 8 Click **OK** to close the **Measure Parameters** dialog and update the report.
- 9 Display the report again, and check it contains the changes you made.

Printing the report

To print the results displayed in the **Report** tab:

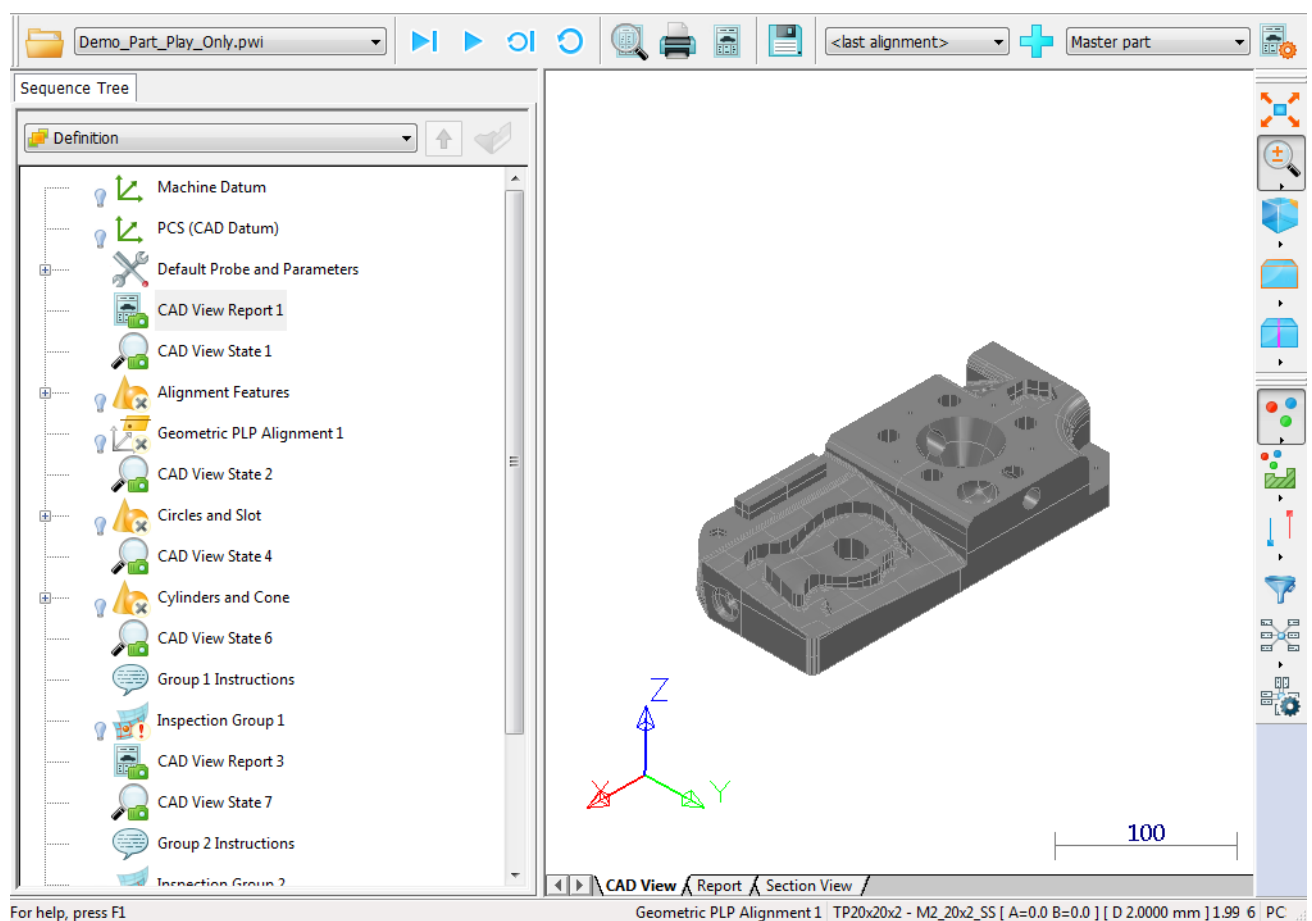


- 1 Click  to open the **Print Manager** dialog.
- 2 Select the required printer and click the **Print** button.

The results are output to the selected printer using the currently selected report template.


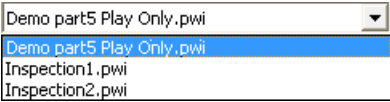








Understanding the screen layout

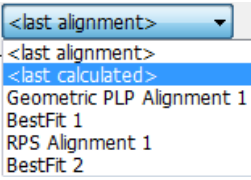



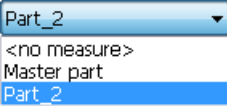



This section describes the **Main** toolbar across the top of the PowerINSPECT Play Only window, and the **CAD View** and **View Options** toolbars displayed on the right.





Using the Main toolbar

The **Main** toolbar enables you load and work with inspections:






Option	Description
	<p>Opens an inspection document (see page 2). The name of the selected file is displayed in the Previous .pwi files list:</p>  <p>This provides quick access to previously opened inspections.</p>
	Measures the currently selected item in the inspection sequence.
	Measures all items in the inspection sequence.
	Clears all measurements for the selected item. The measurements of any dependent items are also cleared.
	Clears all measurements for the part. If a group is open, only the items in the group are reset.
	Displays a preview of how the CAD model or report will look when printed.
	Prints the CAD model or report on the selected printer.
	Saves the report to an Excel or HTML file.
	Saves your measurements.

Option	Description
	<p>Contains the alignments used in the inspection. When you select an entry in this list, PowerINSPECT Play Only recalculates the measurements of all items that have a Coordinate system setting of <i><Active Alignment></i> with respect to the new alignment. You can select:</p> <ul style="list-style-type: none"> ▪ A named alignment. The icon of the alignment item is displayed with gold axes  to indicate it is the active alignment. ▪ <i><last alignment></i>. This uses the alignment nearest the bottom of the inspection sequence as the active alignment. The icon of the alignment item is displayed with silver axes  to indicate it is the active alignment. ▪ <i><last calculated></i>. This uses the last measured alignment as the active alignment. If no alignments are measured, the last alignment in the inspection sequence is used. The icon of the last calculated alignment item is displayed with silver axes to indicate it is the active alignment.
	<p>Adds a Measure to the current .pwi file. The new measure is a copy of the Active measure selected in the list at the right of the button:</p>  <p>Measures enable you to inspect a series of parts manufactured from the same model. Each Measure records the measurements for one example of the part.</p>
	<p>Selects the report template (see page 10).</p>
	<p>Selects and calibrates the probes for the inspection.</p>
	<p>Displays this online help.</p>

Option	Description
	Specifies the temperature when you need to apply temperature compensation to your measurements.
	Closes the current inspection without closing PowerINSPECT Play Only.


Using the CAD View toolbar

Use the **CAD View** toolbar to control the way in which models are displayed in the CAD view.

Button	Description
	Resize to Fit (see page 18)
	Mouse View Mode (see page 19)
	Select View Mode (see page 20)
	Shading Mode (see page 25)
	Section Shading Mode (see page 29)



You can move the toolbar by picking it up using the double lines at the top of the toolbar.

Click  below a button to display the options, then click the option you want to use. For example:



The pull-out toolbar closes and the option is shown on the **CAD View** toolbar.

Resize to fit



Click  to resize the model so that it just fits in the CAD view.




Ctrl+A is the keyboard shortcut for this button.

Mouse button 1 view mode

Use the options on the **Mouse Button 1 View Mode** pull-out toolbar to manipulate the CAD view using only the left mouse button.

To choose a view mode for the mouse button:

- 1 Click  below the **View Mode** button on the **CAD View** toolbar to display a pull-out toolbar of view manipulation options.



- 2 Choose the mode you want to use. Select:



to zoom in, or out of, the CAD view.



to zoom in on an area of the CAD view.












to pan the CAD view.



to rotate the model around the centre of the CAD view.


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

To...	Use...
Zoom in and out of the CAD view	 and 
Zoom in on an area by drawing a box around it	 or  or 
Pan the CAD view	 or 
Rotate the CAD view in any direction — a transformation globe appears to show the rotation direction	 or 

Select view

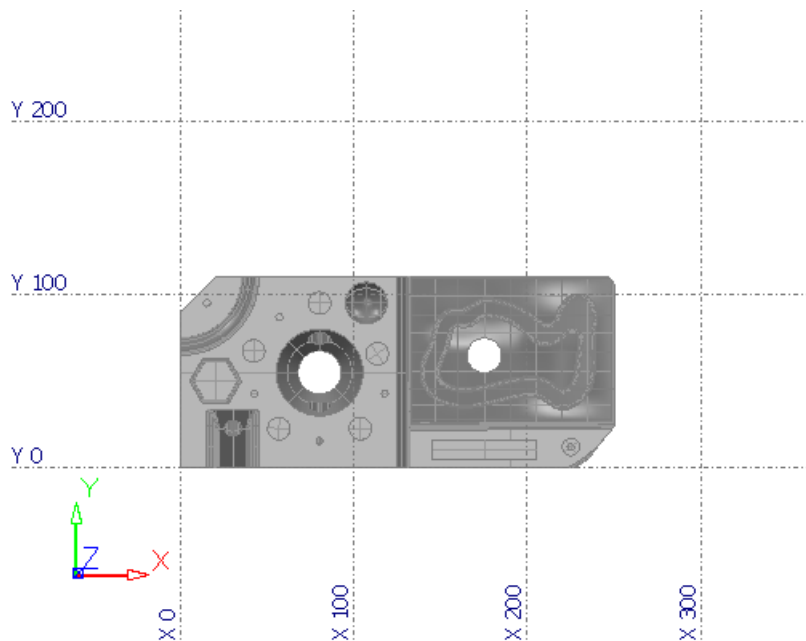
The **Select view** toolbar allows you to manipulate the view of the CAD model displayed in the CAD view.

To choose a view mode:


- 1 Click  below the current view mode shown on the **CAD View** toolbar to display the **Select view mode** pull-out toolbar.
- 2 Select the view mode you want to use.

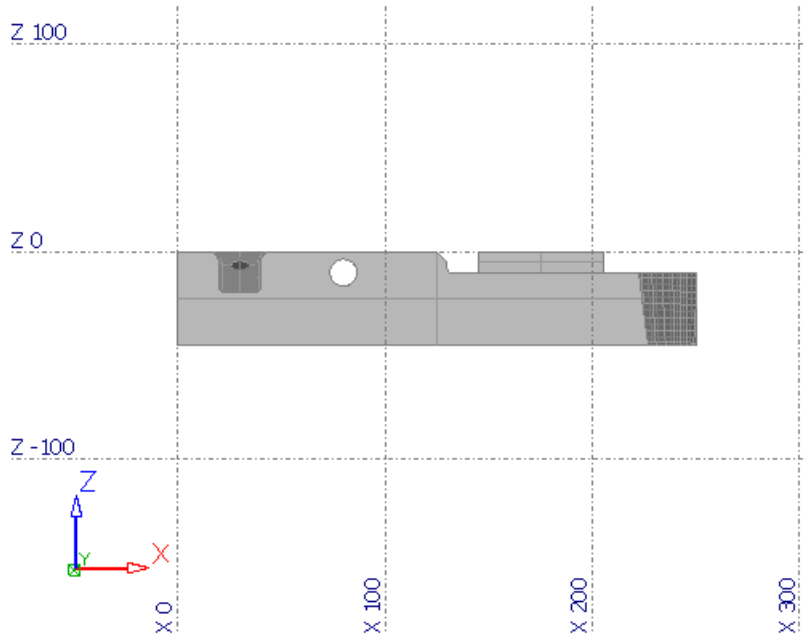
View From Top (Z)

Click the **View From Top (Z)**  button to display the CAD model from the top (X, Y), looking down the Z axis. For example:



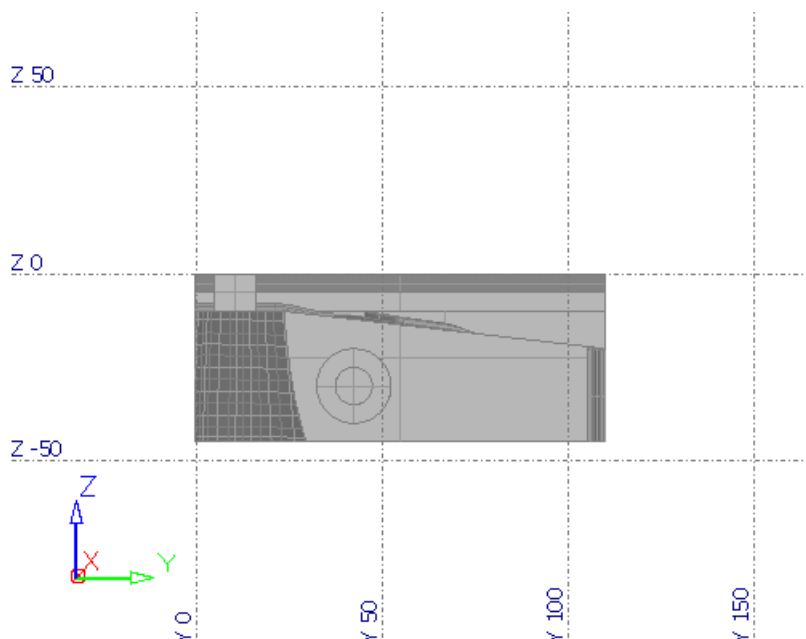
View From Front (-Y)

Click the **View From Front (-Y)**  button to display the CAD model from the front (X, Z), looking down the -Y axis. For example:




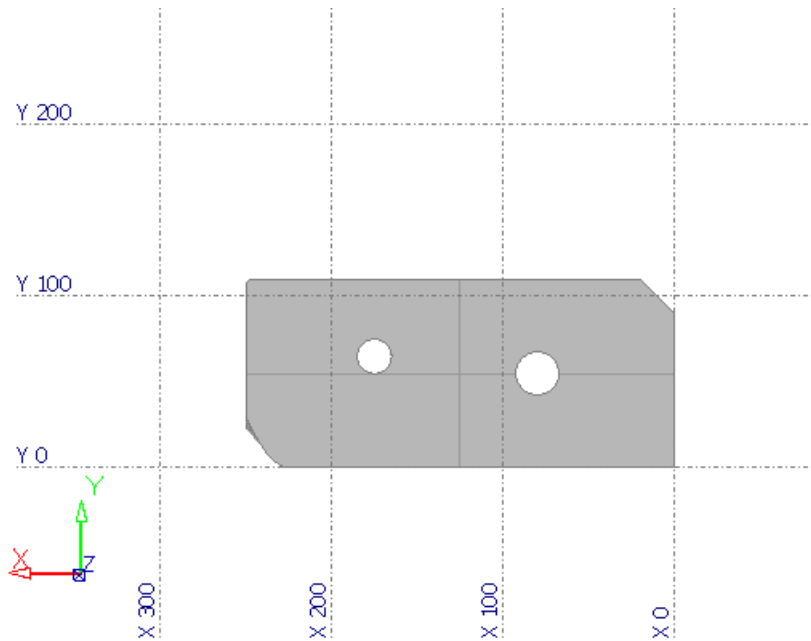
View From Right (X)

Click the **View From Right (X)**  button to display the CAD model from the right (Y, Z), looking along the X axis. For example:



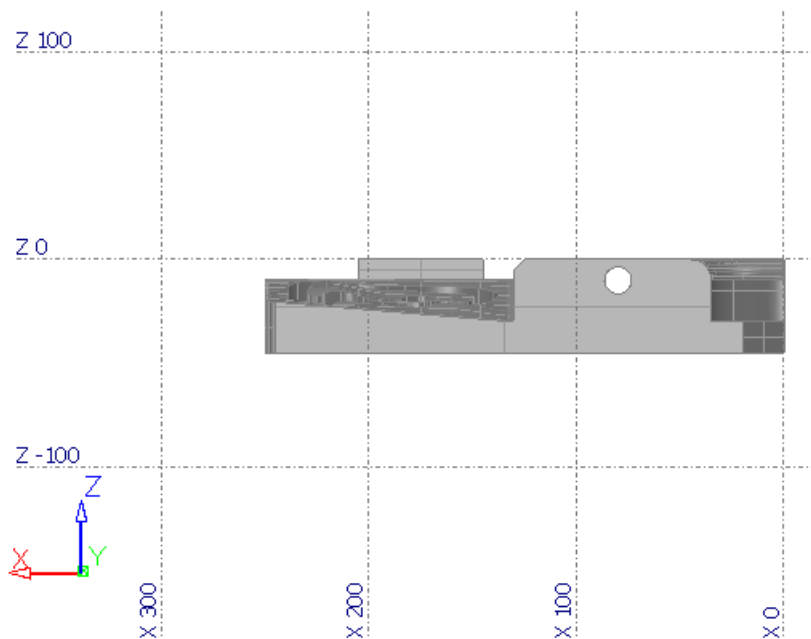
View From Bottom (-Z)

Click the **View From Bottom (-Z)**  button to display the CAD model from the bottom (X, -Y), looking down the -Z axis. For example:




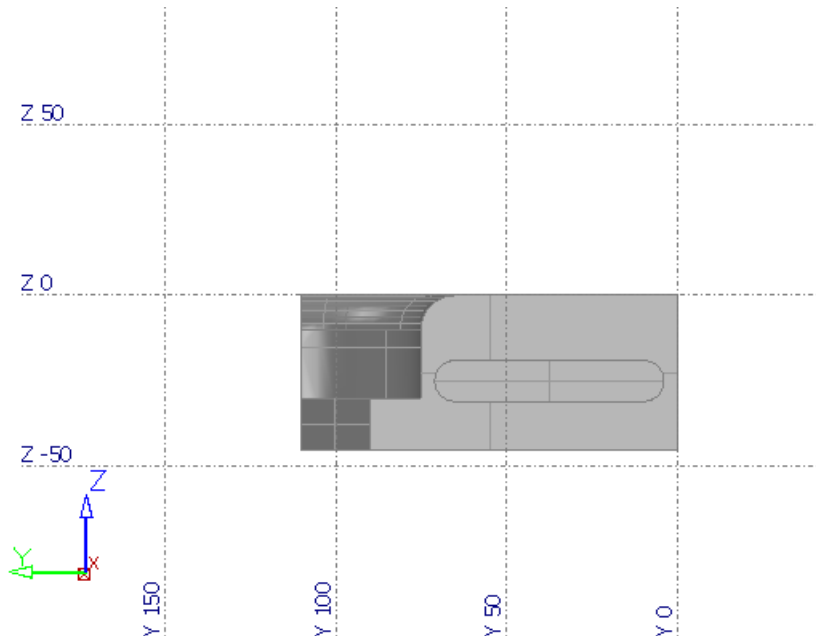
View From Back (Y)

Click the **View From Back (Y)**  button to display the CAD model from the back (-X, Z), looking down the Y axis. For example:




View From Left (-X)


Click the **View From Left (-X)**  button to display the CAD model from the left (-Y, Z), looking down the -X axis. For example:




ISO 1

Click the **ISO 1**  button to display the Isometric 1 view of the CAD model.


ISO 2

Click the **ISO 2**  button to display the Isometric 2 view of the CAD model.


ISO 3

Click the **ISO 3**  button to display the Isometric 3 view of the CAD model.

ISO 4

Click the **ISO 4**  button to display the Isometric 4 view of the CAD model.


Axonomic

Click the **Axonometric**  button to display the axonometric view of the CAD model.

Shading mode for model

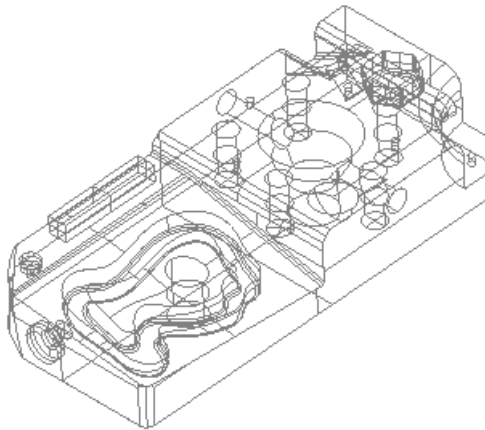
The **Shading Mode For Model** toolbar allows you to change the appearance of the model displayed in the CAD view.

To choose a shading mode:

- 1 Click  below the current shading mode shown on the **CAD View** toolbar to display the **Shading Mode For Model** toolbar.
- 2 Click the shading mode that you want to use.

Show Wireframe

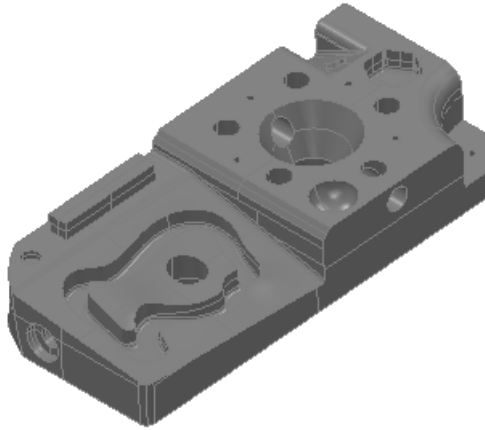
Click the **Show Wireframe**  button to display a wireframe representation of the model in the CAD view. For example:



*To change the density of the wireframe lines, use the **Change Meshing** menu option.*


Show Shaded and Wireframe

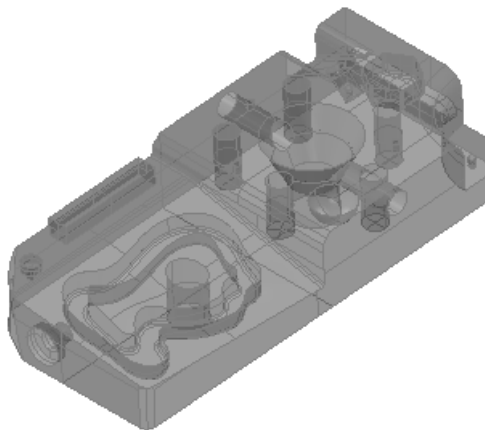
Click the **Show Shaded and Wireframe**  button to display a shaded wireframe representation of the model in the CAD view. For example:



*To change the density of the wireframe lines, use the **Change Meshing** menu option.*


Show Transparent Wireframe

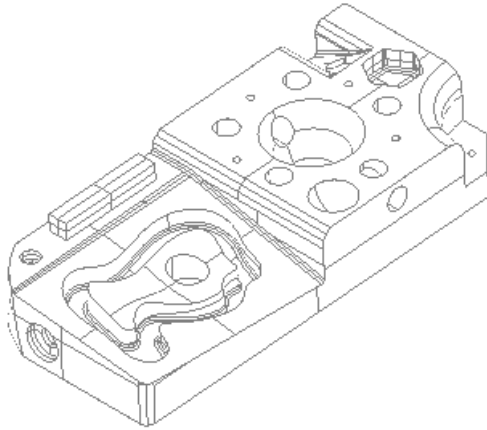
Click the **Show Transparent Wireframe**  button to display a semi-transparent wireframe representation of the model in the CAD view. For example:




*To change the density of the wireframe lines, use the **Change Meshing** menu option.*

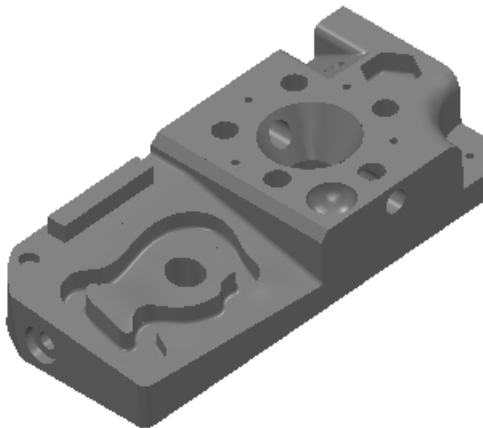
Hidden Line Removal

Click the **Hidden Line Removal**  button to display a wireframe representation of the model in the CAD view, with hidden lines removed. For example:



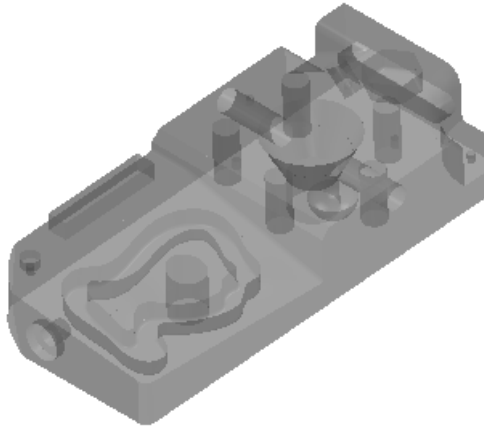
Shaded View

Click the **Shaded View**  button to display a shaded representation of the model in the CAD view, without any wireframe. For example:



Show Transparent




Click the **Show Transparent**  button to display a shaded, semi-transparent representation of the model in the CAD view, without any wireframe. For example:



Section shading mode

The **Section Shading Mode** toolbar allows you to change the appearance of the model displayed in the CAD view when the model contains a section.

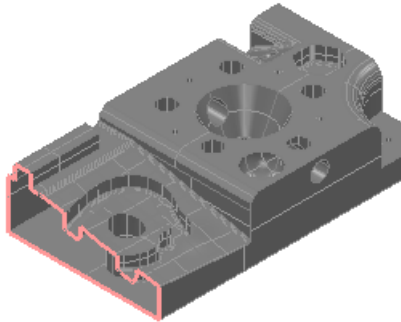
To choose the section shading mode:

- 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 Click  below the current section shading mode shown on the **CAD View** toolbar to display the **Section Shading Mode** toolbar
- 4 Click the shading mode that you want to use.

Hide model



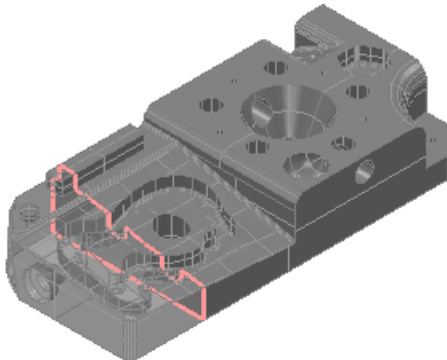
Click the **Hide Model** button to hide the part of the model that appears nearest. For example:



Translucent model



Click the **Translucent Model** button to display a shaded, semi-transparent representation of the part of a model that appears nearest. For example:









Section shading disabled

Click the **Section Shading Disabled**  button to disable the mode.

Using the View Options toolbar

Use the **View Options** toolbar to control the display of information in the **CAD View** tab.

Button	Description
	Display Mode (see page 32)
	Section Measurement Display (see page 34)
	Geometric Feature Display (see page 36)
	Filter Display (see page 39)
	Show/Hide Labels (see page 50)
	Show global label settings (see page 56)

Click  below a button to display the options.




You can move the toolbar by picking it up using the double lines at the top of the toolbar.

Inspection point display

Select the **Inspection point display** button on the **View Options** toolbar to display probed inspection points in the CAD view.

To select the way in which the points are displayed:

- 1 Click  below the **Inspection point display** button to display the toolbar.
- 2 Choose the display mode you want to use. Select:



to display the points as **In-place** labels (see page 32).



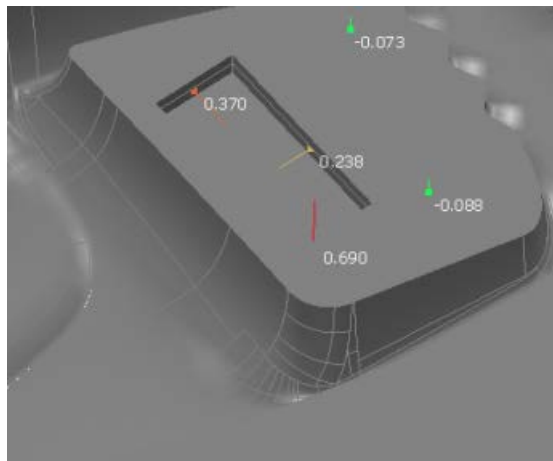
to display the points as **Confetti** (see page 33).

To hide the inspection points, deselect the button.

In-place labels



The **In Place** button displays the measured points of the inspection group as coloured pins on the CAD model. For example:



PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

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

The direction of the pin shows the side from which the point was probed; the length of the pin indicates the size of the deviation.

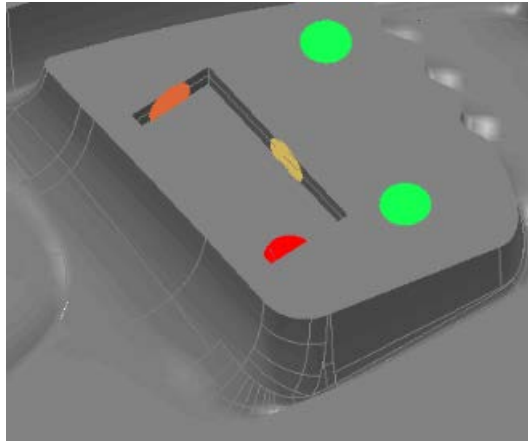


Select the **Display deviation text** check box in the **Filter Display** dialog (see page 39) to show more details for each measurement.

Confetti



The **Confetti** button displays the measured points of the inspection group as coloured discs in the CAD view. For example:



PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

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




Select the **Display deviation text** check box in the **Filter Display** dialog (see page 39) to show more details for each measurement.

Section measurement display

Use the **Section Measurement Display** toolbar to show the measured points of section groups in the CAD view.

To select the way in which the points are displayed:

- 1 Click  below the **Section Measurement Display** button to display the toolbar.
- 2 Choose the display mode you want to use. Select:



to display the deviation of each point as **Spot Confetti** (see page 34).



to display the deviation of each point as **Deviation Lines** (see page 35).



to display the deviation of each point as a **Connected Deviation Lines** (see page 35).



to display the deviation of each point as a **Vector Lines** (see page 36).

To hide the measured points, deselect the button.

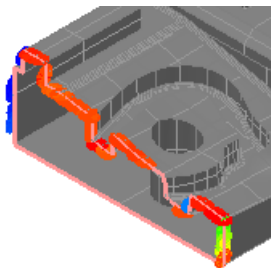


Use the **Tab display** settings in the **Section** dialog to control the sizing of the deviation lines.

Spot confetti



The **Spot Confetti** button displays the measured points in section groups as coloured discs on the CAD model. For example:



PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

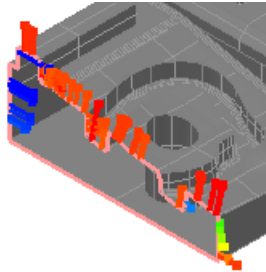
- Green points are within tolerance.

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

Deviation lines



The **Deviation Lines** button displays the deviation of measured points in section groups as pins. The length of a pin indicates the size of the pin's deviation from nominal. For example:



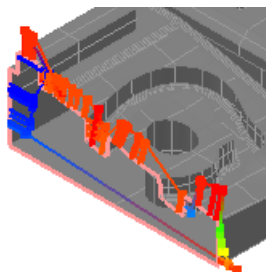
PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

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

Connected deviation lines



The **Connected Deviation Lines** button displays the measured points in section groups as linked pins. The length of a pin indicates the size of the point's deviation from nominal, and the links indicate the shape of the probed section. For example:



PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

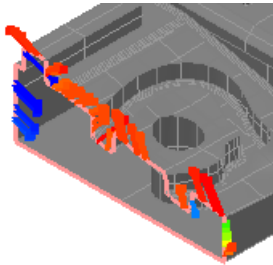
- Green points are within tolerance.
- Red points are above the tolerance band.

- Blue points are below the tolerance band.

Vector lines



The **Vector Lines** button displays the deviation of measured points in section groups as vectors. The direction of the vector indicates the direction of deviation. For example:




PowerINSPECT Play Only compares the measurement results to the CAD data and uses colour to indicate the deviation from the CAD data:

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

Geometric feature display

Use the **Geometric Feature Display** toolbar to show the measured points of geometric items in the CAD view.

To display the measured probed points:

- 1 Click  below the **Geometric Feature Display** button on the **View Options** toolbar to display the toolbar.

- 2 Choose the display mode you want to use. Select:



to display Deviation lines (see page 37).



to display Connected deviation lines (see page 37).



to display Vector lines (see page 38).



to display Discs (see page 38).

- 3 Use the **Geometric Inspection** tab in the **Filter Display** dialog (see page 39) to choose the points you want to display.

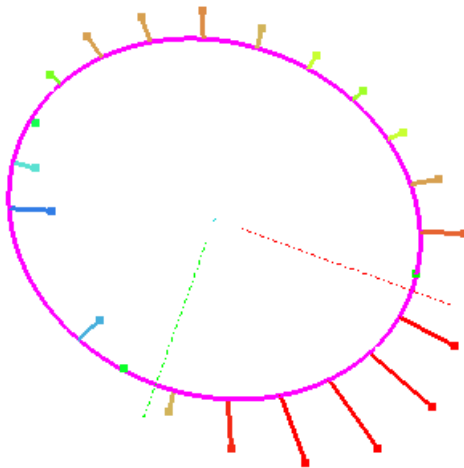


The colours used to display the points are controlled by the **Tolerance band** settings in the **Colours** page of the **Options** dialog.

Deviation lines



Use the **Deviation lines** button to display the probed points of geometric features as coloured pins on the CAD model. For example:

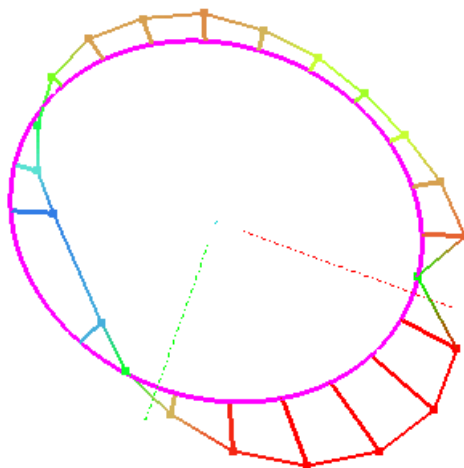


The colour of each pin indicates the deviation of the probed point from the form nominal. The position of each pin relative to the feature indicates the direction of the point's deviation.

Connected deviation lines




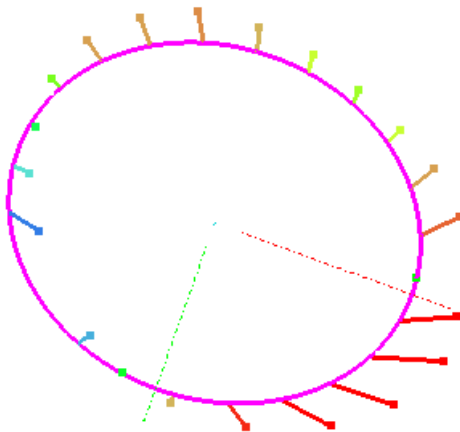
Use the **Connected deviation lines** button to displays the probed points of geometric features as connected, coloured pins on the CAD model. For example:



The colour of each pin indicates the deviation of each point from the form nominal. The position of each pin relative to the feature indicates the direction of the point's deviation. The heads of the pins are connected so you can see how well the probed points match the CAD model.


Vector lines

Use the **Vector lines**  button to display the probed points of geometric features as coloured pins on the CAD model. For example:



The colour of each pin indicates the deviation of each point from the form nominal. The orientation of each pin indicates the direction from which the point was probed.

Discs

Use the **Discs**  button to display the probed points of geometric features as coloured discs on the CAD model. For example:



The colour of each disc indicates the deviation of the point from the form nominal.



To change the size of the discs, enter a new value in the **Diameter of geometric discs** box in the **Size** page of the **Options** dialog.

Filter display

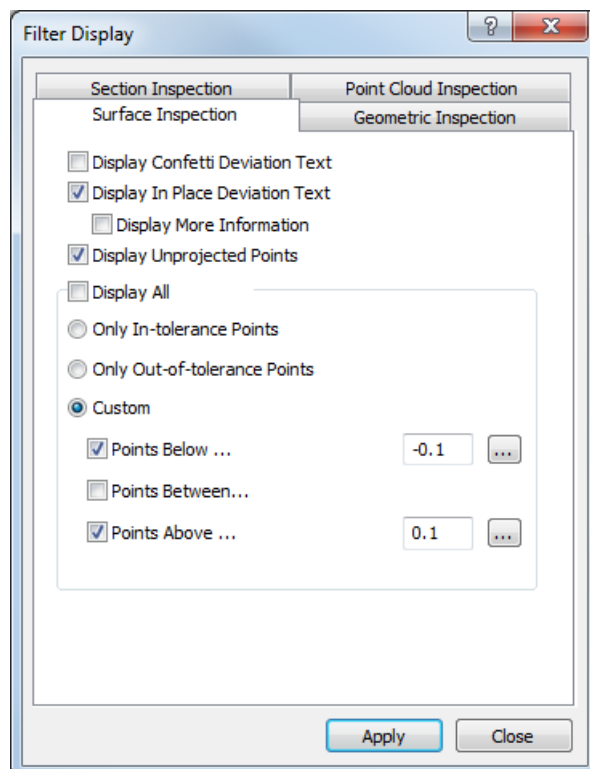


Click the **Filter Display** button to display the **Filter Display** dialog and specify how measured points are displayed in the CAD view. Select the:

- **Surface Inspection** tab (see page 39) to specify how inspection points are displayed.
- **Geometric Inspection** tab (see page 43) to specify how probed points are displayed.
- **Section Inspection** tab (see page 45) to specify how measured points in section groups are displayed.
- **Point Cloud Inspection** tab (see page 47) to specify how point cloud points are displayed.

Surface inspection settings

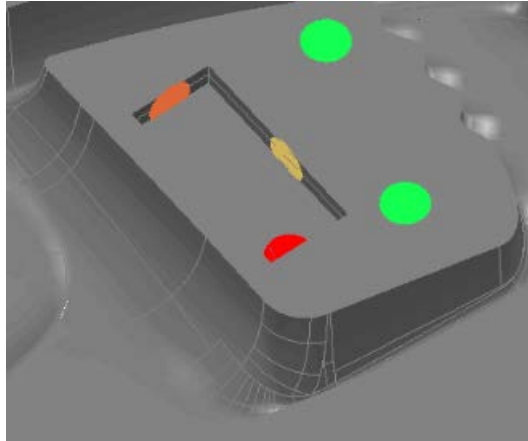
Use the **Surface Inspection** tab to control the display of probed inspection points when you select an **Inspection Point Display** option (see page 32).



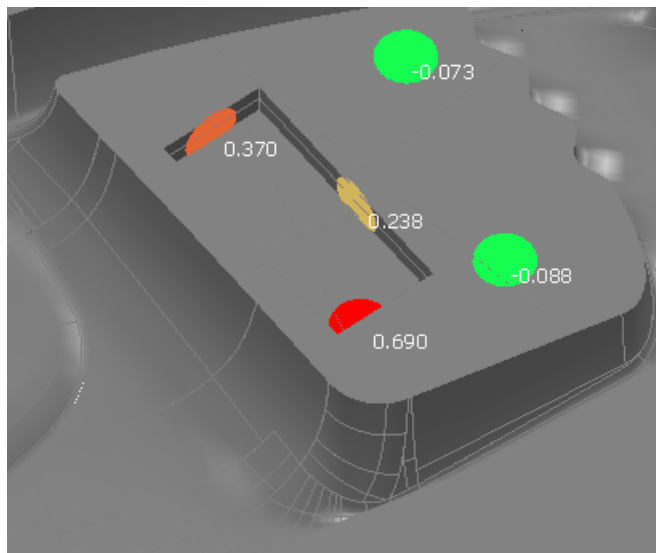
The tab contains the following settings:

- **Display confetti deviation text** — Select this check box to show the deviation of inspection points alongside confetti (see page 32) in the CAD view. Deselect this check box to show confetti without deviations.

Example of confetti when the **Display confetti deviation text** check box is not selected:

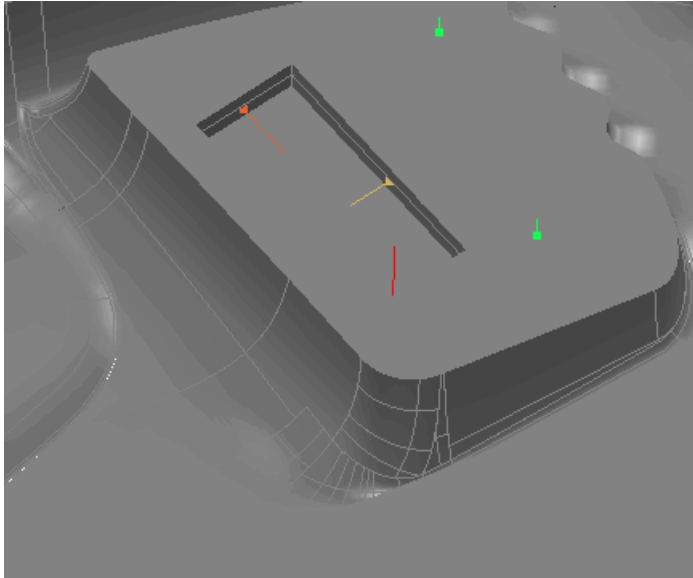


Example of confetti when the **Display confetti deviation text** check box is selected:

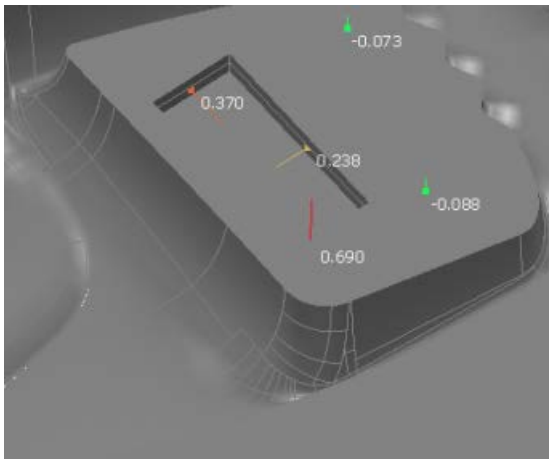


- **Display in-place deviation text** — Select this check box to show the deviation of inspection points alongside **In-place labels** (see page 32) in the CAD view. Deselect this check box to show in-place labels without deviations.

Example of in-place labels when the **Display in-place deviation text** check box is not selected:

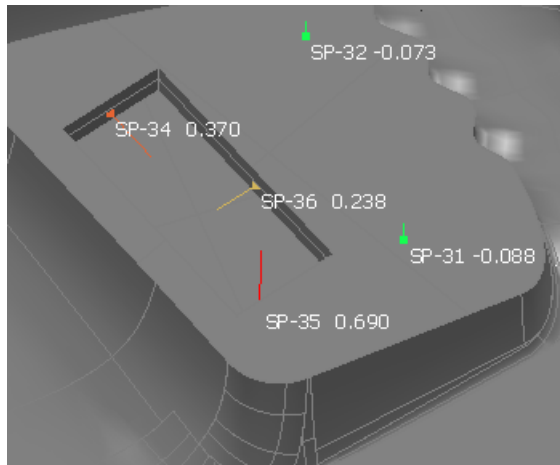


Example of in-place labels when the **Display in-place deviation text** check box is selected:




- **Display more information** — Select this check box to show the name of each probed point when **In-place labels** is selected in the **Inspection point display** toolbar. You must select the **Display in-place deviation text** check box to enable this option.

Example of in-place labels when the **Display in-place deviation text** and **Display more information** check boxes are selected:



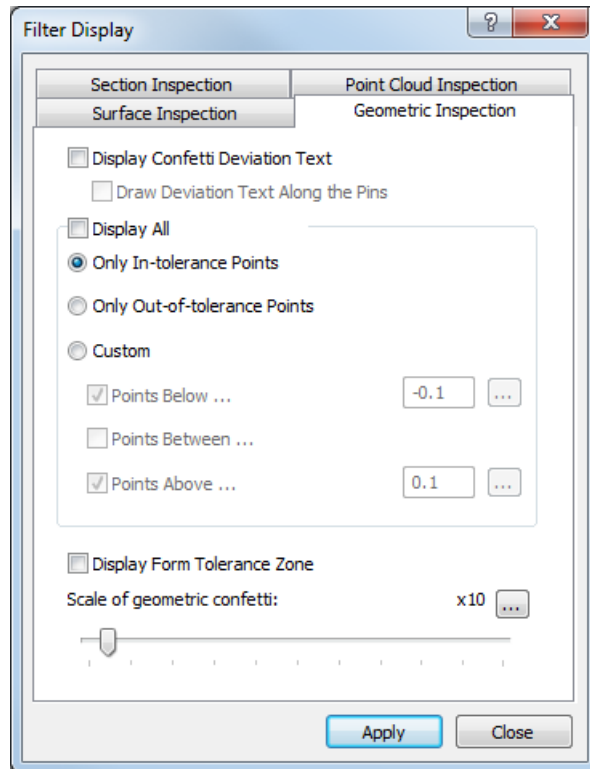
- **Display unprojected points** — Select this check box to display unprojected points.
- **Display all** — Select this check box to display all surface inspection points. To limit which points are displayed, deselect the check box and then select:
 - **Only in-tolerance points** to display points that are within tolerance.
 - **Only out-of-tolerance points** to display points that are out-of-tolerance.
 - **Custom** to specify the lower and upper 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 the **Point between** check box to display points between the specified upper and lower tolerance limits.

To base a display limit on the tolerances of an inspection group, select the group in the inspection sequence, then click .

Click **Apply** to display your changes in the CAD view.

Geometric inspection settings

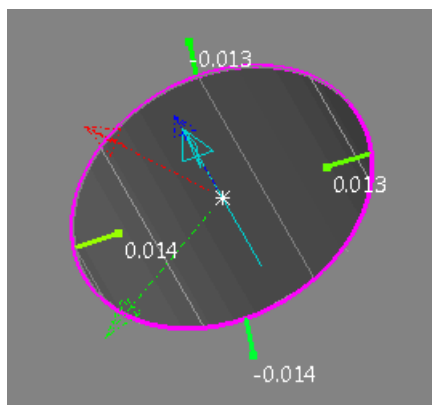
Use the **Geometric Inspection** tab to control the display of probed item points when you select a **Geometric Feature Display** option (see page 36).



The tab contains the following settings:

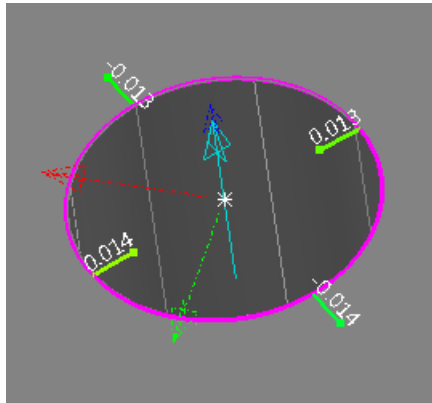
- **Display confetti deviation text** — Select this check box to show the calculated deviation of each probed point alongside its disc or deviation line.

For example:




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

For example:

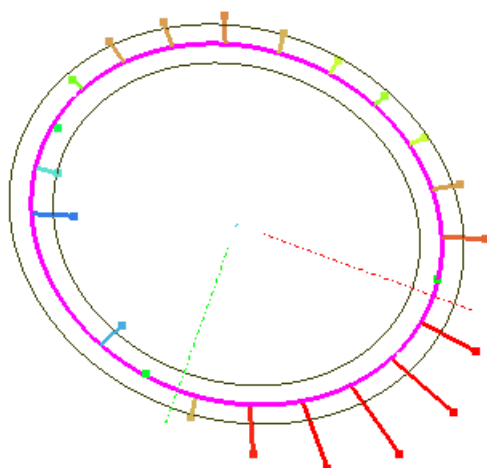



- **Display all** — Select this check box to display all probed points of each geometric feature regardless of the feature's form tolerance. Deselect this check box to choose which points are displayed. Select:
 - **Only in-tolerance points** to display only those points that are within form tolerance.
 - **Only out-of-tolerance points** to display only those points that are out of form tolerance.
 - **Custom** to specify the lower and upper 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 the **Point between** check box to display points between the specified upper and lower tolerance limits.

To base a display limit on the form tolerance of an item, select the item in the inspection sequence, then click . The tolerance is halved and displayed in the adjacent box.

- **Display Form Tolerance zone** — Select this check box to display the limits of each feature's form tolerance in the CAD view.

For example:

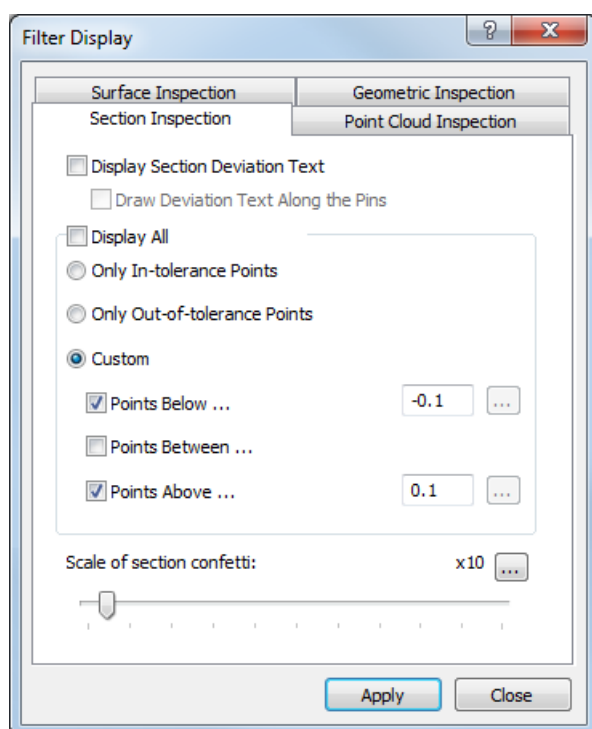


- **Scale of geometric confetti** — Move the slider to change the magnification at which the point indicators and form tolerance zone are displayed. To change the magnification limit, click  and enter a new maximum value.

Click **Apply** to save your changes and display them in the CAD view.

Section inspection settings

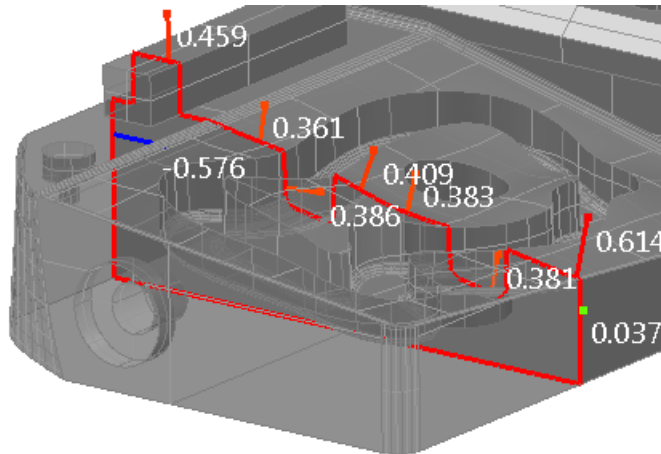
Use the **Section Inspection** tab to control the display of probed points in a measured section group when you select a **Section Measurement Display** (see page 34) option.



The tab contains the following settings:

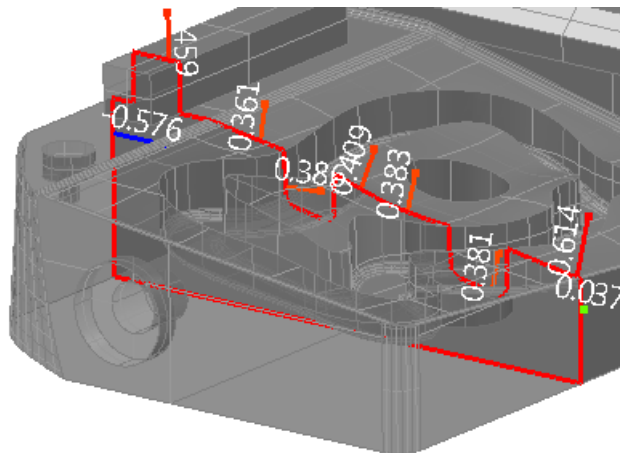
- **Display section deviation text** — Select this check box to show the calculated deviation of each probed point alongside its disc or deviation line.

For example:




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

For example:



- **Display all** — Select this check box to display all probed section group points. To limit which points are displayed, deselect the check box, and then select:
 - **Only in-tolerance points** to display only those points that are within form tolerance.
 - **Only out-of-tolerance points** to display only those points that are out of form tolerance.

- **Custom** to specify the lower and upper 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 the **Point between** check box to display points between the specified upper and lower tolerance limits.
- **Scale of section confetti** — Move the slider to change the magnification at which the deviation indicators are displayed. To change the magnification limit, click  and enter a new maximum value.

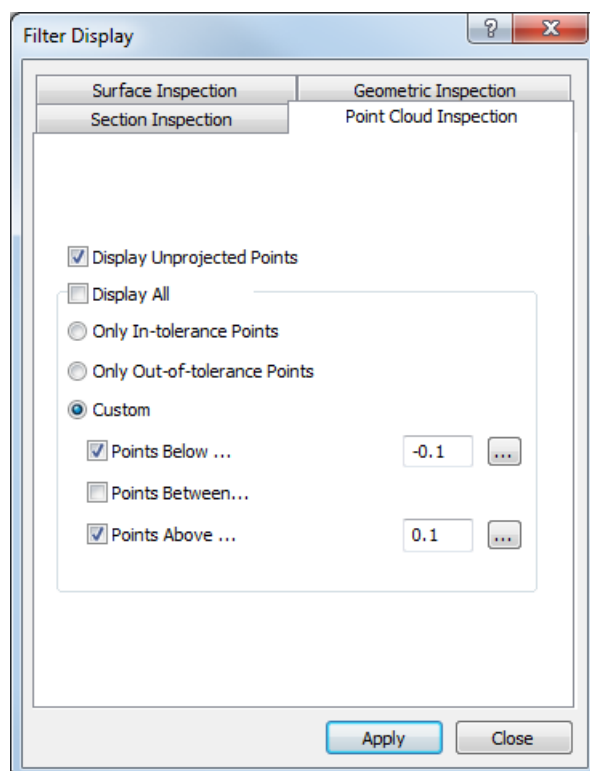


*This has no effect if the deviation is displayed as **Spot Confetti**.*

Click **Apply** to save your changes and display them in the CAD view and the Section view.

Point cloud inspection settings

Use the **Point Cloud Inspection** tab to control the display of point cloud points.



The tab contains the following settings:

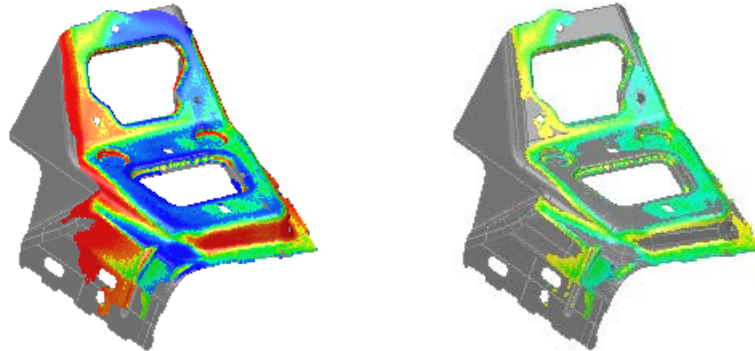
- **Display unprojected points** — Select this check box to display unprojected points.

- **Display all** — Select this check box to display all point cloud points. To limit which points are displayed, deselect the check box and then select:
 - **Only in-tolerance points** to display points that are within tolerance.

Comparison of a point cloud displaying all projected points and then filtered to display points within tolerance:

Display all

Only in-tolerance points

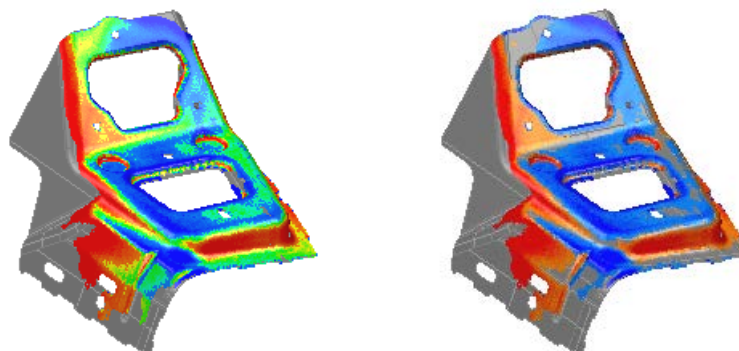


- **Only out-of-tolerance points** to display points that are out-of-tolerance.

Comparison of a point cloud displaying all projected points and then filtered to display points out of tolerance:

Display all

Only out-of-tolerance points



- **Custom** to specify the lower and upper 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 the **Point between** check box to display points between the specified upper and lower tolerance limits.

Click **Apply** to display your changes in the CAD view.




*These settings apply to all Point Cloud items except those you have filtered using the **Filter Point Cloud** dialog.*

Show/Hide labels

Select the **Show/Hide Labels** toolbar to show and hide details of features in the CAD view using labels.

To select the way in which the labels are displayed:

- 1 Click  below the **Show/Hide Labels** button to display the pull-out toolbar.
- 2 Select the type of label you want to display. Click:



to display all labels horizontally (see page 53).



to display labels horizontally and vertically (see page 54).



to position and size labels manually (see page 55).

The colours of each label (see page 50) are determined by the measurement status of the item.

Working with labels

- To choose the information displayed in each label, right-click the label and select **Label settings** (see page 56).
- To hide individual measurements in a label, right-click the label, and select **Edit Item**. In the item dialog, deselect the **Output in report** check boxes of the measurements you do not want to display.
- To control the appearance of CAD view labels and specify the measurements to be displayed by default, right-click a label and select **Settings** (see page 56) from the context menu.
- To edit the nominal values of the item, right-click the label and select **Edit item**. In the item dialog, edit the nominal values.

By default, PowerINSPECT Play Only hides the labels of features that are not currently visible in the CAD view. To show the labels for all features, select the **Show hidden labels** check box in the **Labels** page of the **Options** dialog.

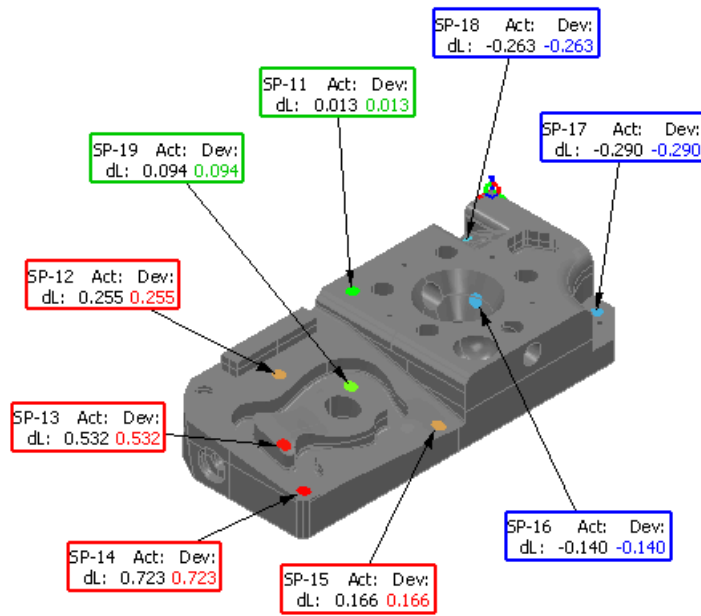
Label colours

PowerINSPECT Play Only uses colour to indicate the measurement status of points displayed in the CAD view.

Surface inspection points

The colour of the confetti (see page 33) or in-place pin (see page 32) indicates the measurement status of points:

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



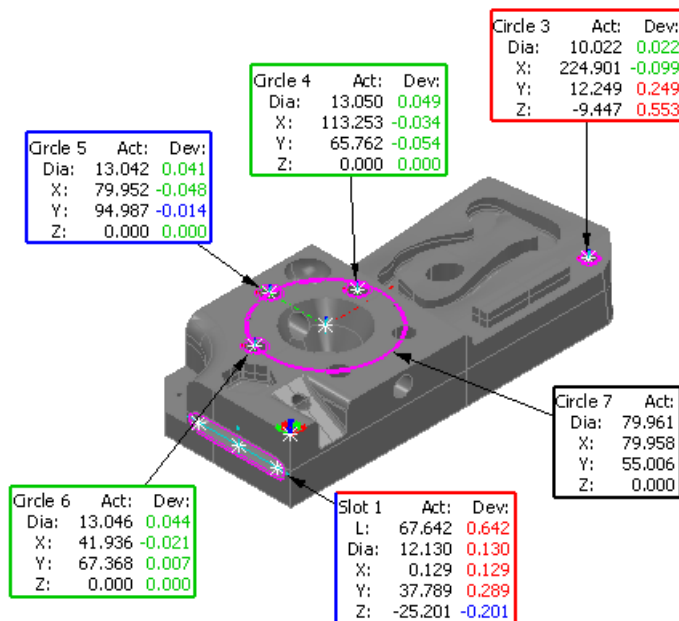
*You can change the tolerance colours using the **Colours** pages of the **Options** dialog.*

Geometric items

The colours of the text within labels indicate the measurement status of each item. By default:

- Green indicates the measurement is within tolerance.
- Red indicates the measurement is above the tolerance band.
- Blue indicates the measurement is below the tolerance band.
- Gold indicates the item is unmeasured.
- Black indicates the item has not been measured (for example, constructed items), or the measurements have not been compared with the item's nominals.

Colours are also used for the borders of item labels, and, when some of an item's measurements are above tolerance and some are below tolerance, the label border is displayed in multiple colours. For example:



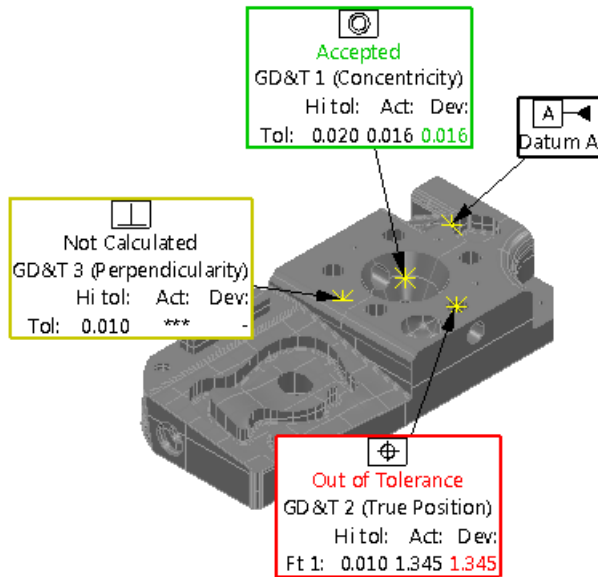
*PowerINSPECT Play Only displays tolerance indicators only when **Use nominals** is selected in the item's dialog. When this check box is selected, the item's **Output in report** settings determine which tolerance indicator is displayed. If any out-of-tolerance measurements are included in the report, the item is shown with an out-of-tolerance icon and label border. If no out-of-tolerance measurements are included, the item is shown with in-tolerance indicators, even when some measurements are out of tolerance.*

GD&T items

The colours of the borders of labels indicate the measurement status of each item:

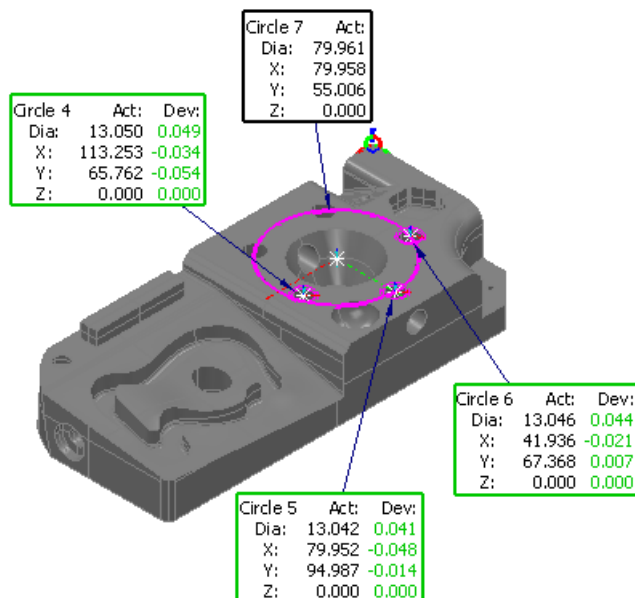
- Green indicates the measurement is in-tolerance.
- Red indicates the measurement is out-of-tolerance.
- Gold indicates the measurement has not been calculated.

- Black indicates the item is unmeasured, for example, a GD&T Datum item.



Show auto labels horizontally

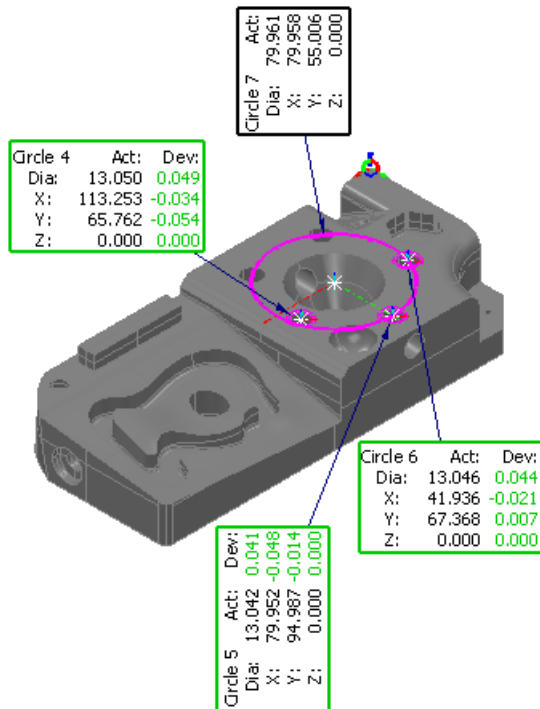
Use the **Show auto labels horizontally** button to display all label text horizontally. For example:



Show auto labels standard



Use the **Show auto labels standard** button to display the label text vertically or horizontally. For example:

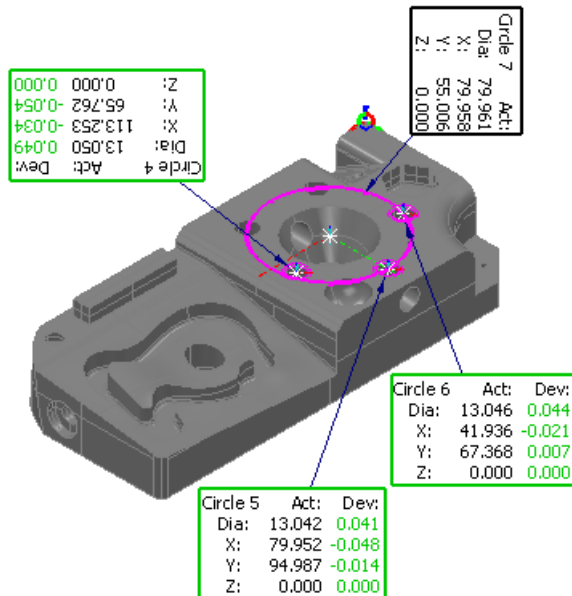


The orientation of each label depends on its position in the CAD view.

Show manual labels



Use the **Show manual labels** button to size and position each label manually. For example:

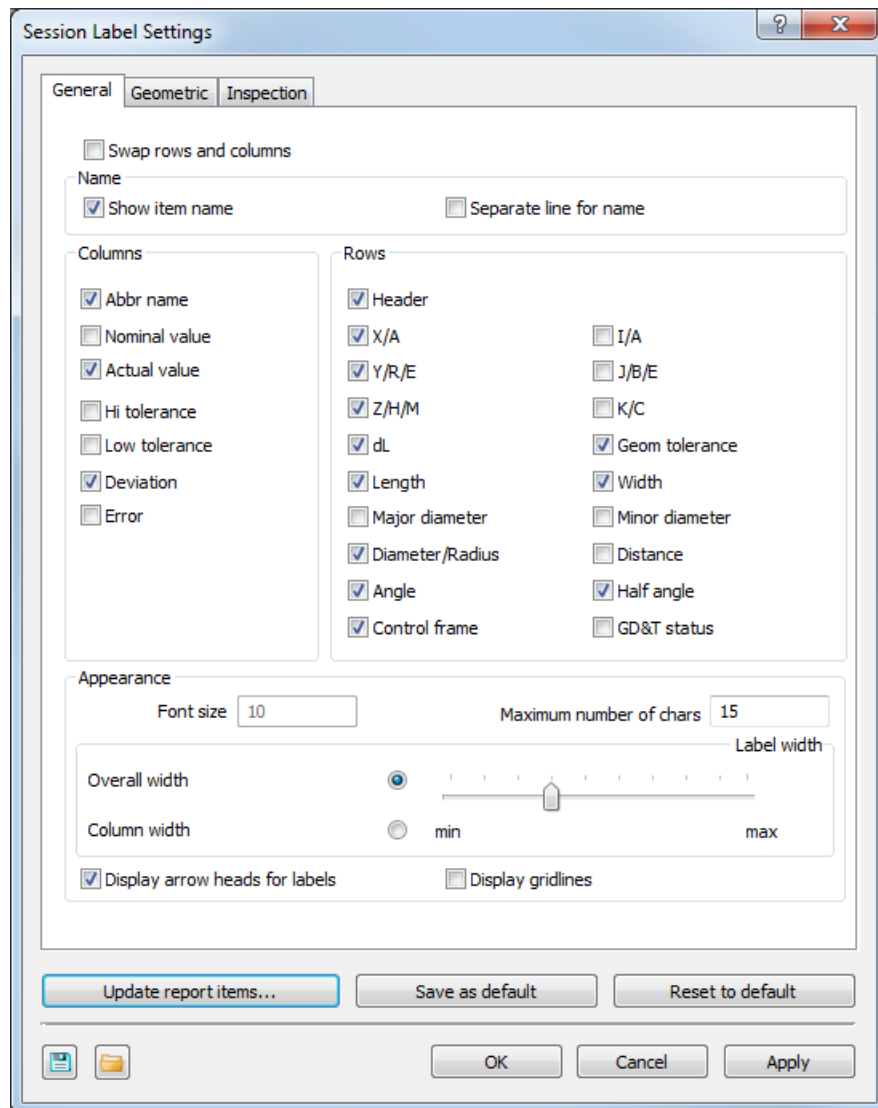


To:

- Move a label, left-click and hold the label box, drag the label to its new position, then release the mouse button
- Rotate a label through 90 degrees, right-click the label and select **Rotate label clockwise** or **Rotate label counter-clockwise** from the context menu.
- Hide a label, right-click the label and select **Label visible** from the context menu.
- Size the labels, right-click a label and select **Label settings** (see page 56) from the context menu.

Session Label Settings

Use the **Session Label Settings** dialog to control the appearance of CAD view labels (see page 50), and to specify which measurements are displayed by default.



To specify the label settings:

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



button. Alternatively, right-click a label in the CAD view and select **Label Settings** in the context menu. The **Session Label Settings** dialog is displayed.

- 2 Select the **General** tab.
- 3 Select the settings you want to use:

Swap rows and columns

Select this check box to display the label column as rows and the label rows as columns.

Name

Use the **Name** settings to control how the names of items are displayed in labels. Select:

- **Show item name** to display name of the item in each label.
- **Separate line for name** to display the name of the item on a separate line at the top of each label.

Columns

Use the **Column** settings to control which values are displayed in labels. Select:

- **Abbr name** to display the abbreviated name of each measurement.
- **Nominal value** to display the nominal value of each measurement.
- **Actual value** to display the actual value of each measurement.
- **Hi tolerance** to display the high tolerance settings for each item.
- **Low tolerance** to display the low tolerance settings for each item.
- **Deviation** to display the difference between nominal and measured values.
- **Error** to display the amount by which actual values exceed the specified tolerances.


Rows

Use the **Rows** settings to specify which measurements are displayed in each item's label. Measurements are displayed only when they are recorded for the individual items. For example, a **Length** measurement is not recorded for spheres, so a length value is never shown in the labels of sphere items. Select:

- **Header** to display a title at the top of the columns in each the label.
- **X/A** to display the **X** or **A** measurement.
- **Y/R/E** to display the **Y**, **R**, or **E** measurement.
- **Z/H/M** to display the **Z**, **H**, or **M** measurement.
- **dL** to display the shortest distance between a probed point and the part surface.
- **Length** to display the **Length** measurement.
- **Major diameter** to display the **Major diameter** measurement.

- **Diameter/Radius** to display the **Diameter** or **Radius** measurement.
- **Angle** to display the **Angle** measurement.
- **Control frame** to display the summary of the tolerancing requirements for GD&T items.
- **I/A** to display the **I** or **A** measurement.
- **J/B/E** to display the **J**, **B**, or **E** measurement.
- **K/C** to display the **K** or **C** measurement.
- **Geom tolerance** to display the form tolerance measurement, such as **Flatness**, **Conicity** or **Sphericity**. If the **Control frame** check box is selected, PowerINSPECT Play Only summarizes the form tolerance requirements in a control frame at the top of the label.
- **Width** to display the **Width** measurement.
- **Minor diameter** to display the **Minor diameter** measurement.
- **Distance** to display the **Distance** measurement.
- **Half angle** to display the **Half angle** measurement.
- **GD&T status** to display the measurement status for GD&T items.



*To control the display of measurements for individual items, use the **Output in report**  buttons in the item dialog. You can also show or hide the contents of groups using the **Output in report** check box in the dialog of the parent group.*

Appearance

Use the **Appearance** settings to control the sizing of labels and label text, and to control how connection lines are displayed.

- **Font size** — Enter a value to specify the display size for characters in labels.



*You can change the **Font size** only when **Show manual labels** is selected in the **Show/Hide labels** toolbar (see page 50).*

- **Maximum number of chars** — Enter a value to specify the maximum number of characters that can be used to display each field in a label.
- **Overall width** — Select the option and move the slider to specify the maximum width of the labels as a percentage of the width of the CAD view.

- **Column width** — Select the option and move the slider to specify the maximum width of each column within each label.





*You can change the width settings only when **Show manual labels** is not selected in the **Show/Hide labels** toolbar.*

- **Display arrow heads for labels** — Select this check box to display arrow heads on the lines connecting labels to features on the model.
- **Display gridlines** — Select this check box to display grid lines within each label.



*You can change the grid line colour using the **Colours > General** page in the **Options** dialog.*

- 4 To use different settings for the labels of geometric items:
 - a Select the **Geometric** tab.
 - b Select the **Override general settings** check box.
 - c Select the settings for geometric items.
- 5 To use different settings for the labels of inspection group items:
 - a Select the **Inspection** tab.
 - b Select the **Override general settings** check box.
 - c Select the settings for inspection point labels.
- 6 If you want to:
 - apply your changes to images already shown in the **Report** tab, click **Update report items**.
 - use your **General** selections as the default settings for new inspection documents, click **Save as Default**.
 - return the options to the current default settings, click **Reset to Default**.
 - save the settings to a file, click . You then import the settings into other installations of PowerINSPECT Play Only.
 - load the label settings from a file, click .
- 7 Click **OK** to save your changes and close the dialog.



Some settings may apply to features that are not available in your version of PowerINSPECT Play Only. Changing these settings has no effect.